EasyEDA Pro Tutorial

v2.1.X

EasyEDA Online Editor： https://pro.easyeda.com/editor

EasyEDA Desktop Client： https://easyeda.com/download

Remark

- This document will be updated as the new features of the editor are updated.
- The latest revison please refer at EasyEDA Tutorials.pdf

FAQ

Editor

THIS TOTURIAL IS TRANSLATING BY GOOGLE, WILL CORRECT IT CONTINUOUS

EasyEDA Pro Vs EasyEA Std

EasyEDA is currently available in two editions, EasyEDA Professional and EasyEDA Standard. The Standard Edition is geared towards students and education, with simpler features and usage; the Professional Edition is geared towards enterprises and teams, with more powerful features and higher constraints.

For specific functional differences, please see： [EasyEDA Edition Comparison]

Tutorials Download/Video Tutorials

PDF Tutorials Download： https://easyeda.com/page/download

Preparation

No graphics card or low performance graphics card when running the editor

When the computer does not support WebGL, this message will pop up because the PCB needs to use graphics acceleration (WebGL engine), otherwise the PCB will not run properly. Usually computers and graphics cards from 2015 onwards are supported.

Please try the following.
1, please use the latest Google or Firefox browser, in the browser settings to enable hardware acceleration.
2, update the graphics card driver to the latest version.
3、If still prompted, please add the graphics display of the browser to the graphics acceleration list in the graphics setting software.
4、If the above operation is invalid, you need to update your device, your computer may not support WebGL.
5、Or please use the standard edition of EasyEDA.

https://easyeda.com/editor

Recommended computer configuration: post-2017 computer and graphics card, CPU i5, 8GB of RAM, 1080P display.

The current browser environment does not support running EasyEDA Pro

Because EasyEDA Pro uses indexedDB technology, if your browser does not support it, you will get this message.

1、Please use the latest Google or Firefox browser
2、If you are using Firefox, please do not open the editor in incognito mode (anonymous mode)
3、If you can do it before, please restart the browser or client, if the client restart is invalid, please download the latest version to install.

Library

How to create a library file

1、In the editor home page, click New Device.
2, device binding symbol and footprint, if there is no symbol and footprint, you need to create a new one first.
3、After the completion of New, in the bottom panel (shortcut key S), or in the device Place dialog box (shortcut key SHIFT+F) to find the device list, select the device to place in the schematic.

How EasyEDA's component libraries are differentiated

EasyEDA Pro's libraries are divided into: device library, symbol library, footprint library, and reuse block library.
Device = Symbol + footprint + 3D model + Properties. It is possible to reuse symbols, footprints, 3D libraries, etc. very well. New devices need to be associated with symbols footprints, etc.

Only support placing device in schematic or PCB, when the device is placed in schematic, it will take symbol with it and instantiated as component at this time. The device on the canvas is a component.

Schematic

How to modify the size of the drawing

In the properties panel on the right, click "Drawing Symbols" and select other sizes of drawing symbols.

Specific reference.

Project Drawing Setting：https://prodocs.easyeda.com/en/schematic/design-project-drawing-setting/index.html

Drawing Symbol Creation：https://prodocs.easyeda.com/en/symbol/file-new-symbol-drawing-symbol/index.html

How come my updated library is still old when I place it again?

EasyEDA Pro Professional supports the concept of a project library. The placed library will automatically enter the project library as a template, and the next time you place a library from the same source will give priority to the template inside the project library, avoiding the update of individual libraries to affect the existing design of the project.

The project library can be edited, the entrance is: bottom panel - Component Library - Project - Device/Symbol/Footprint.

You can also update the project library directly, the entrance is: Top Menu - Design - Update Symbols/Devices

Please refer：https://prodocs.easyeda.com/en/schematic/side-bottom-panel-project-library/index.html

How to place net labels

EasyEDA Pro Professional no longer supports separate net labels like the standard version, the Professional version's Place net labels (shortcut N) actually automatically sets the name for the wire. Please draw the wire and then set a name attribute to the wire, the wire name is the network name. You can also create a network by placing a net labels one the wire.

How to add a horizontal line above a network name

Enter a wavy number at the top of the wire's network name, e.g. `~VSS`.

If you need a network name with and without an upper horizontal line, enter a wave again, e.g. `~VSS~/GND`, then VSS has an upper horizontal line above it and no slash after it.

The same is true for symbolic pin names.

How to create a hierarchy schematic

In the top menu - Place - Reuse Blocks, enter the name of the block, when finished, right click on the selected block and expand it, at this time you can place ports, device symbols, etc. Then create the pin information of the Reuse block symbols, etc. by using the: Top menu - Design - Update Block Symbols.
After creation, you can switch to the project library in the bottom panel (shortcut S to open) and switch to the Reuse blocks for Place in the schematic.

Multi-level nested hierarchical diagrams can be supported.

How to batch rotate the circuit

When the batch selection circuit has included wires, the whole rotation is not allowed. Press the space bar to rotate when the wires are not included.

How to create multiple subprojects in a project

In the top menu - New - Board, a new board, schematic, PCB will be created automatically, at which point multiple board designs can be supported.

PCB

Cannot open PCB properly

1, please ensure that the computer has a graphics card, and not very old graphics card. Because opening the PCB needs the support of the graphics card.
2, re-open the editor to try or log out (choose to remove the cache), if you still can not open, there may be data abnormalities, please contact technical support.
3, if the client, please use Google Chrome and start hardware acceleration. Currently found that some Win7 computer installation client can not properly open PCB, the browser is normal.

How to set design rules

In the top menu - Design - Design Rules

How to generate PCB order file

In the Top Menu - Export - PCB Fabrication File Gerber

How to modify the copper zone heat welding

Just modify the rules for copper zone in the design rules.

How to show hidden copper

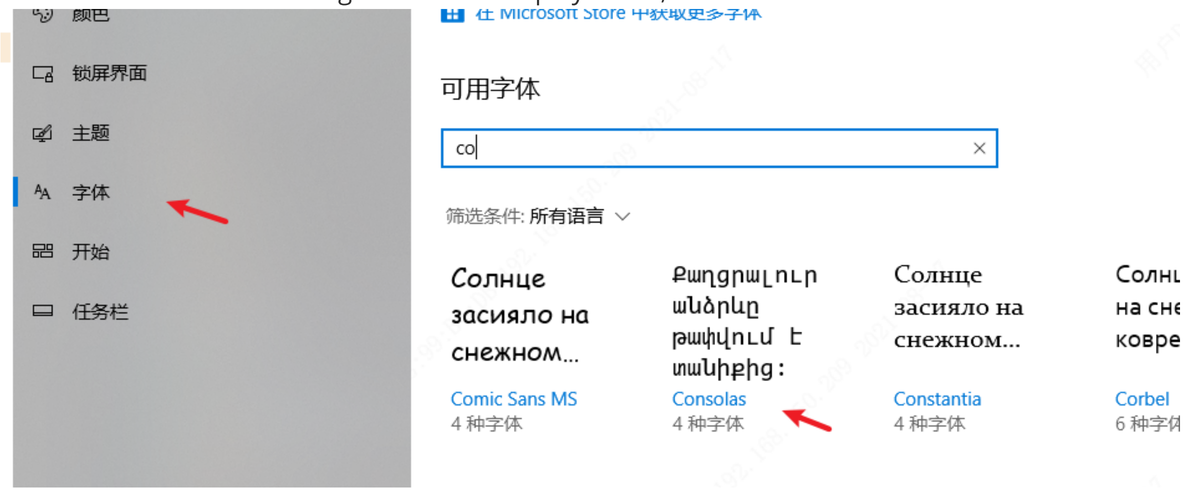
In the Filter tab on the right side, hide the "Copper Pavement Area (Fill)". Or click on the copper pavement and right-click to hide it. Shortcut SHIFT + M to show or hide all.

Add commonly used fonts

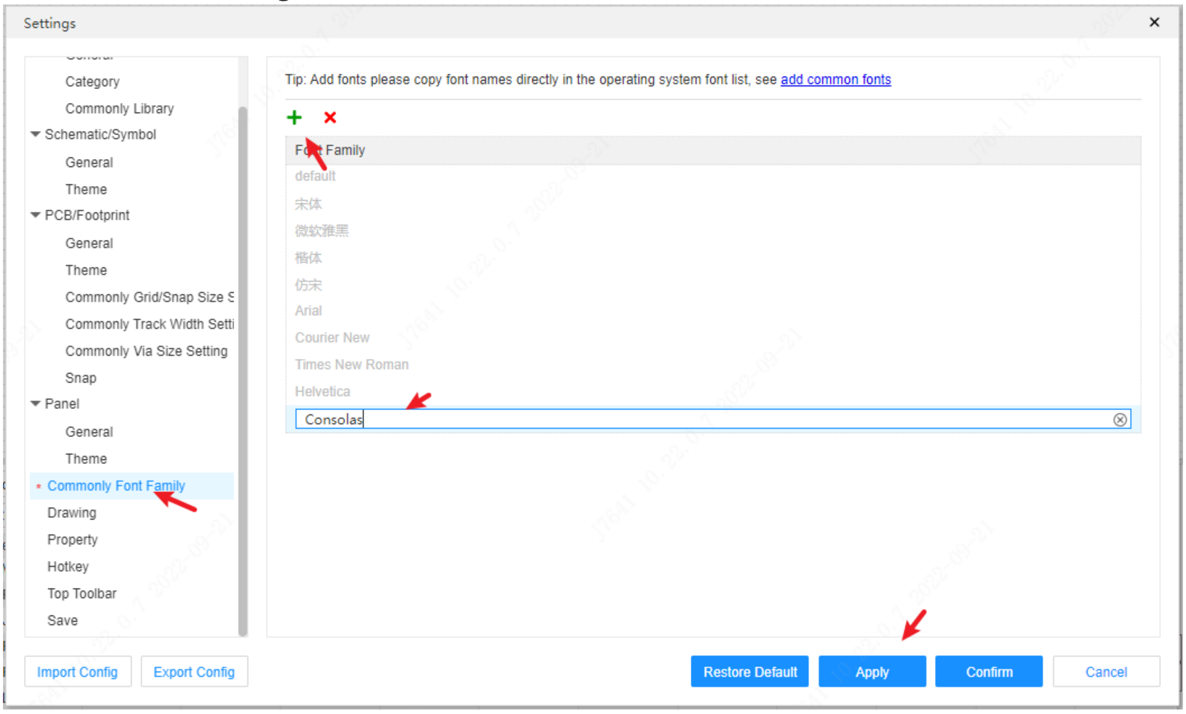
Support setting your own commonly used fonts. These fonts need to be installed on the local computer to be called by the editor, otherwise the editor will automatically use the default font provided by the browser for rendering.

To add custom fonts.
(1) Install the fonts you need on your local computer first, if you already have fonts, you can ignore this step.

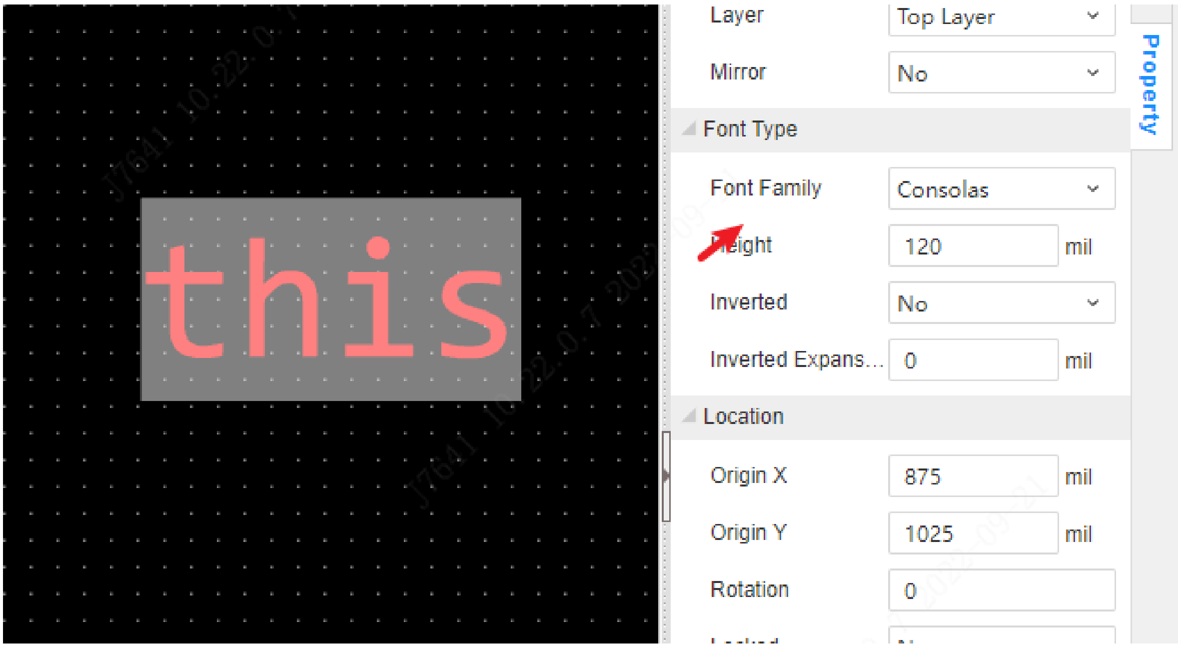
(2) Take Windows system as an example, find the font settings inside the system settings and get the font name. You need to get the exact display name, not the file name of the font file.



(3) Add a font according to the font name.



(4) You can see this font in the text font switch of the schematic or PCB, and you can switch the font at this time.



The added fonts can be used by schematics, panels, PCBs, etc.

Import

How to import files from EasyEDA Pro Standard Edition

1、Open the Professional Edition, click the "Migrate Standard Edition" icon on the start page to import the files of the Standard Edition.

2、When the project is not opened in the Professional Edition, first in EasyEDA Pro Standard Edition - Project List - Right click - Download Project, then in EasyEDA Pro Professional Edition - Import EasyEDA Pro (Standard Edition), select ZIP archive to import.

If it is a library file, then in EasyEDA Pro Standard Edition first export EasyEDA Pro format, and then import can be. When importing the schematic library, you can choose whether to generate devices or symbols, or both when importing.

Note: If there is a PCB, please make sure to import the schematic and PCB together after compression.

3, in the professional version of the project has been opened, you can choose to import EasyEDA Pro standard version of a single JSON file, the import will be inserted in the current open project.

How to import AltiumDesigner or Protel or Kicad files

See: [Import Altum file]

For other file imports, see the Import and Export section.

Desktop Client

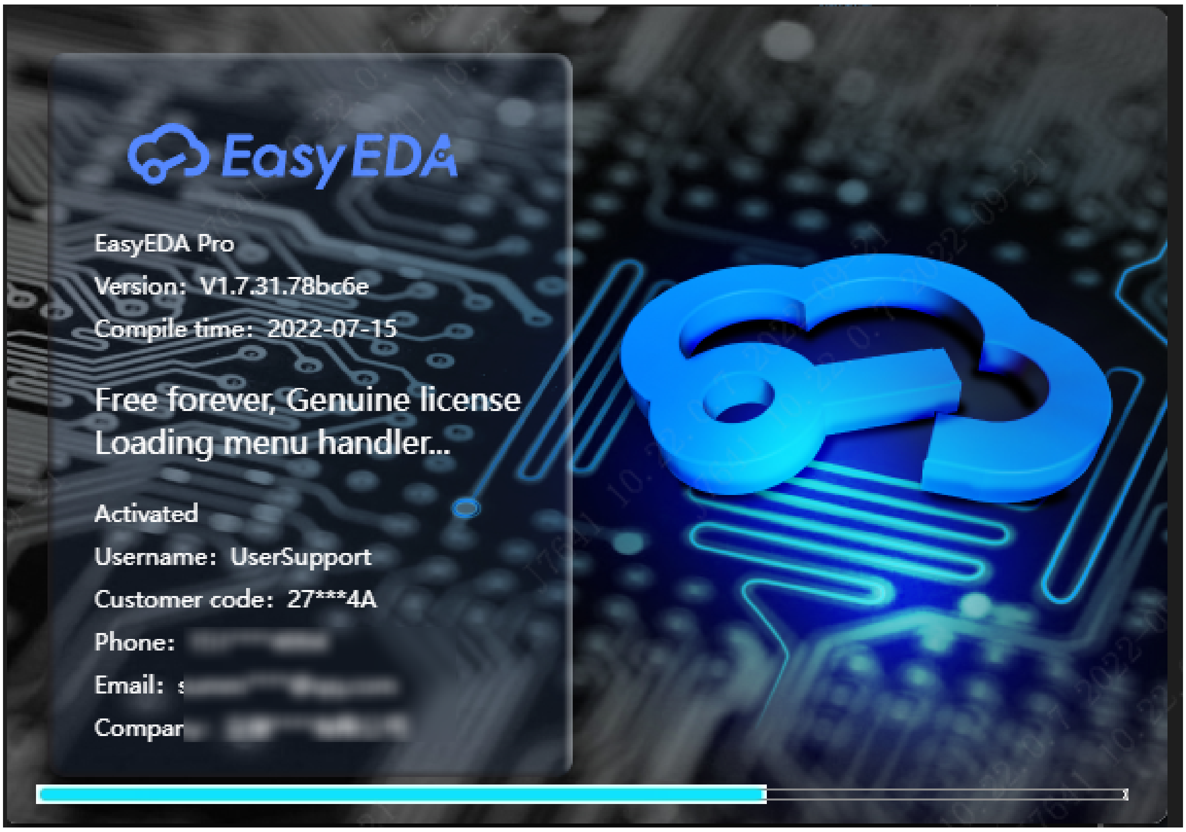
Introduction

Introduction

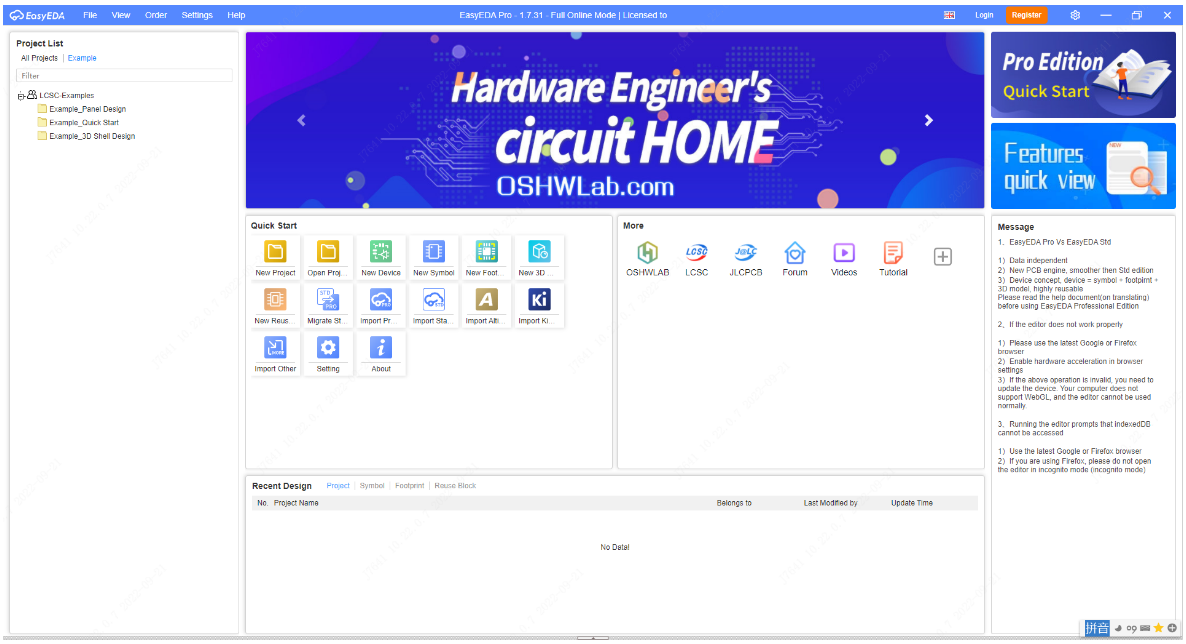
EasyEDA Pro desktop client support three modes:

- Full online mode: login is required, libraries and projects are stored in the cloud, collaboration is supported, and cloud projects are automatically backed up to the local;
- Half-offline mode: no login required, both libraries and projects exist locally, collaboration is not supported, and cloud system libraries are supported. This mode is recommended.
- Full offline mode: no login is required, both libraries and projects exist locally, collaboration is not supported, and cloud system libraries are not supported. Built-in 1w+ commonly used system libraries.

Start interface:



Main interface:



About the menu and top bar:

Client installation instructions

1. If the beta version has not been installed before, it can be installed directly;
2. If the beta version has been installed before, it is recommended to uninstall the old version before installing it to avoid the effect of cache;
3. Please ensure that your computer has a graphics card, a discrete graphics card is better;
4. Only 64-bit is supported, and XP system is not supported.

Download Link

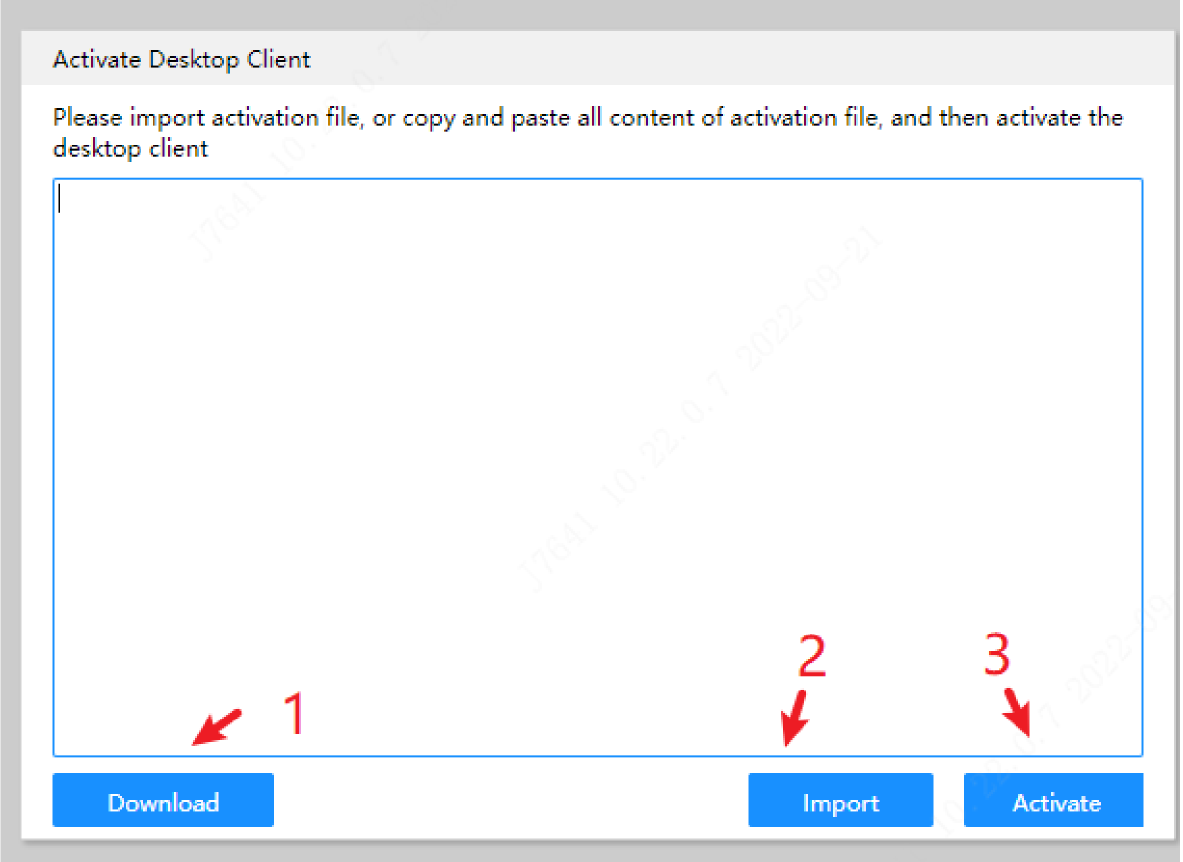
Only support x86_AMD64 bit PC

https://easyeda.com/page/download

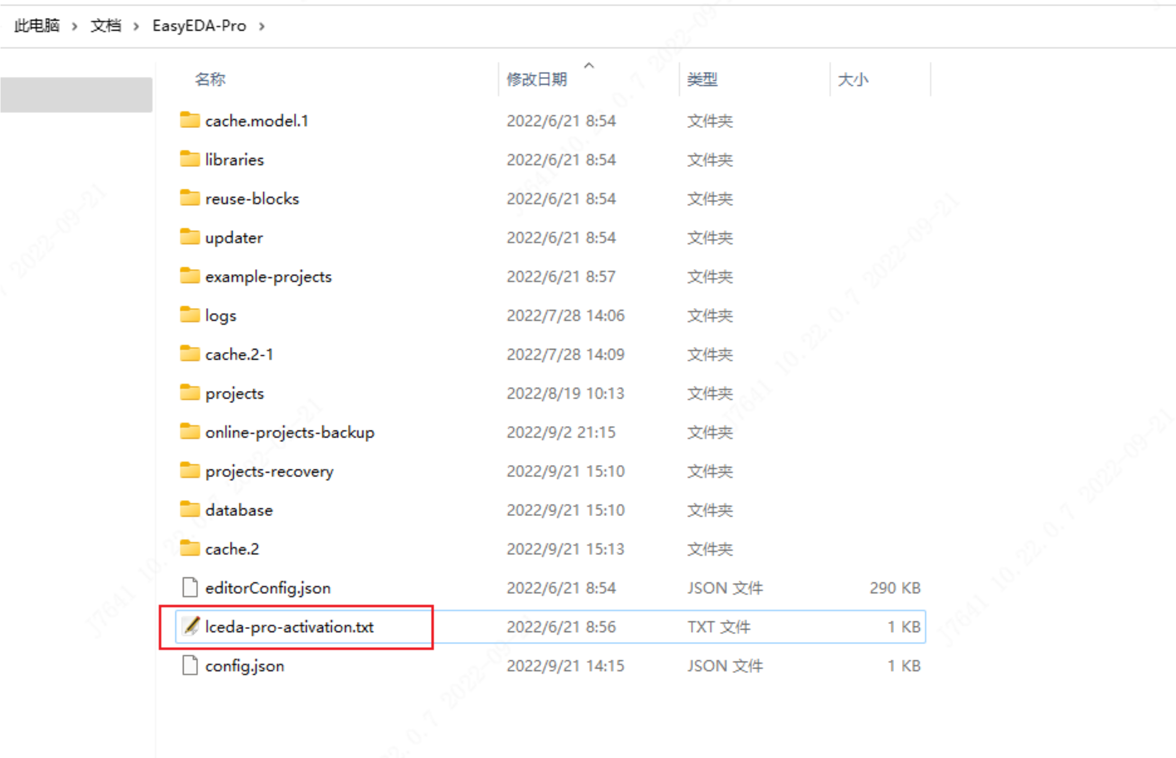
For "Windows version", due to system compatibility issues, the "windows version" may have a white screen on the startup interface after installation. It may not be loaded into the editor, or it will take a long time to load it. If you encounter a white screen on the loading interface, please turn off the firewall and try again, or contact Technical support solution. Email: support@easyeda.com
For "Linux version", please follow the instructions in the downloaded compressed footprint to install it.
For "Mac version", please follow the instructions in the downloaded compressed footprint to install it.

Load Activation File

Activation file free download address：https://easyeda.com/page/desktop-client-activation when you open the desktop client first time, it will open a activate dialog, you can download the activation file to acitvate.



After activation, the activation file will be stored in the data storage directory: `This computer/Documents/LCEDA-Pro`, you can find it by viewing the default project storage path set. Manual deletion removes the active state.



Notice:

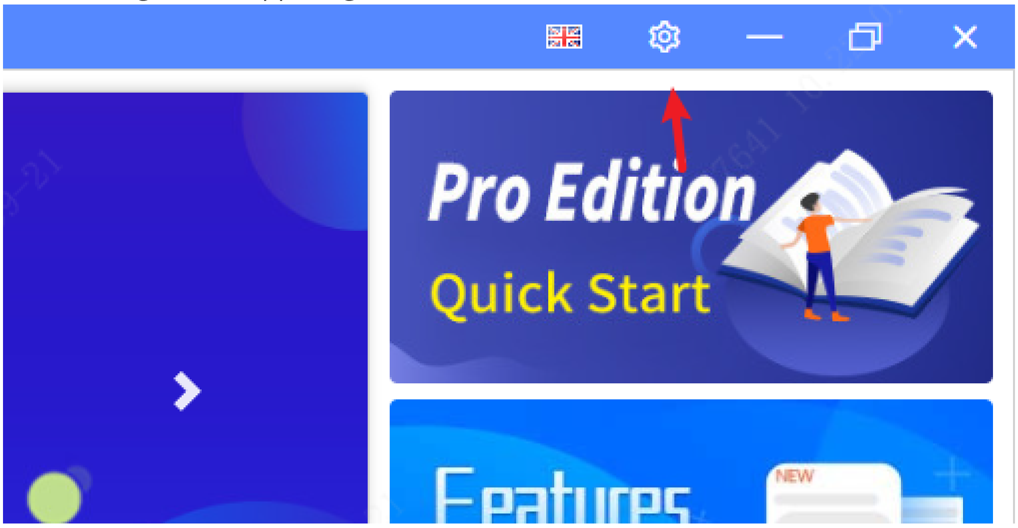
1. The activation file is free to download, no need to crack, the genuine license can be downloaded after registration;

2. The activation file contains your account information, please do not disclose it to the public and keep it properly;
3. The activation file cannot be modified, and the import cannot be activated after modification.

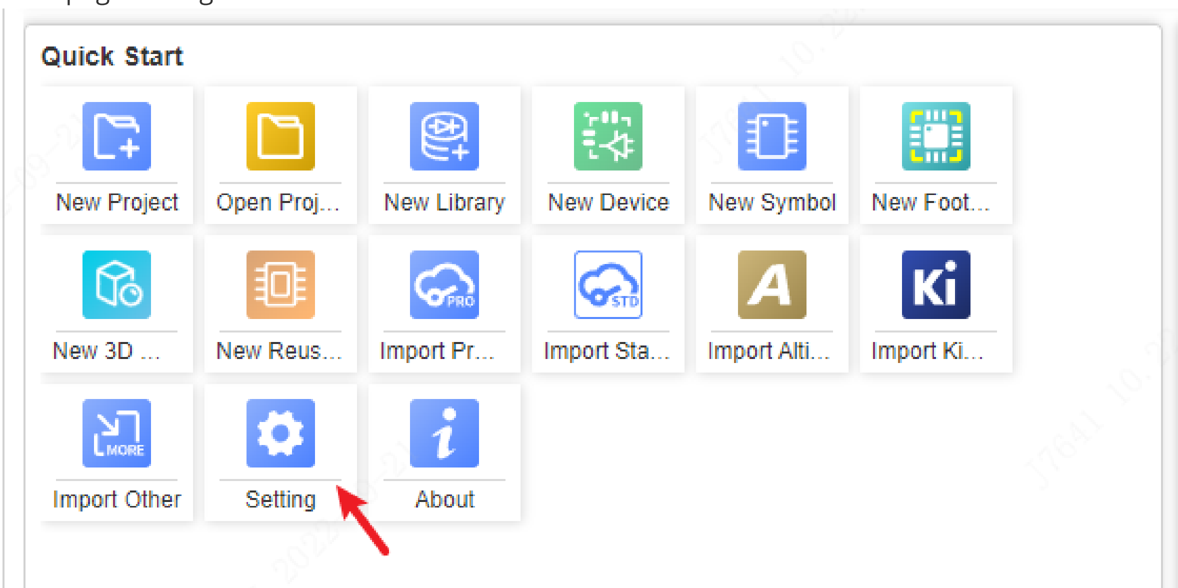
Set Running Mode

After importing the activation file, you can open the client interface and click the "gear" icon in the upper right corner or the "settings" icon on the start page to set the operating mode.

Gear settings in the upper right corner:



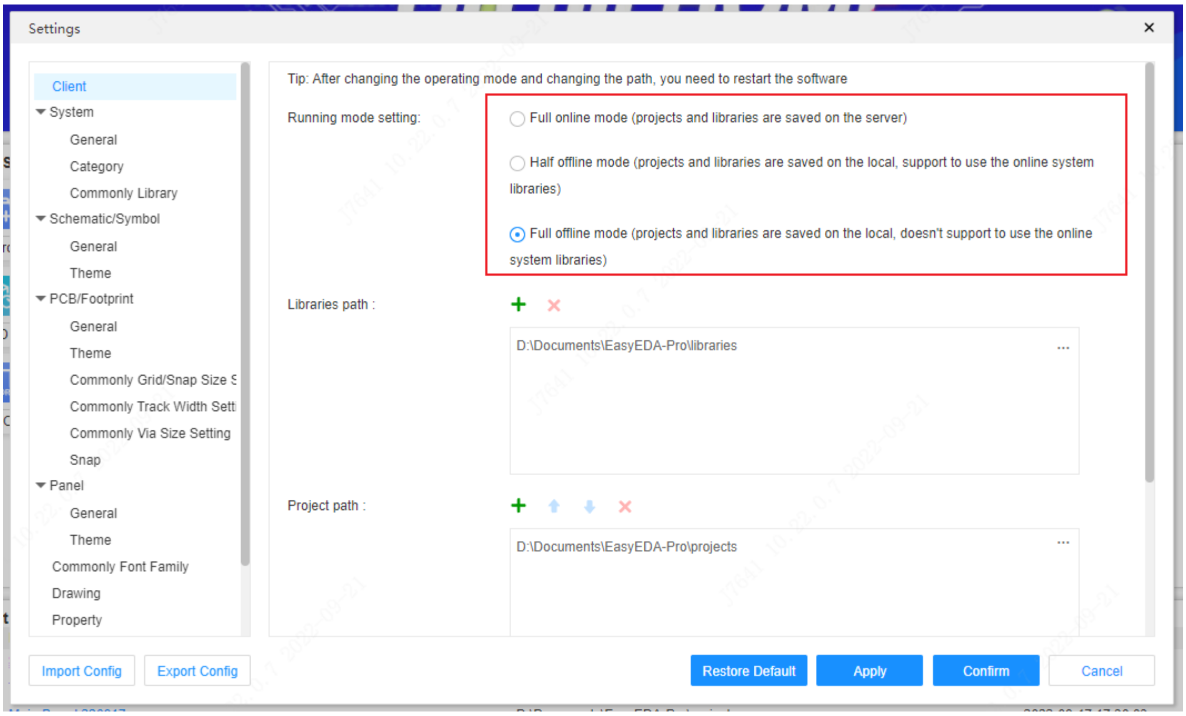
Start page settings button:



Operation mode description:

- Full online mode:
- Need to use the Internet, need to log in, support team collaboration.
  - All data is stored in the cloud server.
  - Supports automatic backup of online projects locally, and the editor will back up the project compressed footprint in this folder according to the set backup interval.
- Hafl-offline/Full offline mode:
- Semi-offline mode supports the use of online system libraries.
  - Personal libraries and projects are stored locally.

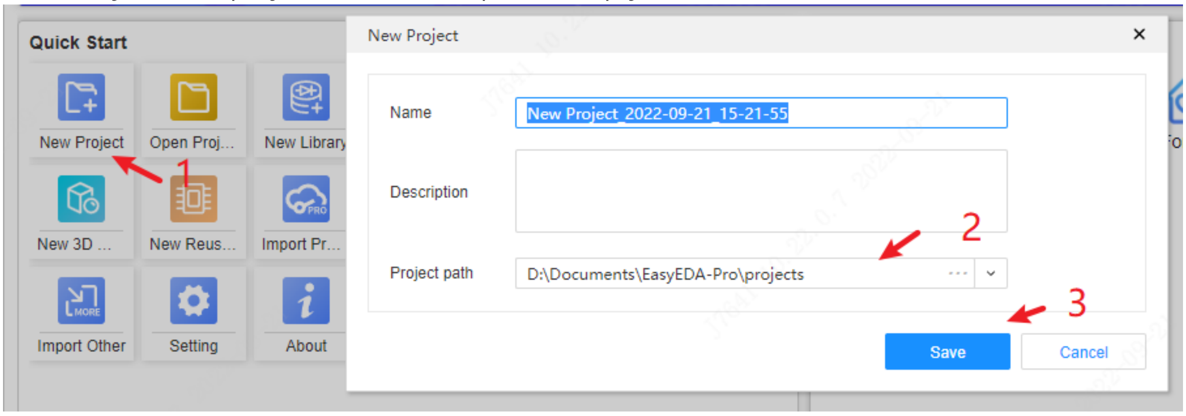
- Support setting library path, support adding multiple library paths.
- Supports setting the default storage path for new projects.



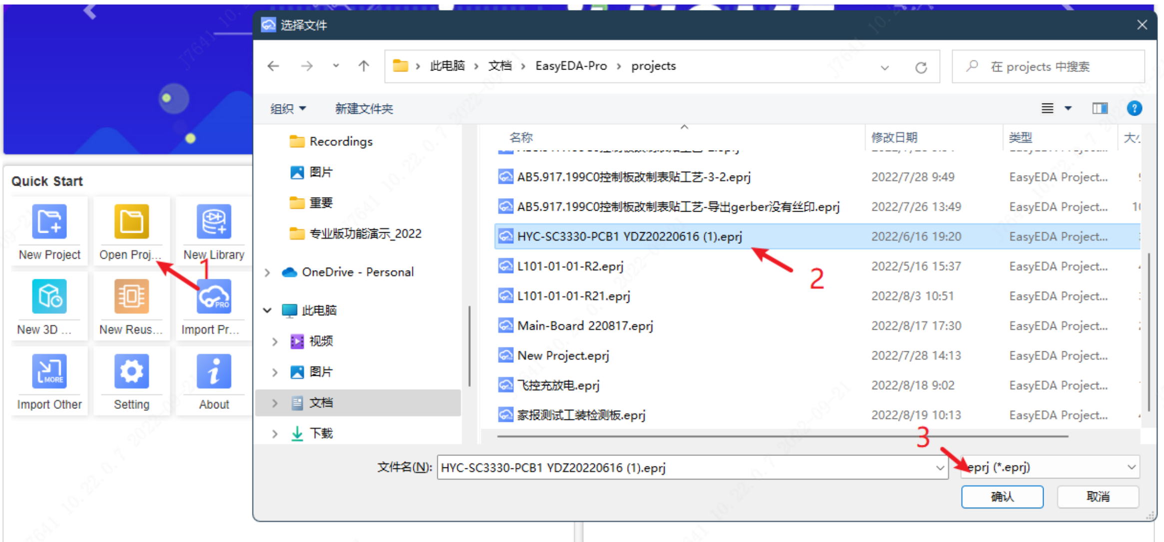
Create or Open a Project

If you use half-offline/full offline mode, you can create the local project files.

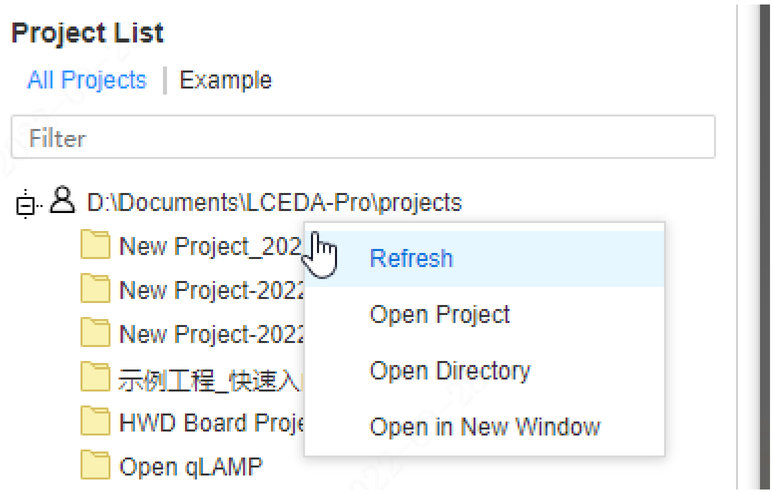
The newly created project file is an independent eprj file



When opening, you can also select the **eprj** file to open, support directly double-clicking the `eprj` file to open the project directly at local.



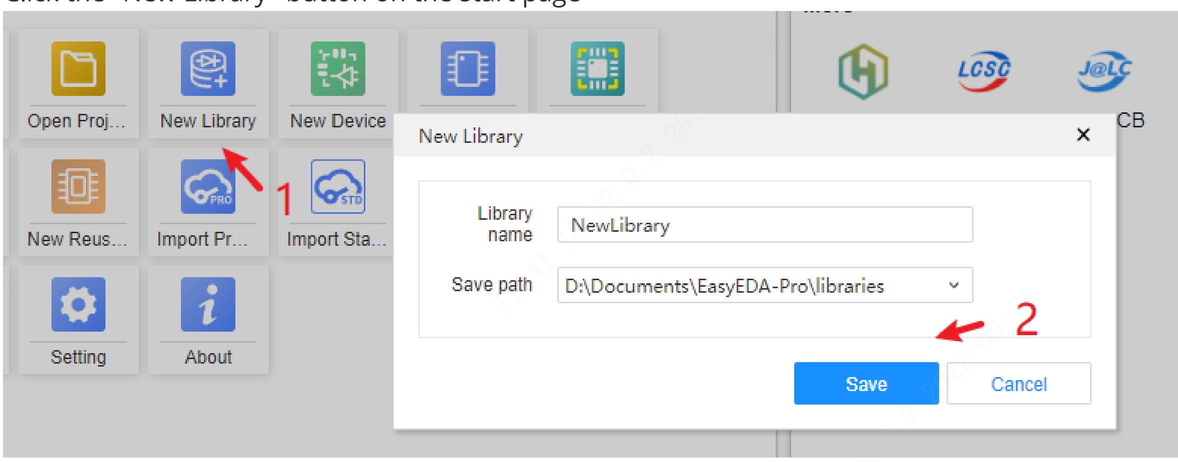
You can also open the project in the project list on the left, and opening the client will automatically load all the projects in the project path, or you can right-click to refresh the project list



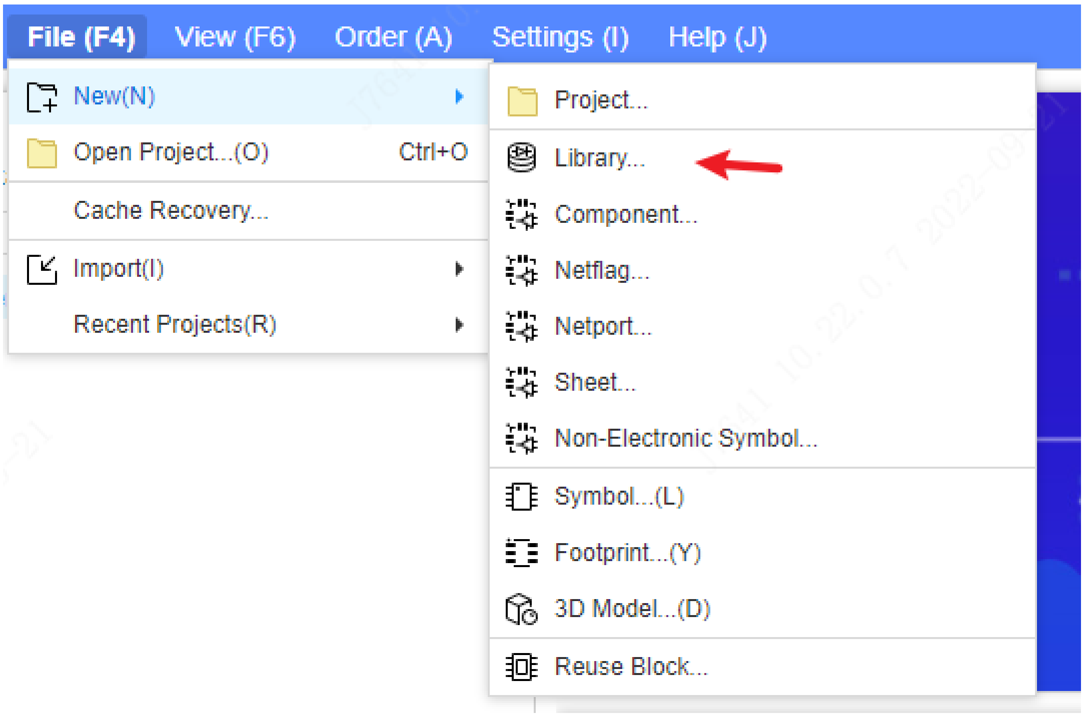
This eprj file is only used by the client. If you need to export the project and import it to the online version, please go to the file menu, save as local, and then import the project compressed footprint to the online version.

Create a Library

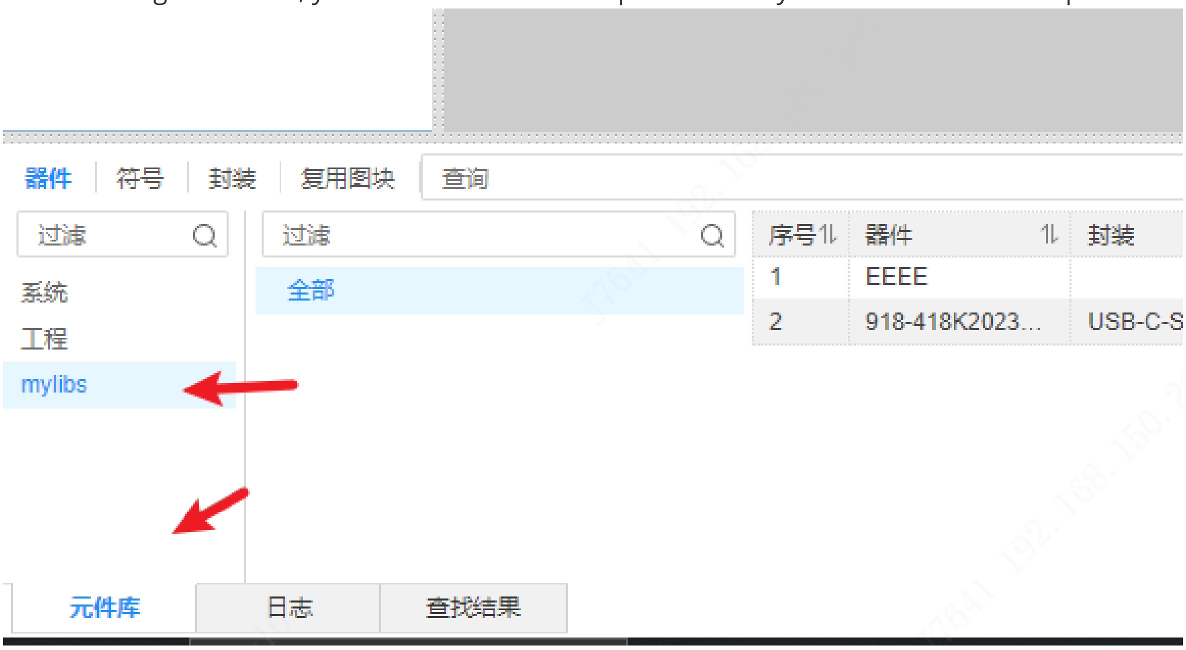
Click the "New Library" button on the start page



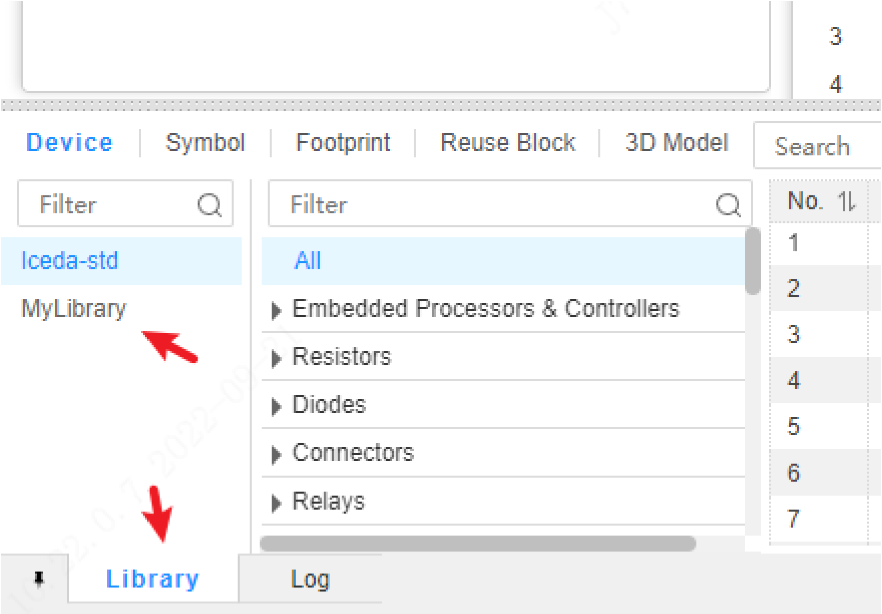
Or "after opening the document", click: `File > New > Library` in the Top Menu



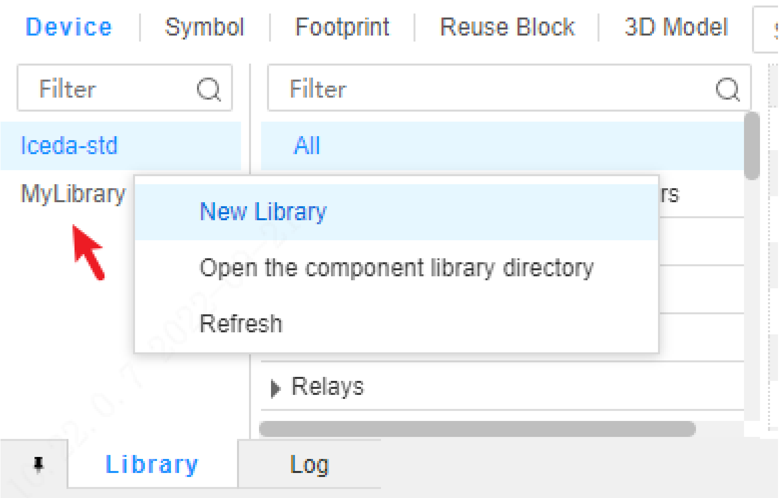
After creating a new one, you can find it in the component library list on the bottom left panel



When creating a new device, symbol, footprint, etc., you can choose to save it in your personal library.



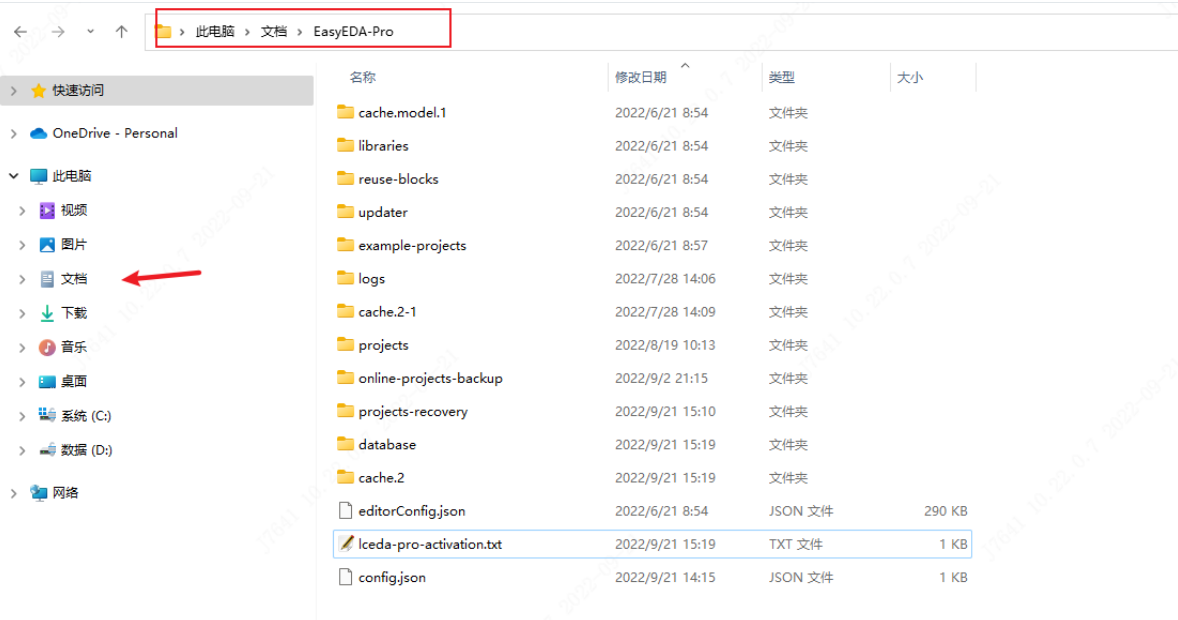
right-click the library name, you can open the path



Library Search : Search support fields for local component library files are: title, partial attributes (footprint, supplier footprint, value, supplier part, part number, part code), description. When you build a device library with your own company's part number or part code, you can search by part number or part code.

Data Directory Description

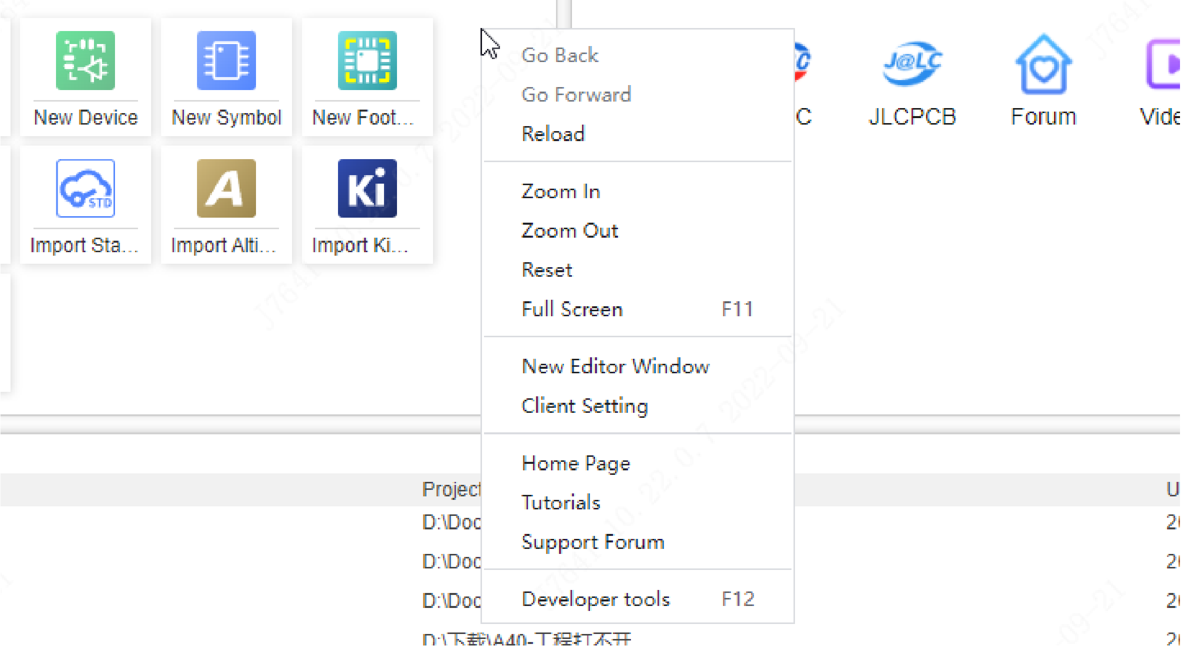
After the EasyEDA Pro client is installed, a data folder will be created by default in `This PC\Documents\EasyEDA-Pro`. Please do not delete or modify the files in this directory to avoid errors.



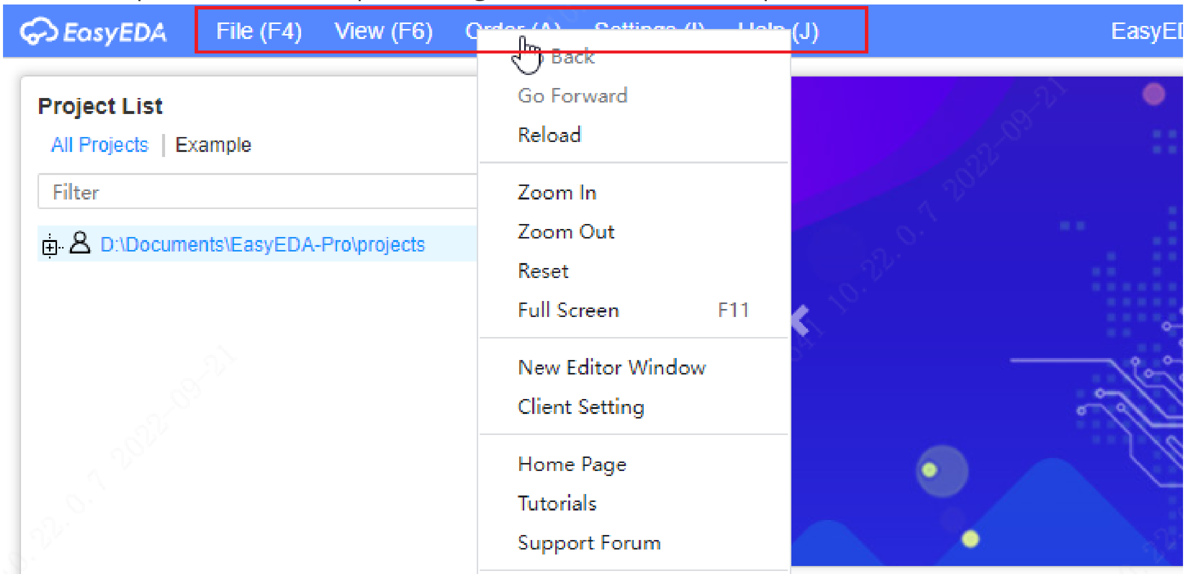
Folder Name	Type	Remarks/Description
cache	data cache directory
cache.model	Data cache directory
database	database directory
example-projects	Example project directory
libraries	Default library file directory	Personally created library file storage directory. Note: Starting from v1.9, the system library file lceda-std.elib is no longer placed in this directory.
logs	Running log directory	Store the running logs of the client. Note: Starting from the v1.9 client, this configuration is no longer used, and has been changed to create a trace or error or debug folder directly under the log folder. When the client detects that there are folders such as trace, it will automatically generate a corresponding log file. The folder needs to be deleted after use to avoid performance degradation of the client.
online-projects-backup	Online project backup directory	In the default full online mode, the online project automatically backs up the project zip file storage directory.
projects	Default project file storage directory	Corresponding menu: File - Version Switching (Backup and Restoration). There is a "project name_backup" folder in the directory, which is the storage directory for the backup zip file of the project.
projects-recovery	Project cache data storage directory	Corresponding menu: File - Cache Recovery, which is the cache zip archive of the project.
updater	The storage directory of the installation package automatically downloaded by the client
config.json	client configuration file
editorConfig.json	Client configuration file
lceda-pro-activation.txt	The activation file of the client	After manually deleting, the client becomes inactive

Right-Click Menu

You can operate the right-click menu on the start page



Or after "Open Document", open the right-click menu at the Top Menu text.



- Forward/Backward: When other web pages are opened on the client, you can use this forward/backward to return to the previous page or the next page
- Reload: Reload the current editor or page
- Zoom In/Zoom Out/Reset/Full Screen: Scale the client interface size.
- New editor window: the same behavior as clicking the logo in the upper left corner of the editor, you can open multiple editors. When opening multiple editors, you can open different projects respectively, and copy and paste documents and components; open them separately The schematic diagram and PCB of the project can be mutually positioned and updated by importing the PCB.
- Client Settings: Opens the client's settings dialog.
- Developer tools: After opening, you can check whether there is an error in the console console. If the editor runs incorrectly, you can send a screenshot of the console's error message (red content) to technical support, and press F12 again to close the developer tools.

Enable Hardware Acceleration

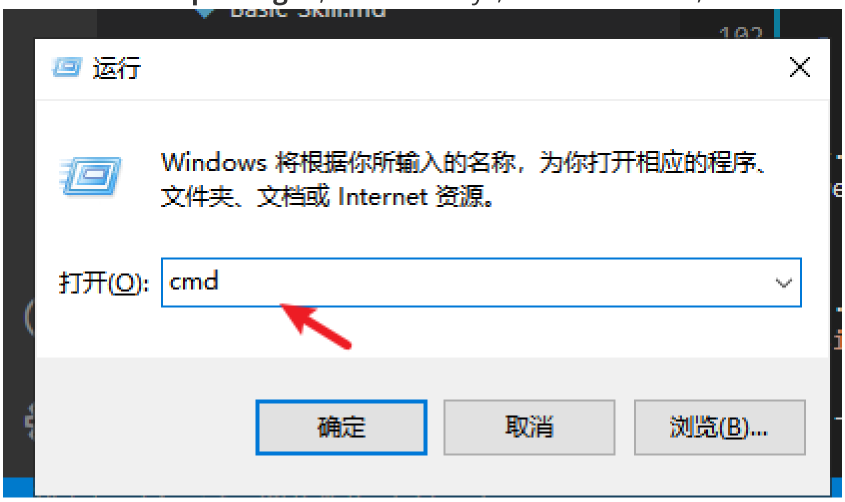
EasyEDA Pro PCB uses GPU rendering PCB. Please upgrade your PC GPU driver or Graphic card if you cannot use PCB smoothly.

Suggesting Win10 and above, at least 8GB ram.

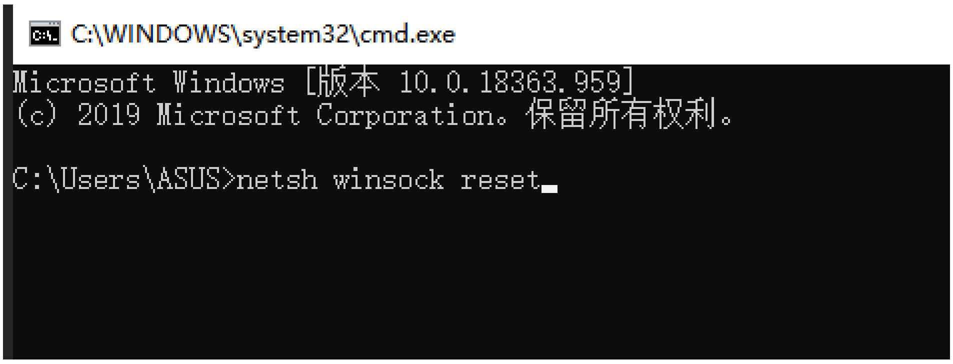
Known Issues

1. At present, it is found that some Win7 systems cannot run the client normally, and the interface cannot be opened normally. The reason is unknown, which belongs to the system compatibility problem. Sometimes it takes a long time to load the display page.
   If win7 opens the PCB black screen, you need to upgrade the graphics card driver. If the PCB still cannot be opened after upgrading the driver, it is recommended to upgrade to win10, or use Google Chrome to access.
2. The detection update in the settings has not been implemented yet. It is expected that the V1.7 version will support detection updates, and V1.8 supports downloading update footprints.
3. The maximized area offset in the upper right corner of the Linux version. The Mac version of the launch icon does not appear in the launcher.
4. At present, it is found that some Win7 systems cannot run the client normally, and the interface cannot be opened normally. The reason is unknown, which belongs to the system compatibility problem. Sometimes it takes a long time to load the display page.
5. If the windows system opens a white screen, you can try:

- 1. Turn off the firewall first, and then start the client. If it runs normally after shutdown, you need to add the client to the firewall whitelist.
  How to turn off the firewall: https://jingyan.baidu.com/article/cd4c2979d55c41756e6e60a1.html
- 2. If the firewall is turned off and the screen cannot be loaded, please try.
  - 1. Close the client first, and open the cmd command window of the system with **administrator privileges**， WIN + R keys, then enter cmd, enter



- 2. Enter in the opened cmd window: `netsh winsock reset`



- 3. Enter
- 4. Open the client again

If you find any bugs in the client during use, please give us feedback in time to fix it. Thank you for your continued support to Litron EDA!

Import Online Project

If you want to import the project from the online cloud to the offline mode of the client, you can:

1. Open the project in the browser, go to: File - Save as Local to get the project zip file
2. In the offline mode of the client or browser editor, start page: Import Professional Edition to import

Batch download of professional version projects is not supported, and batch import is not supported.

Export project to online

Same as above, after opening the project, go to: File - Save as Local to get the project zip file, and then import the project into the online editor.

Contact Us

Contact

PCB Order Problems:

- support@jlcpcb.com
- At present, the EasyEDA PCB fabrication and PCB Assembly service is transferred to JLCPCB. Although EasyEDA is part of the same company group, for any PCB order problems please contact with JLCPCB using the email address above.

Parts Order Problems:

- support@lcsc.com

- EasyEDA provides direct links to thousands of components which can be ordered directly from LCSC. For any problems with LCSC parts orders, please contact LCSC using the email address above.

All Other Inquiries About EasyEDA:

- Tutorials: EasyEDA tutorial
- User forum: EasyEDA forum
- support@easyeda.com

(Don't accept the advertising)

Business Cooperation About EasyEDA:

please contact

dillon@easyeda.com

(Don't accept the advertising)

Address:

- Mail address: Unit 21 on 28th Floor, Metropole Square, No.2 On Yiu Street, Shatin, New Territories, HONG KONG, China

Introduction

Introduction

EasyEDA Pro currently has two versions, EasyEDA Pro and EasyEDA Pro Standard.

EasyEDA Pro is a brand-new version that the EasyEDA Pro team has spent a year working on. It will be launched at the end of 2019. Please treat it as a new tool. Some functions and loose restrictions of the EasyEDA Pro standard version may not necessarily be provided.

The data of EasyEDA Pro does not communicate with the data of EasyEDA Pro Standard Edition. Please refer to the import chapter at the bottom to import the project file of the Standard Edition.

EasyEDA Pro

- The function is more powerful, the legacy problems of the standard version will be solved, and the new PCB drawing will be solved.
- The PCB is based on the WebGL engine, which can smoothly provide PCB design with tens of thousands of pads, and various constraints will be strengthened to provide more powerful rule management.

- Various constraints will also be strengthened, providing more powerful rule management, etc.
- More powerful device selection function, you do not need to switch back and forth between LCSC Mall and EasyEDA Pro editor frequently.
- Provides the concept of device, device = symbol + footprint + 3D model + attribute, only allows placing the device in the schematic/PCB canvas. Enhance library reuse.
- Supports hierarchical drawing, can support up to 500 schematic pages, and PCB supports 50,000 components and can still zoom, pan and route smoothly.
- Supports multiple single-board designs in one project.
- More powerful DXF import and export, more powerful PDF export.
- Built-in auto-routing function, the standard version requires an external auto-routing plug-in.

EasyEDA Pro is still under development, and some of the original functions of EasyEDA Pro Standard Edition will be gradually provided.

EasyEDA Standard Edition

- EasyEDA standard version project was established in 2011, the first overseas version EasyEDA was launched, and the domestic version EasyEDA was officially launched in 2017
- Some features and relaxed restrictions of EasyEDA Standard Edition may not be available in Professional Edition.
- The standard version of EasyEDA Std is based on SVG, and it will lag on some larger boards.
- The standard version of EasyEDA Std will mainly focus on fixing bugs, and the major functions will not be added.

data:

- The data of EasyEDA Pro is not interoperable with the data of EasyEDA Standard Edition. Data from the Standard Edition is provided to be migrated to the Professional Edition. There is a migration entry on the start page of the Professional Edition.
- Support importing Standard Edition to Professional Edition. Please see the Import section at the bottom for importing standard version project files.

Library file:

- EasyEDA Pro provides a device concept, device = symbol + footprint + 3D model + attribute parameters, to achieve high reuse, only allow devices to be placed in the schematic canvas.
- Standard version cannot reuse symbol library.
- The professional version does not support schematic modules and PCB modules.

Specific function comparison

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
Engineering	Public Sharing	V	V
	Teamwork	V	V
	Version switch	V	V
	Engineering Log		V
	Multi-Board Design		V
	Copy and paste across projects	V	V
Interface	Recent Projects/Libraries		V
	Show both project and documentation	V	
	Custom Shortcuts		V
	Example Project	V	V
	Customize Top Toolbar		V
	Bottom Panel		V
	Multi-window positioning	V	V
Import EDA files	Import Altium Designer	V	V
	Import EAGLE	V	V
	Import KiCad	V	V
	Import EasyEDA Standard	V	V
	Import EasyEDA Pro		V
	Import PADS		V
	Import Protel		V
	Import LTspice		V
Other	Recycle Bin	V	
	Document Recovery	V	

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Plugins/Scripts	V	
	Document History	V	
Schematic	Save As Module	V	
	Print	V	
	Drag	V	V
	Replace		V
	Find Similar Objects	V	V
	Assign Tag	V	V
	Tag Position Adjustment	V	V
	Array Object	V	V
	Multiple selection objects		V
	Delete Globally	V	V
	Empty Canvas/Delete All	V	V
	Lock/Unlock	V	
	Conductor	V	V
	Bus	V	V
	Bus Branch	V	V
	Web Tab	V	V
	Network Port	V	V
	Non-connection flag	V	V
	Reuse tiles		V
	Voltage Probe	V	
	Pin	V	
	Canvas origin	V	V
	Text	V	V
	Arc	V	V
	Image	V	V
	Lines	V	V
	Bezier curve	V	

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Form		V
	Arrow	V	
	Free drawing	V	
	Rectangle	V	V
	Polygon	V	
	Ellipse	V	
	Break up/combine into components	V	
	Edit component pin information	V	
	Project Library Template/Edit Project Library		V
	Rotate/Flip/Align/Distribute	V	V
	Element moves up and down	V	V
	Zoom	V	V
	Fit All	V	V
	Adapt to check/box selection		V
	Full Screen	V	V
	Toggle grid type	V	V
	Highlight Network	V	V
	Theme Settings	V	V
	Document Preview	V	
	Schematic to PCB/Update PCB	V	V
	Generate/Update Tile Symbols		V
	Design Rules/Check DRC		V
	Update Component from Component Library	V	V
	Reset Component Unique ID	V	V
	Engineering Drawing Template Settings		V
	Cross Select	V	V
	Layout pass	V	V

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Symbol Wizard	V	
	footprint Manager	V	V
	Circuit Simulation	V	
	Export BOM	V	V
	One-click component order	V	V
	Upload BOM to Litron ERP	V	V
	Canvas Display Component Properties		V
	Global Component Properties		V
	Primitive filtering		V
	Left network tree	V	V
	Left Component Tree	V	V
	Left Object Tree		V
	Common library on the left	V	V
	Custom common library		V
	Bottom Component Library		V
	Advanced Schematic Settings		V
	Auto Save		V
PCB	Repeat		V
	Advanced adsorption		V
	Array Object	V	V
	Add fillet/chamfer		V
	Lock/Unlock	V	V
	2D/3D Preview	V	V
	3D Shell Preview		V
	3D Shell Design		V
	Flip Board Subview		V
	Profile View		V
	Toggle unit	V	V

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Toggle grid size/style	V	V
	Polar Coordinate System		V
	Flying line show hide	V	V
	Network Highlight	V	V
	Object Highlight		V
	Import Update	V	V
	Select Import Updates		V
	Design Rules	V	V
	Advanced Design Rules		V
	Real-time DRC	V	V
	Cross Select	V	V
	Network Class Manager		V
	Differential Pair Manager		V
	Isometric Netgroup Manager		V
	Update Component from Component Library	V	V
	Break up/combine into components	V	
	Edit footprint pads directly	V	
	Right-click to edit the footprint		V
	Via	V	V
	Blind and buried vias		V
	Stitch Holes / Bulk Vias	V	V
	Pad	V	V
	Frame	V	V
	Copper area	V	V
	Fill Area	V	V
	Grooving Area	V	V
	Prohibited area		V
	Conductor	V	V

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Lines		V
	Test Points		V
	Text	V	V
	Image	V	V
	Dimensions	V	V
	Stacked Table		V
	Drilling table		V
	Set canvas origin	V	V
	Object Filtering	V	V
	Object Hide	V	V
	Layer Hidden	V	V
	Layer Lock		V
	One-way wiring	V	V
	Stretch Wire	V	V
	Differential Pair Routing	V	V
	Isometric adjustment	V	V
	Autorouting	V	V
	Export Import Autorouting File	V	
	Routing Block	V	V
	Routing Ignore	V	V
	Wiring Push		V
	Routing Wraparound	V	V
	Routing Corner	V	V
	Trace Width	V	V
	Remove loop	V	V
	Clear Routing	V	V
	Clear Component Connections		V
	Right-click menu during drawing		V
	Combining Objects		V

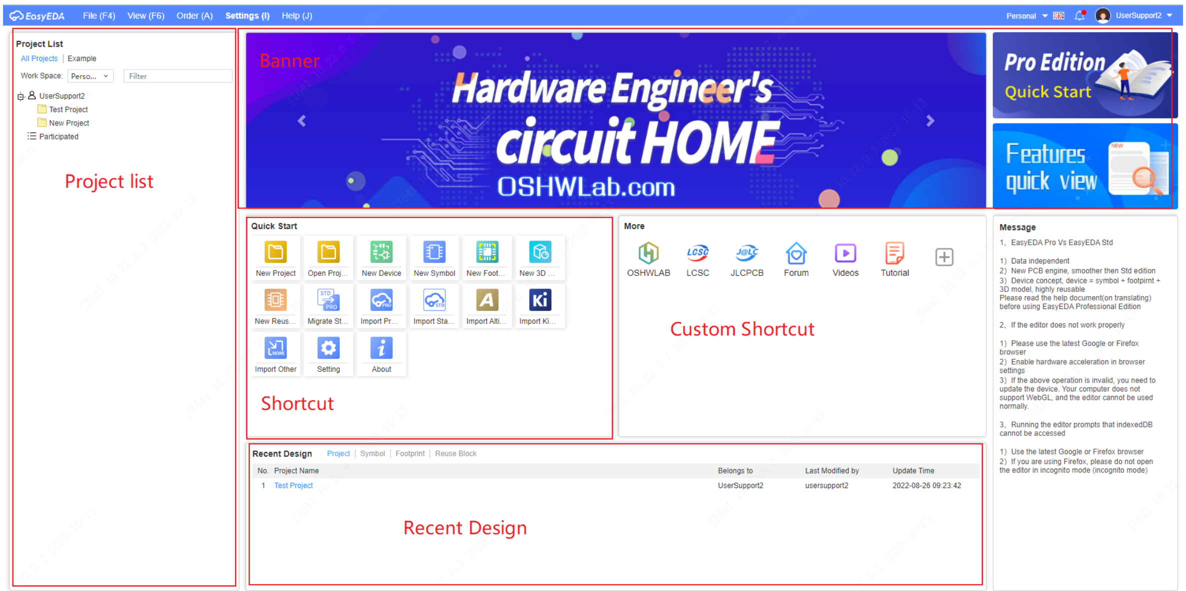
Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Property Location	V	V
	Swap positions		V
	Offset/Align/Distribute/Rotate/Flip	V	V
	Specify spacing distribution		V
	Measuring distance	V	V
	Command		V
	Teardrop	V	V
	Jigsaw	V	V
	Log		V
	Layer Manager	V	V
	Copper Manager	V	V
	3D Model Manager	V	V
	Jia Li Chuang SMT selection tool		V
	Welding aids		V
	Export BOM	V	V
	Export Manufacturing File Gerber	V	V
	Custom Layer/Object Export Gerber		V
	Export Coordinate File	V	V
	Export 3D file OBJ	V	V
	Export 3D file STEP		V
	Export 3D Shell File		V
	Export Test Point Report		V
	Export PDF	V	V
	Custom Object Export PDF		V
	Export DXF	V	V
	Custom Object Export DXF		V
	Export Altium Designer	V	V
	PCB Information	V	V
	Advanced PCB Settings		V

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Common Font Settings		V
Panel	Panel Design		V
	2D Preview		V
	Intersection/Difference/Union/Mutual Exclusion		V
	Export Panel Fabrication File		V
Device Library	New Device		V
	Binding Symbols/footprints/3D Models		V
	Add a picture		V
	Custom Classification		V
	Add property		V
Symbol Library	New Symbol	V	V
	New Drawing/Net ID/Net Port/No Electrical Symbol		V
	New Drawing Symbol	V	V
	Symbol Wizard	V	V
	Advanced Symbol Wizard		V
	Drawing Symbol Wizard		V
	Strip/Rectangle Multi-Pin Place		V
	Pin Manager	V	V
	Create Multipart/Sublibrary	V	V
	Bulk renumbering pins		V
	Add property	V	V
	Binding footprint	V	
Footprint Library	Create Component Footprint	V	V
	Add property	V	V
	Binding 3D Model	V	
	Pad Manager		V
	Packaging Wizard	V	

Categories	Features	EasyEDA Std Edition	EasyEDA Pro Edition
	Smart Size	V	V
	Check Dimensions	V	V
	Find Similar System footprints	V	
	Automatic cutting silk screen	V	V
3D Warehouse	Import OBJ	V	V
	Import WRL	V	
	Import STEP		V
	Replace 3D Module	V	V
Reuse Block/Hierarchy Gallery	New Reuse Block		V
Schematic Module Library	New Schematic Module	V	
PCB Module Library	New PCB Module	V	
Simulation Symbol Library	New Simulation Symbol	V	

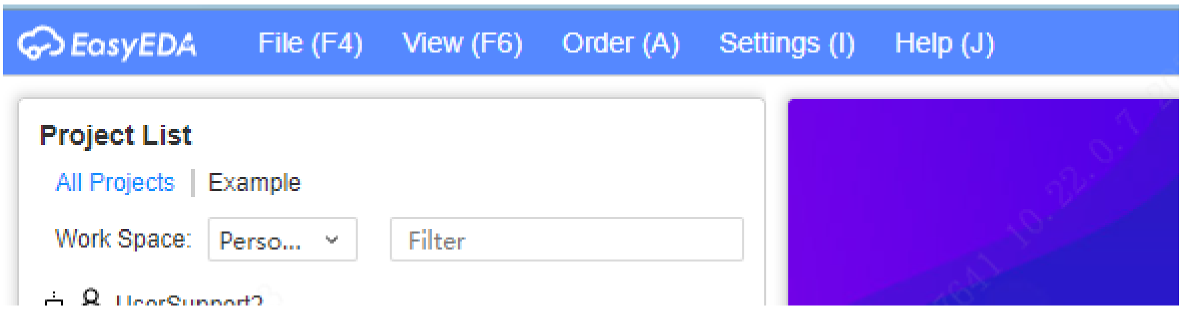
User Interface

EasyEDA has a friendly UI, you can find the feature entry

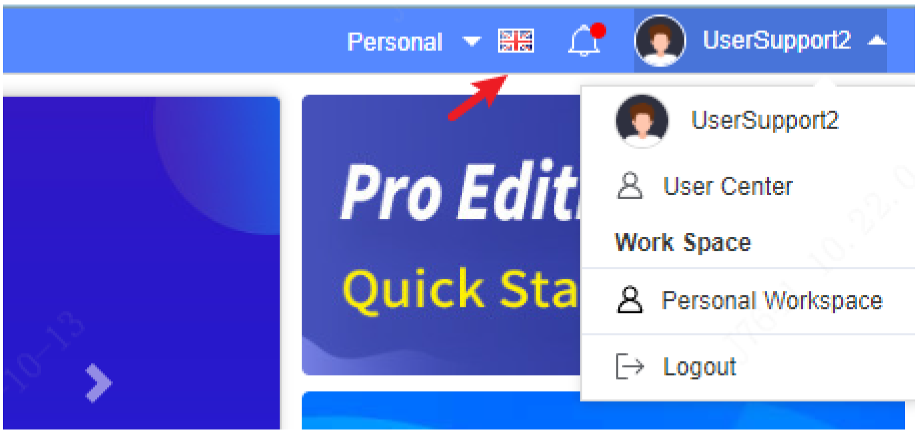


Top Menu

The start page provides the common top menu

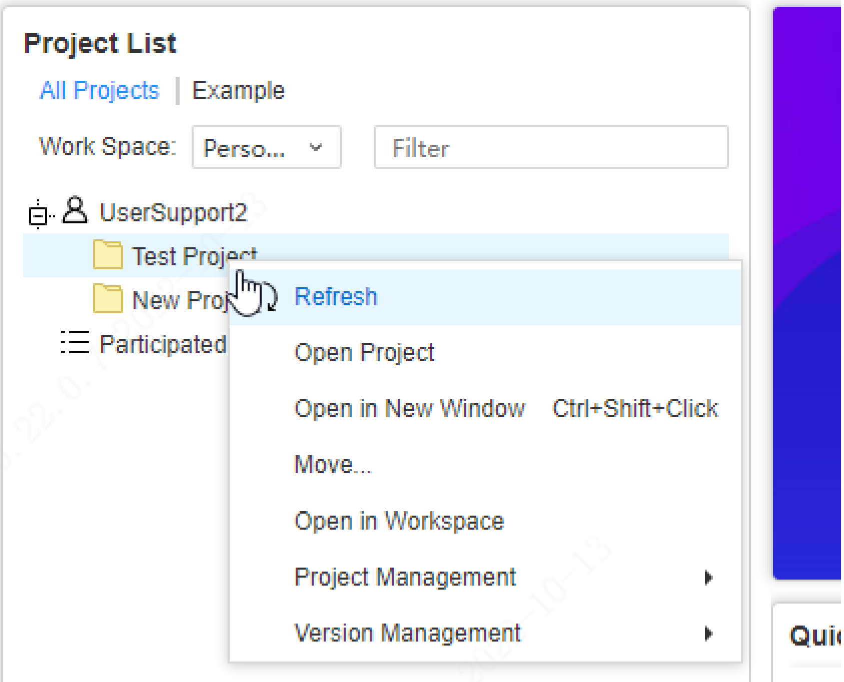


at the right-top corner, you can change the editor language and workspace, and goto user center or workspace



Project List

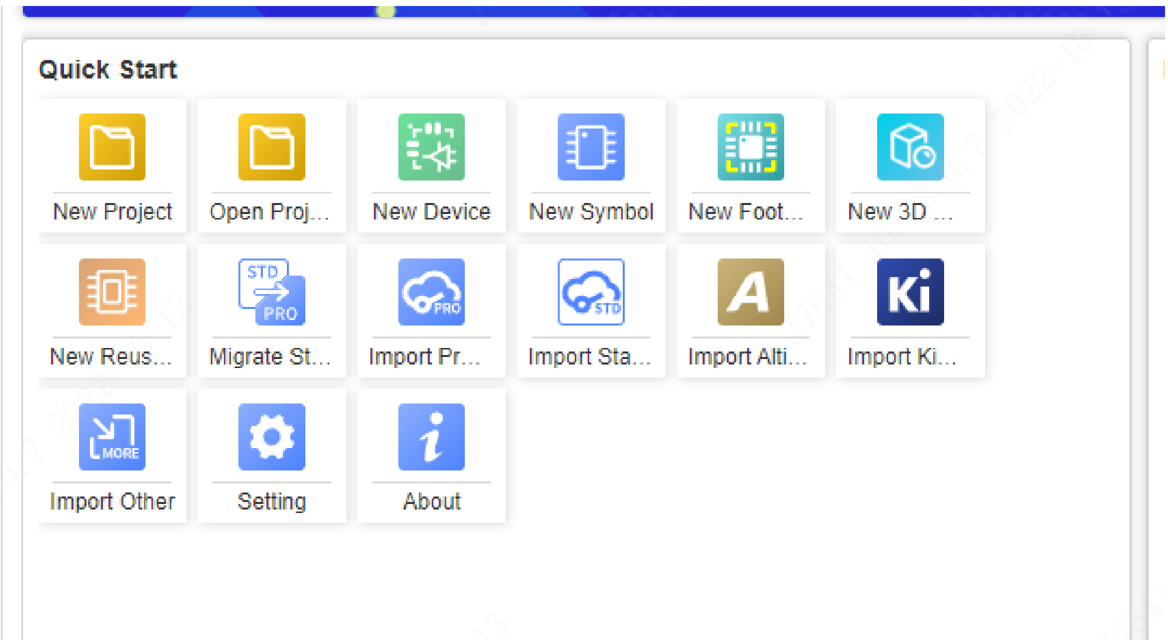
The left panel of EasyEDA Pro displays all the projects of the current user, including the joined team projects. Double-click to open the project.



And you can right-click the project to get right-click menu

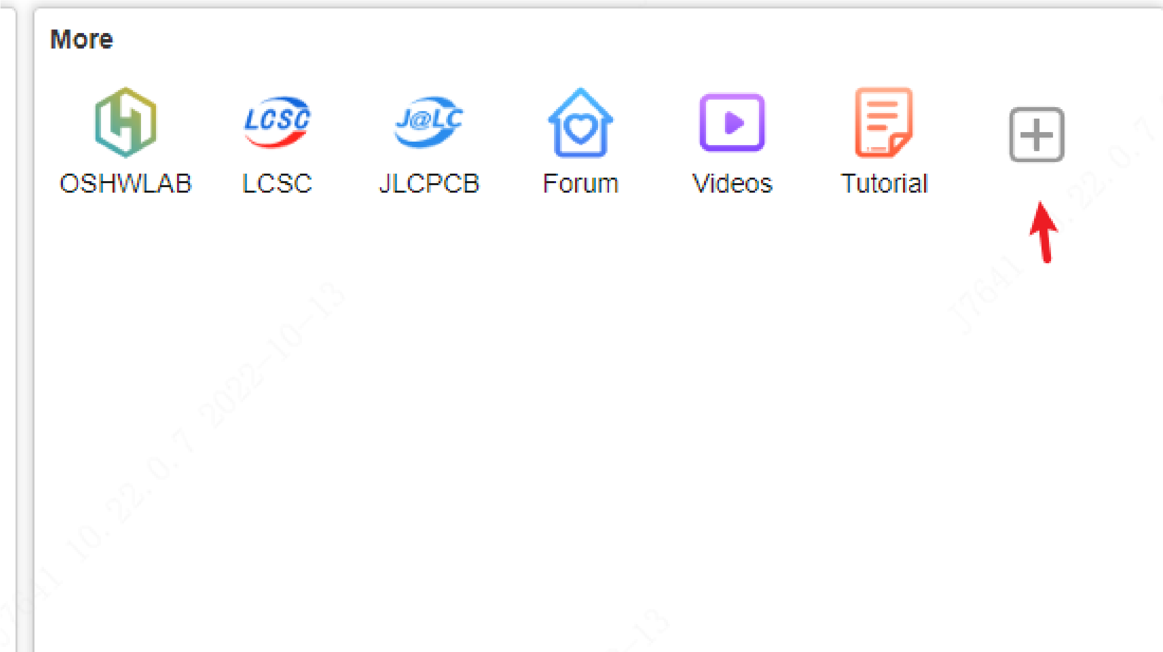
Quick Start

On the left side of the middle of the editor is a list of quick creation methods, which can quickly create projects, symbol devices and other operations on the home page.



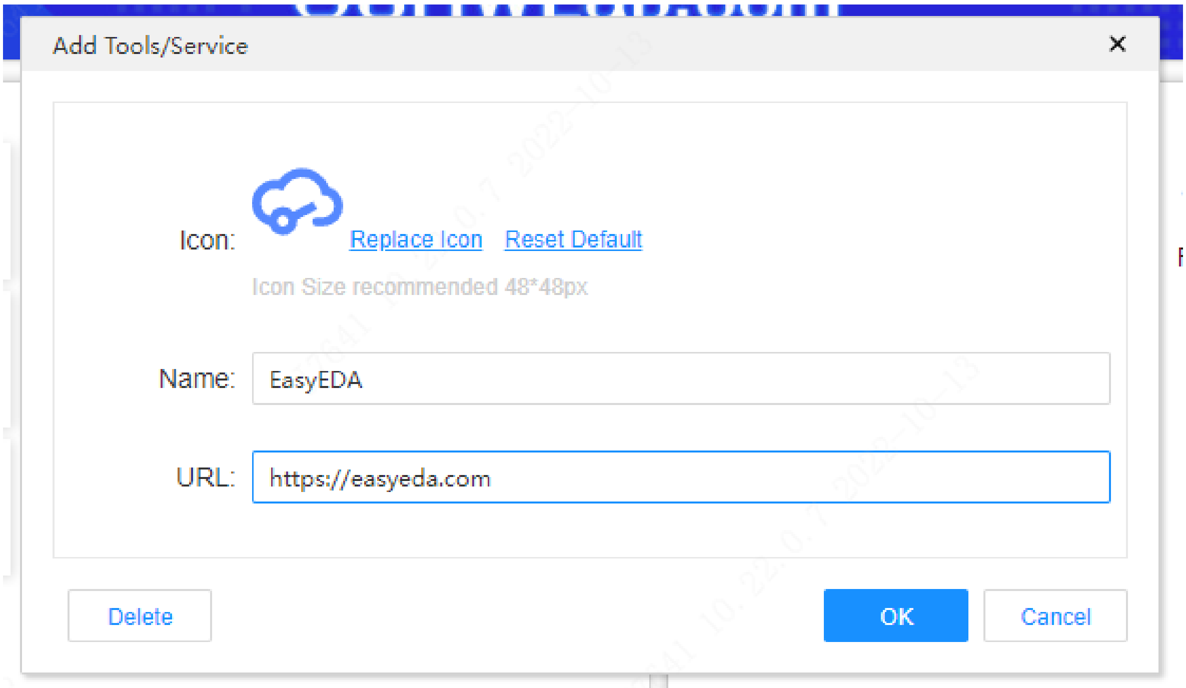
Shortcut

On the right side of the middle are some commonly used website shortcuts, you can manually add your own frequently used websites.



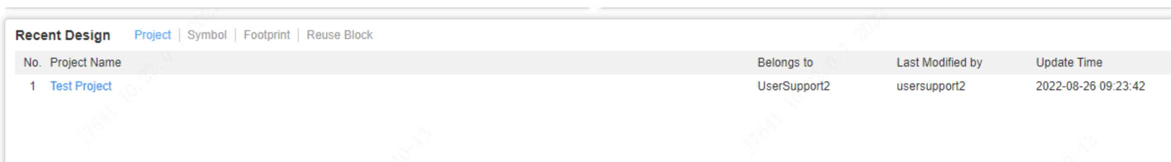
Click the "+" sign to add a website

Enter the name, URL and change icon in the popup. The size of the icon is recommended to be 48x48px. If the icon is too large, an error will be displayed.



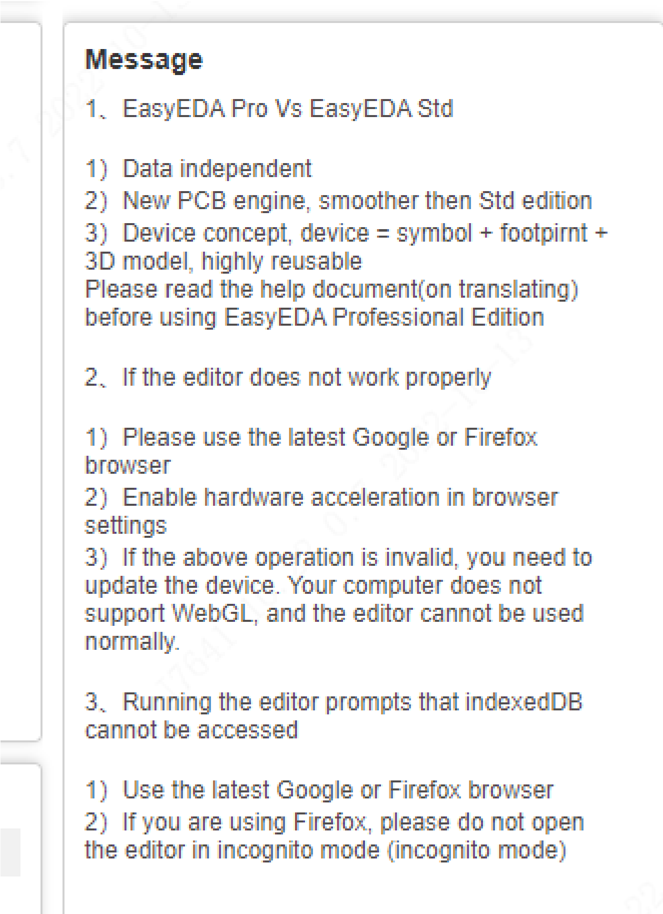
Recent Design

The bottom panel is to display recently designed projects, as well as symbols, footprints, reuse blocks. Double-click to open the corresponding project.



Message

The lower right of the editor's homepage shows some information published by our EasyEDA Pro.



Banner

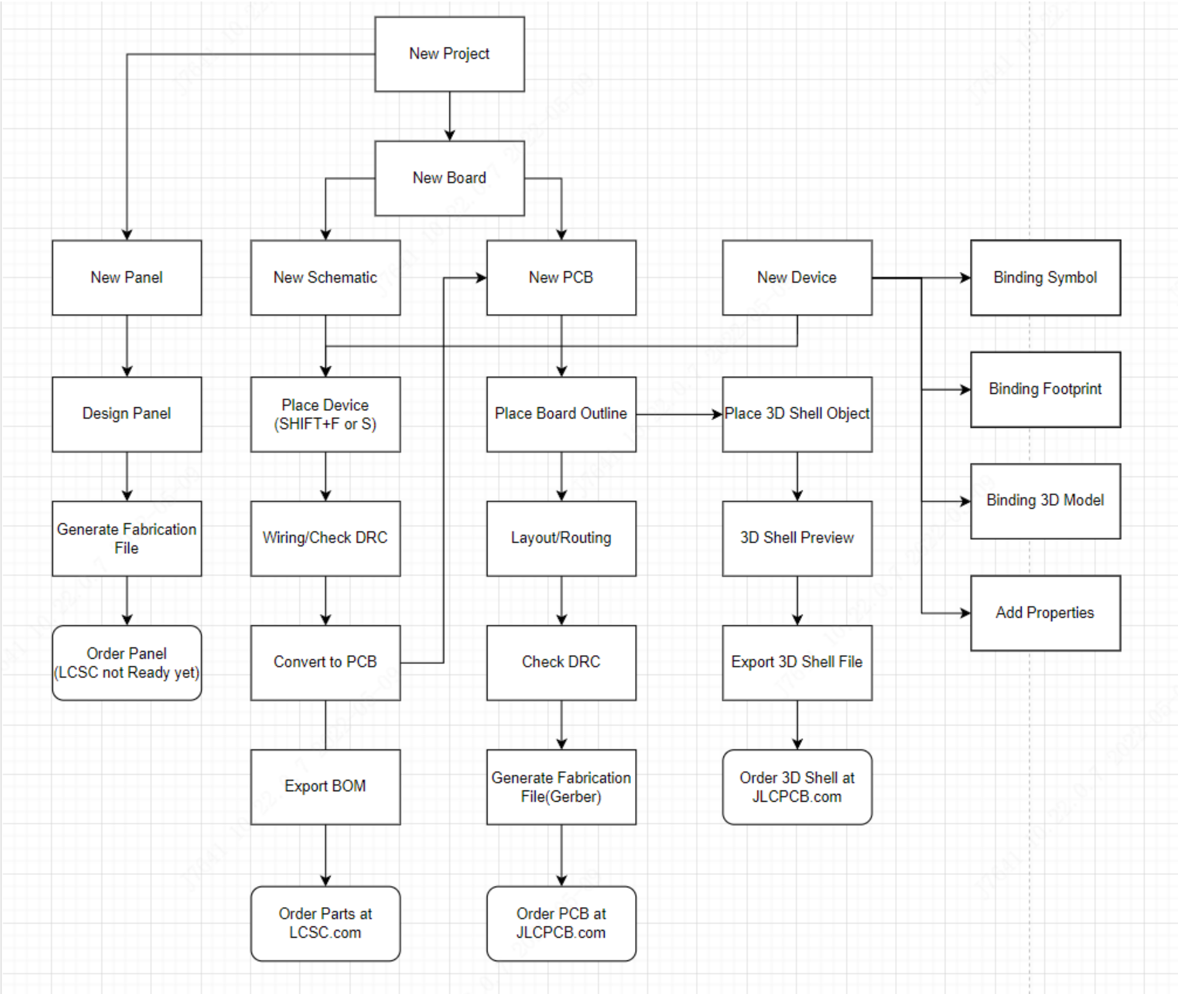
The icon on the upper right side is the link of our professional version tutorial, click yes to jump to our EasyEDA Pro tutorial.

Design Flow

The simple design process is as follows:

1. If there is no required library, draw the library first: device, symbol, footprint. The device requires a binding symbol and footprint.
2. Create a new project and place the device on the schematic

3. Schematic to PCB
4. Export BOM and Gerber order.

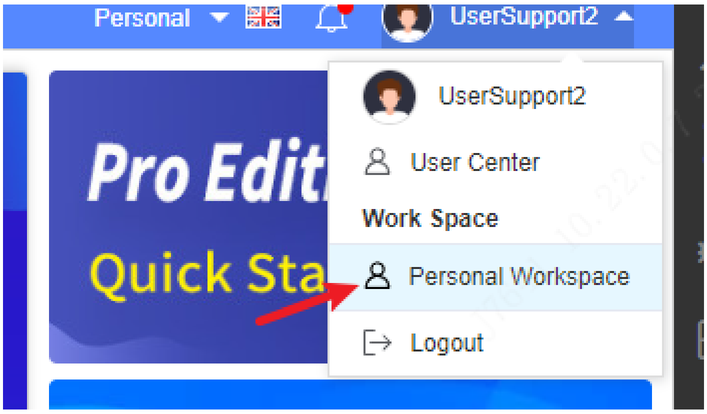


Teamwork

EasyEDA Pro is a very good EDA for online team collaboration, adding project members, and setting project members permission management.

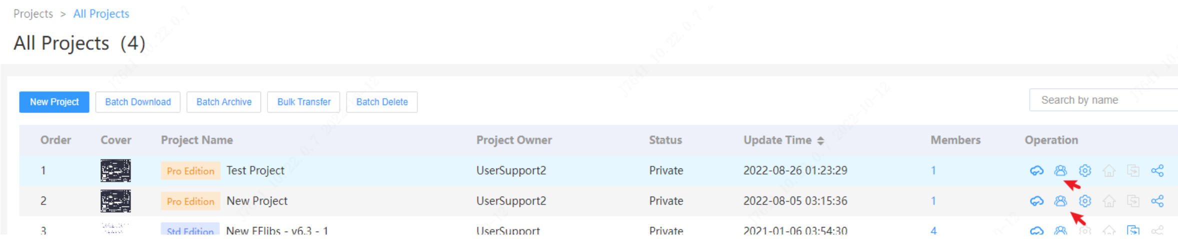
Steps:

- Click Edit Avatar in the upper right corner to enter the project management interface.

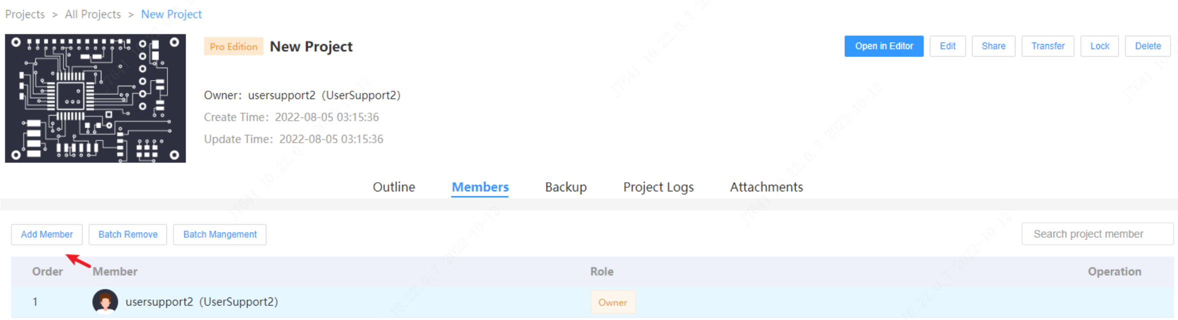


In the project management interface, click Member to enter the project member management interface.

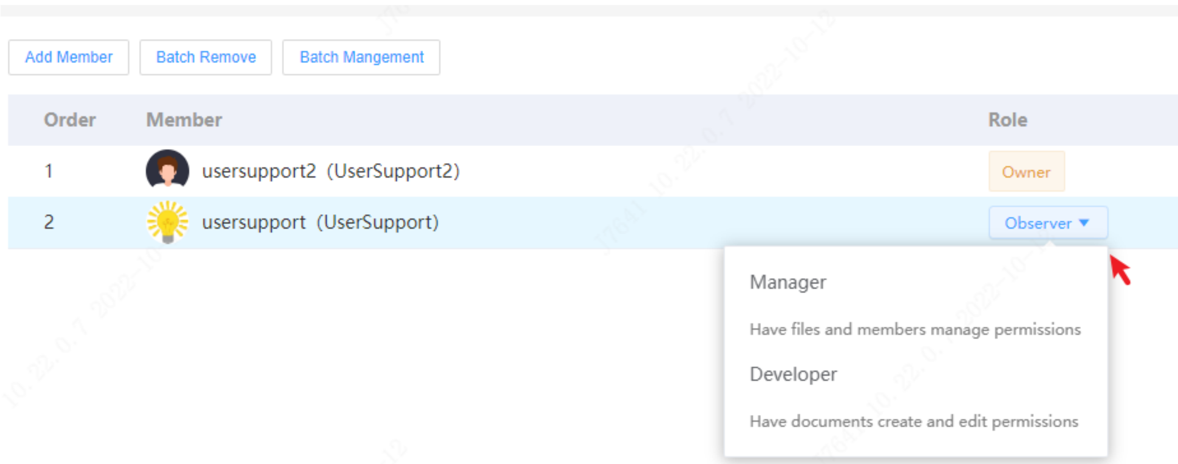
In the project member management interface, you can see the owner of the project, the creation time, the last modified person and the last modified time.



then add the members:



search the username to add as member and then you can set the rule

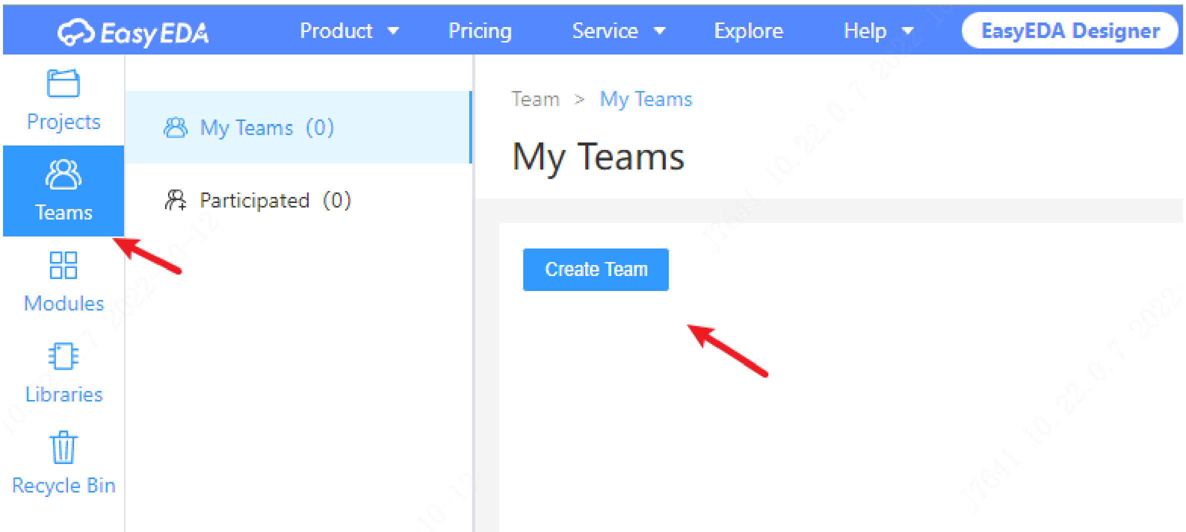


Select the permission of the member and click Add to add a member to the project.

- **Super Admin**: The owner of the personal project. Possess all operation rights to the project
- **Administrator**: Has the operation rights of project documents, project settings, project downloads, and removal of project members (except super administrators).
- **Developer**: Have the permission to create and edit project documents and attachments.
- **Observer**: has the right to view project documents and attachments.

create team

In a workspace, collaborative development is possible by creating a team.

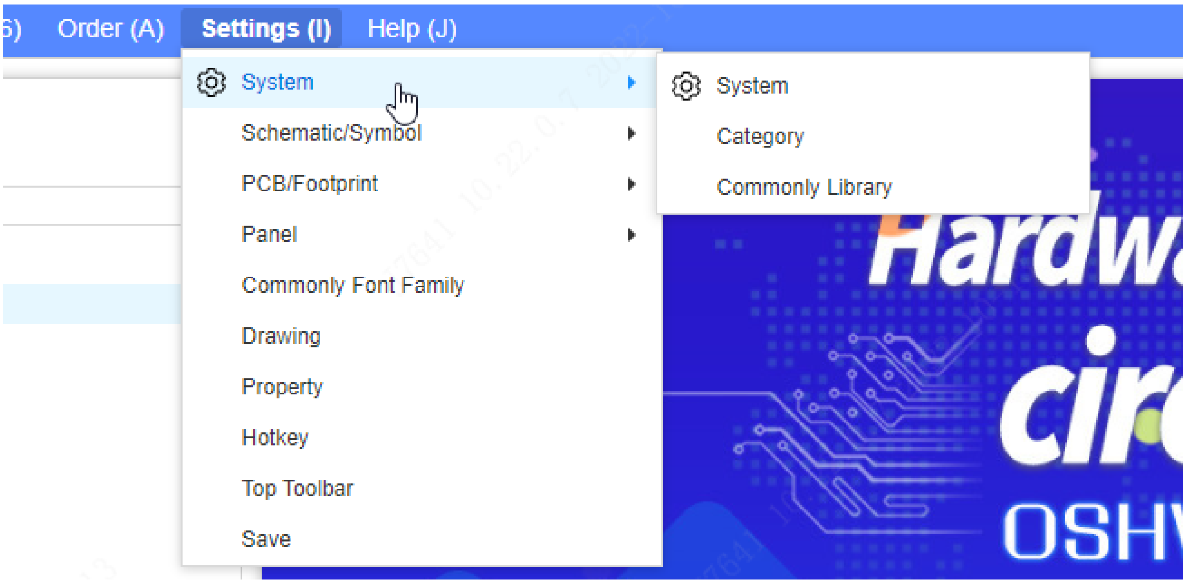


Add members to the team and set permissions for the team.

When creating a library or project, select a team, then you can set different project permissions for members of the team.

Personal Settings

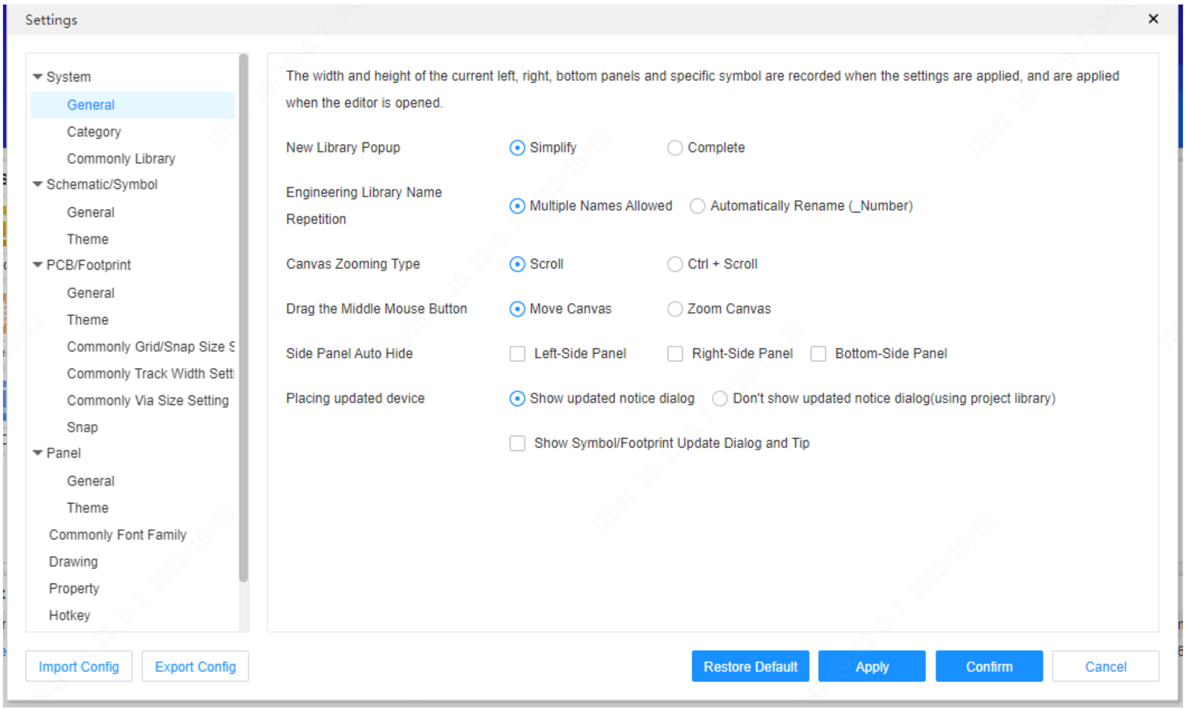
The personal settings of EasyEDA Pro are synchronized to the server, no matter which browser you log in to, it can be automatically synchronized, reducing multiple configuration operations.



System

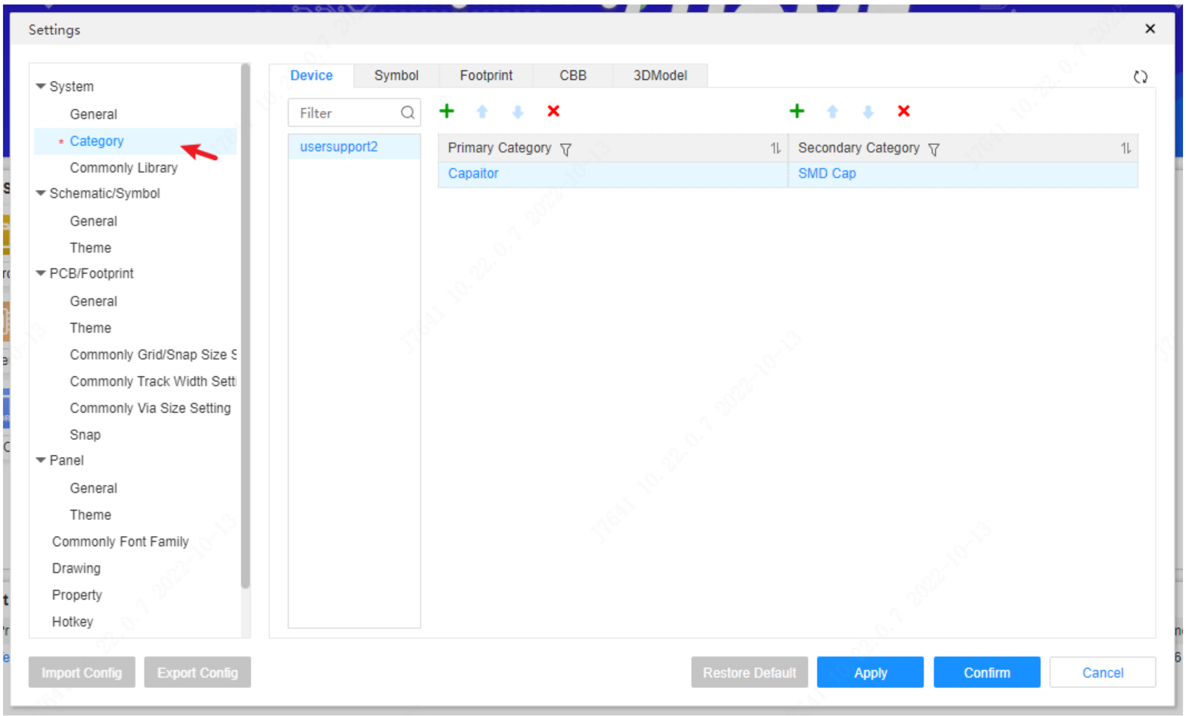
Canvas zoom: Default mouse wheel zoom, you can modify Ctrl + wheel zoom according to personal preference. During the drawing process, long press the right mouse button to move the canvas, and by default, click the right mouse button to cancel the drawing.

Show symbol/footprint update dialog and prompt: After checking this box, a prompt box will pop up every time a symbol or footprint is updated.



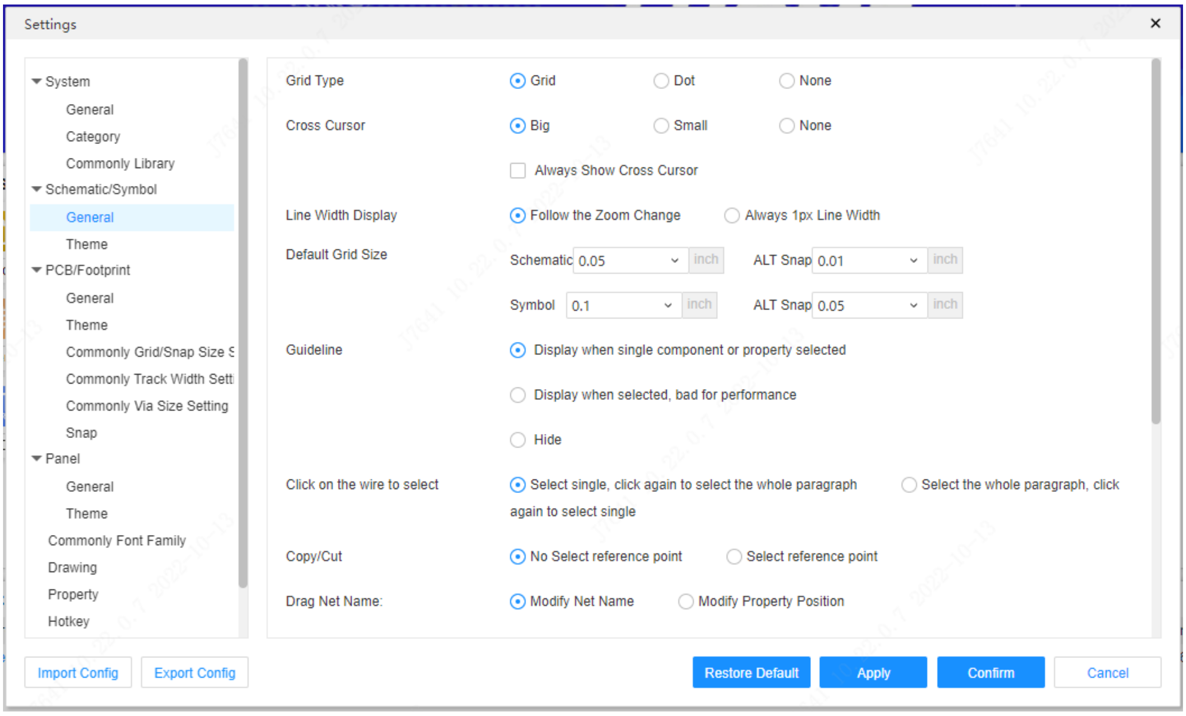
Category

Add and remove settings for device, symbol, footprint etc. Category options.



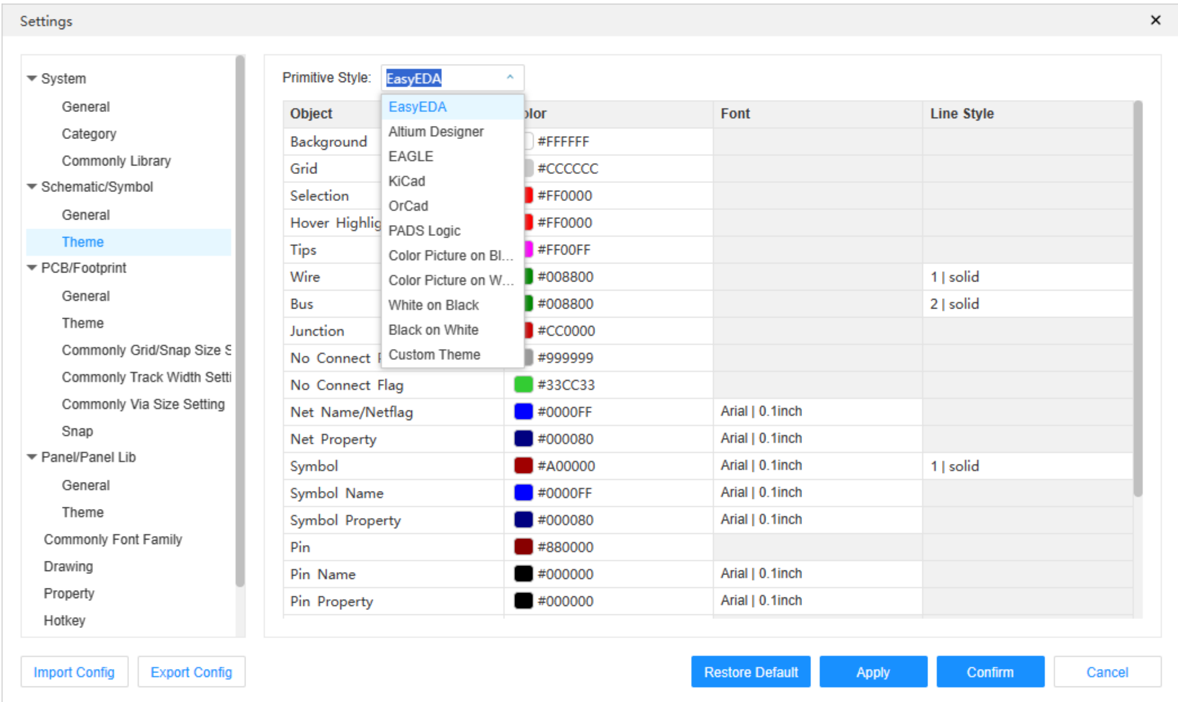
Schematic/Symbol

Some personal manipulation settings for the schematic and symbol interface.



Theme

Set the schematic sheet color theme.



Support click to modify parameters such as color/stroke color, fill color, font, and line type.

PCB/footprint

Settings for the PCB and footprint drawing interface.

Theme

Drawing interface color settings for PCB and footprint interface.

Common grid sizes

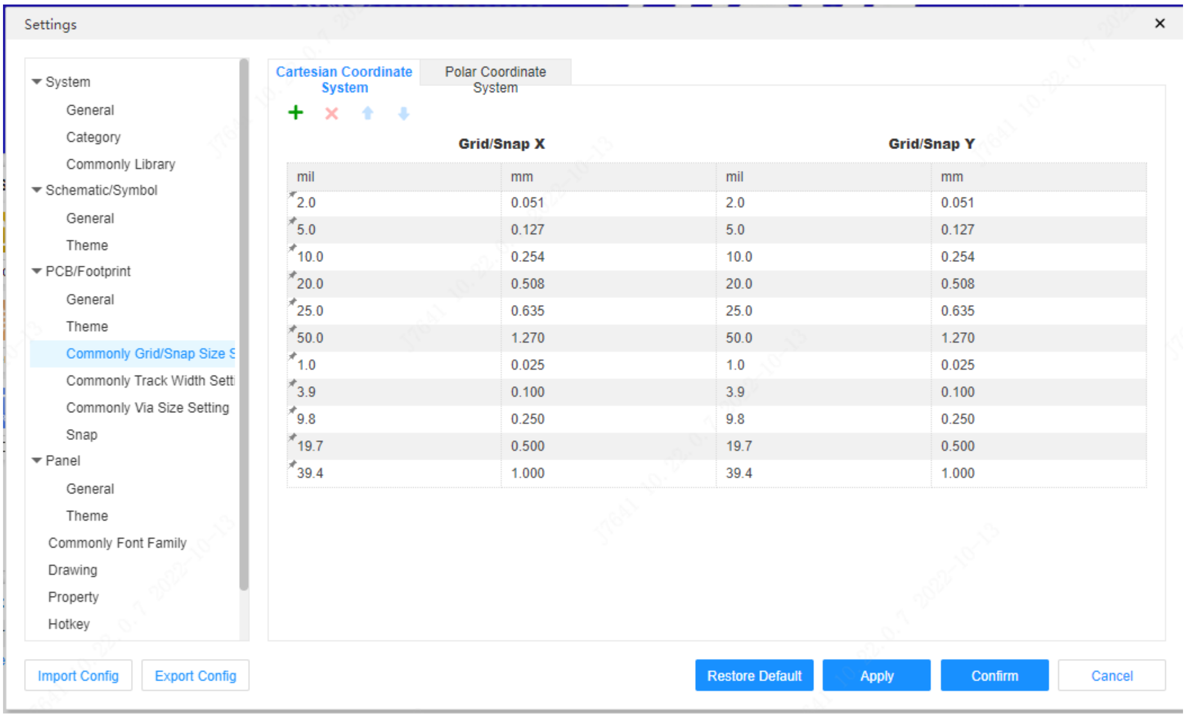
Set common sizes for PCB and footprint interface grids for easy change.

Common line width

Sets the line width settings for PCB traces.

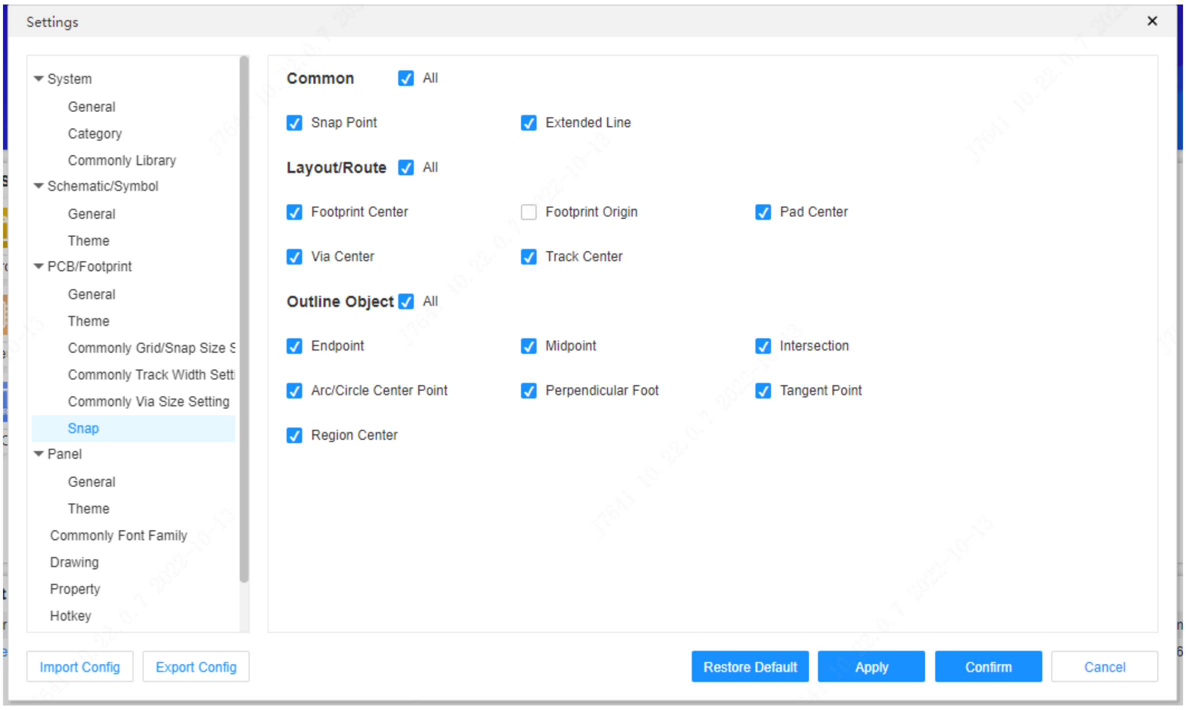
Common via size

Sets common via settings for PCB traces.



Snap

It supports a variety of adsorption settings, which can be very convenient for various adsorption.



Panel

Supports defining panel document related settings.

theme

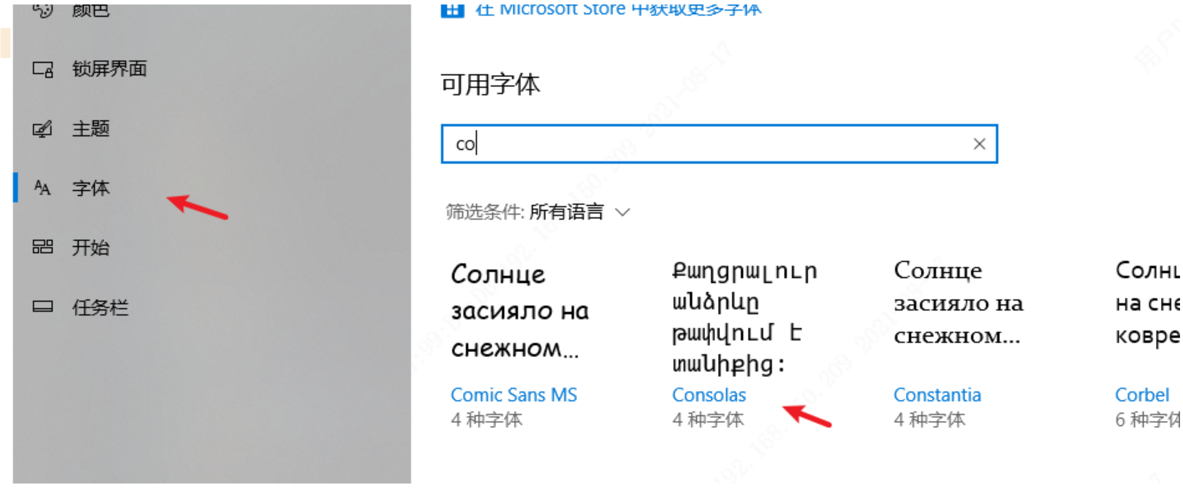
The theme to apply when the panel document is created.

Common fonts

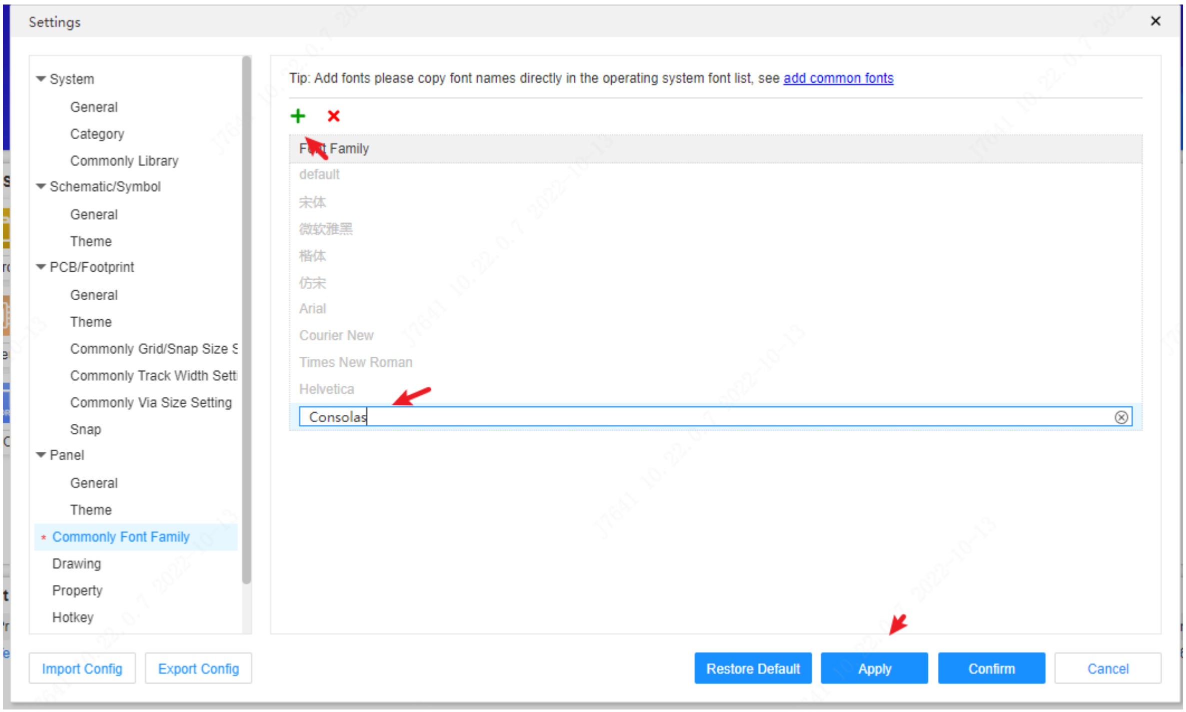
Support to set your own common fonts. These fonts need to be installed on the local computer before they can be called by the editor, otherwise the editor will automatically use the default font provided by the browser for rendering.

Add custom fonts:
1) First install the fonts you need on your local computer. If you already have fonts, you can ignore this step.

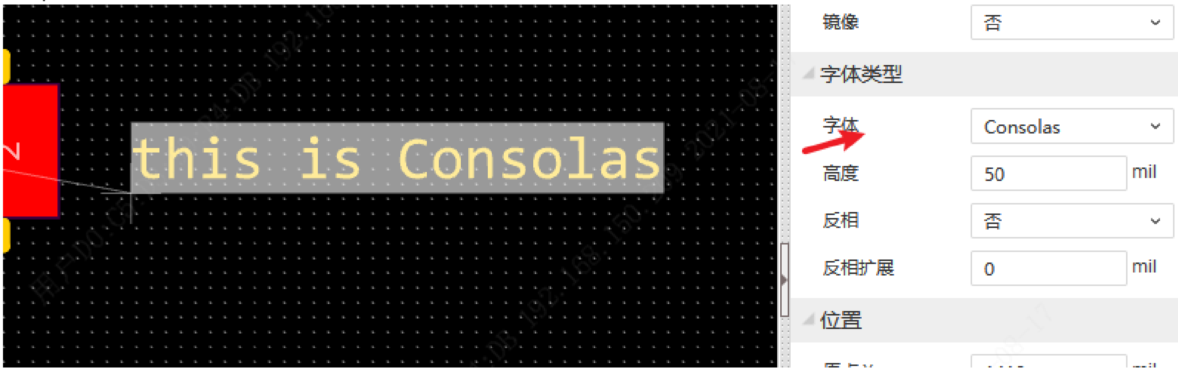
2) Take the Windows system as an example, find the font settings in the system settings, and get the font name.



3) Add a font based on the font name.



4) This font can be seen in the schematic or PCB text font switching, and the font switching can be performed at this time.

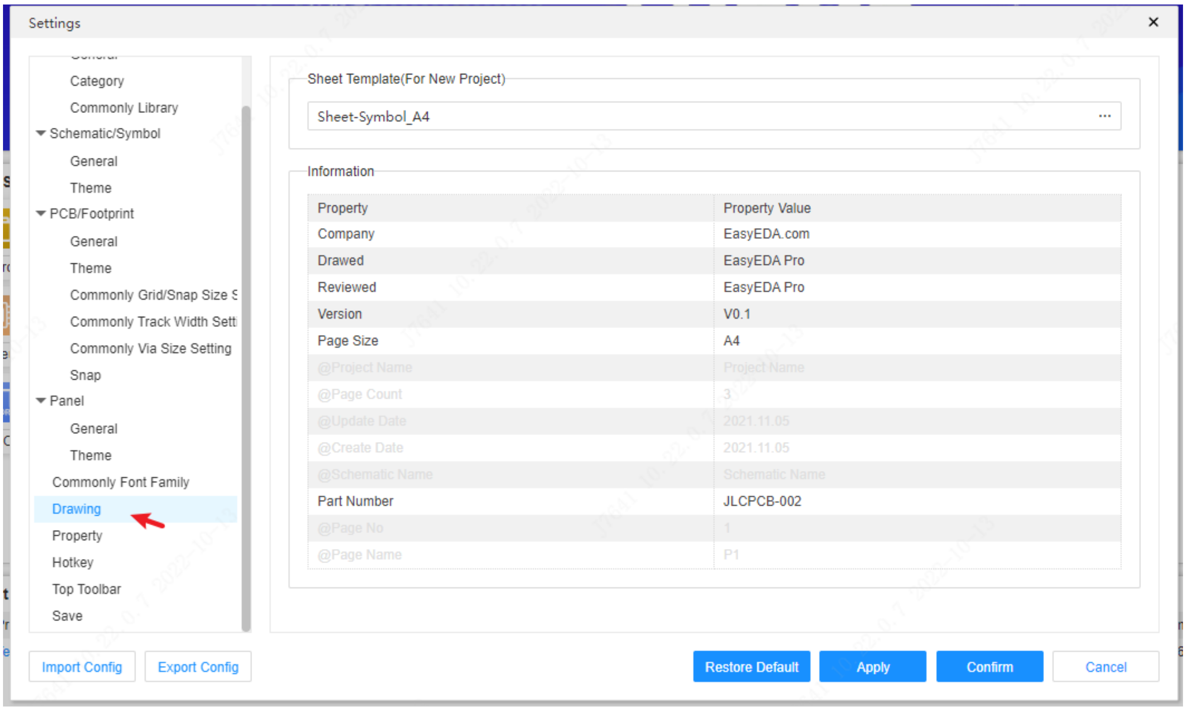


This added font can be used by schematics, panels, PCBs, etc. When the text switches fonts, you can see the new font.

Attention:
1, the editor uses the operating system native fonts, if you need to use these fonts for commercial purposes, please make sure that you have the corresponding font use copyright. It is recommended to use free open source fonts.
2, because the editor is running in the browser (including the client), when the operating system does not install the corresponding name of the font, the browser will automatically assign the default font of the current browser to display, at this time there may be the text font name is correct, but the text style is not correct.

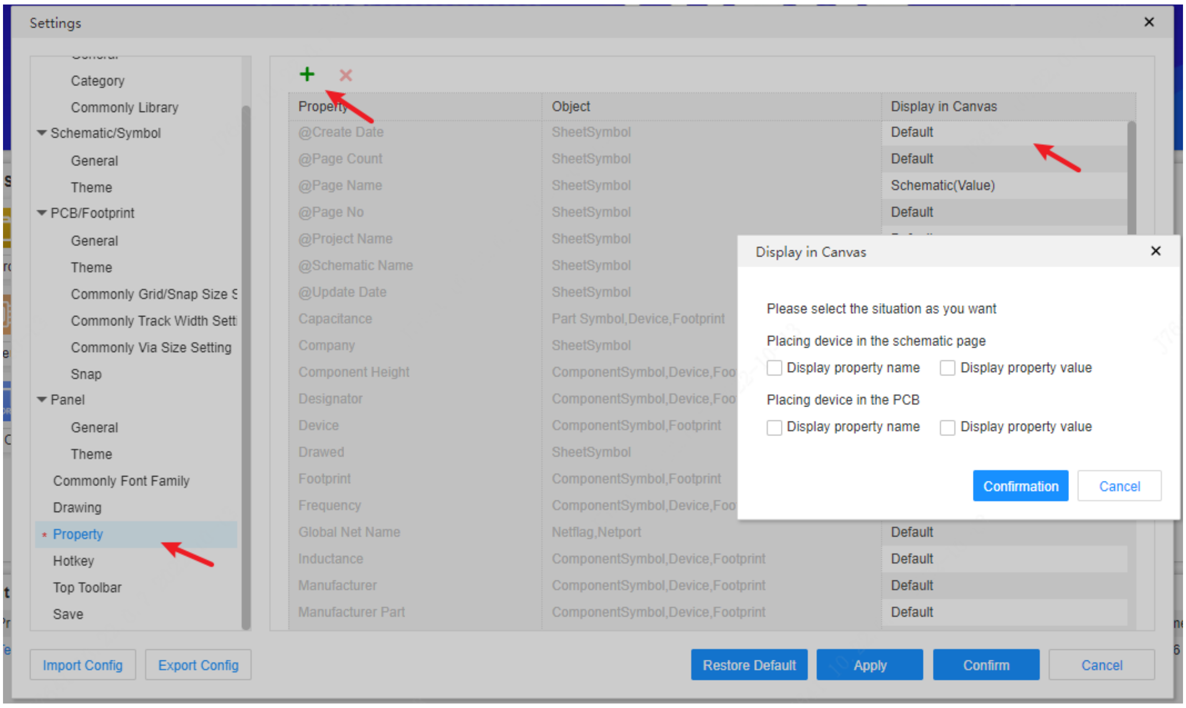
Drawings

Set the basic information of the default drawing when creating a new project.



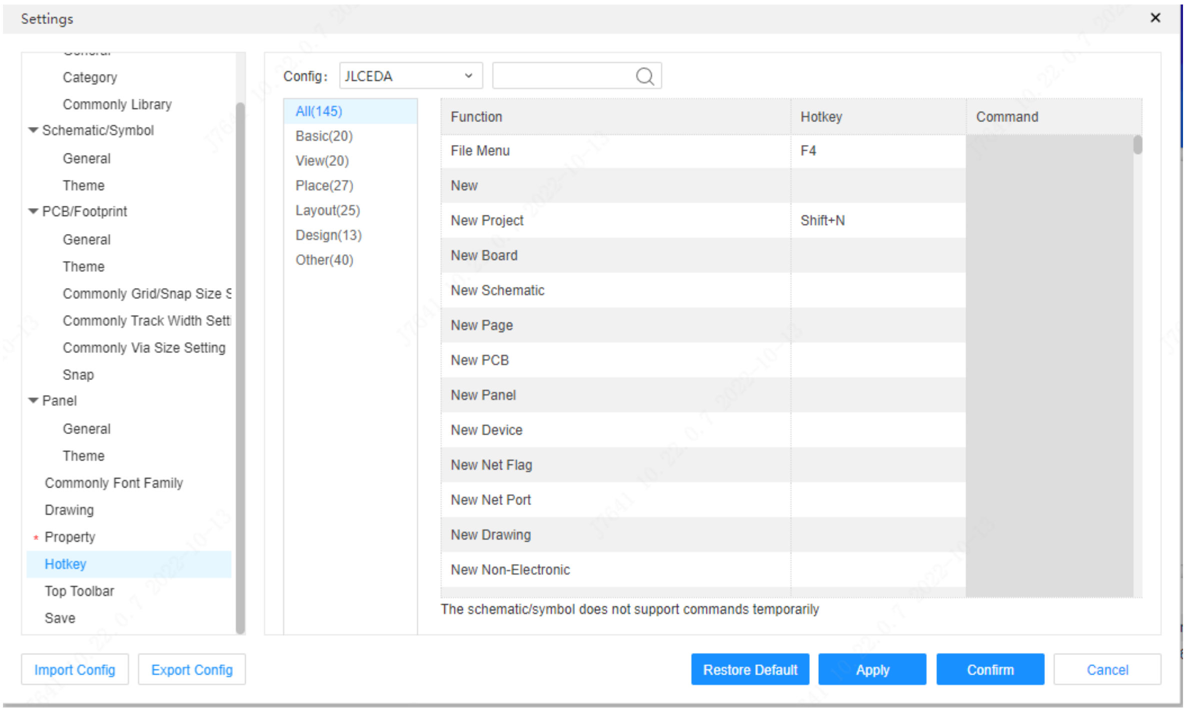
Property

Add or delete custom properties for devices, footprints, and symbols.



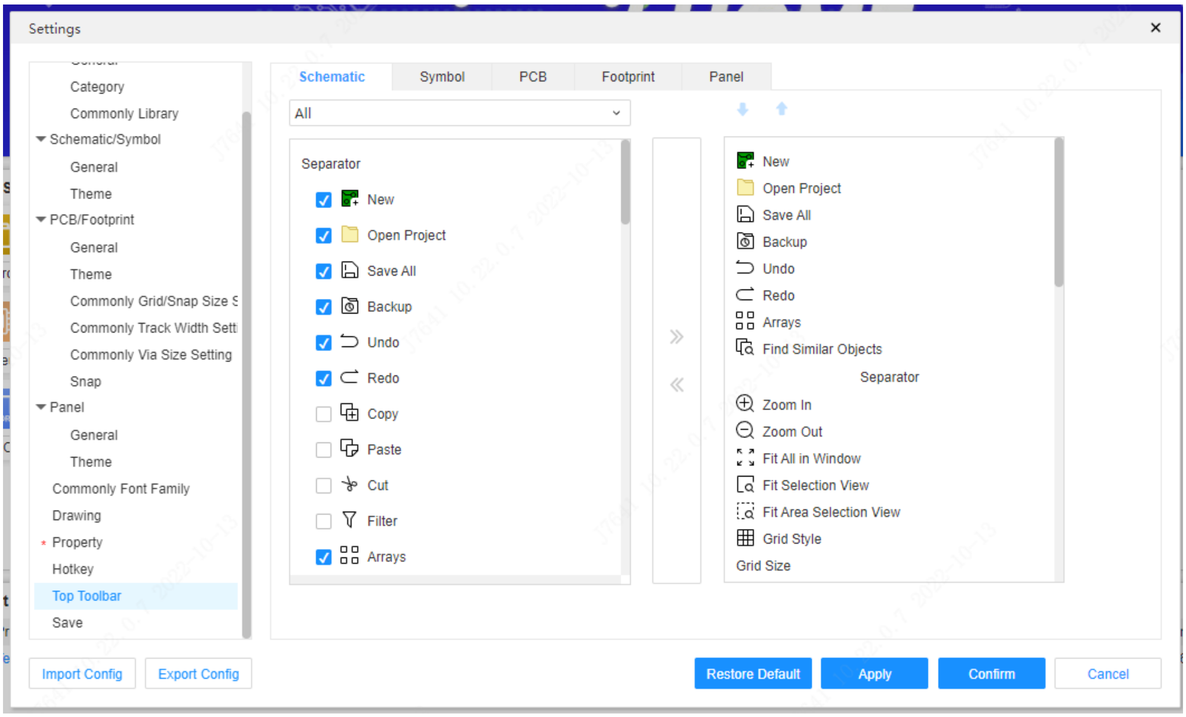
Hotkey

View and modify custom shortcut keys and commands.



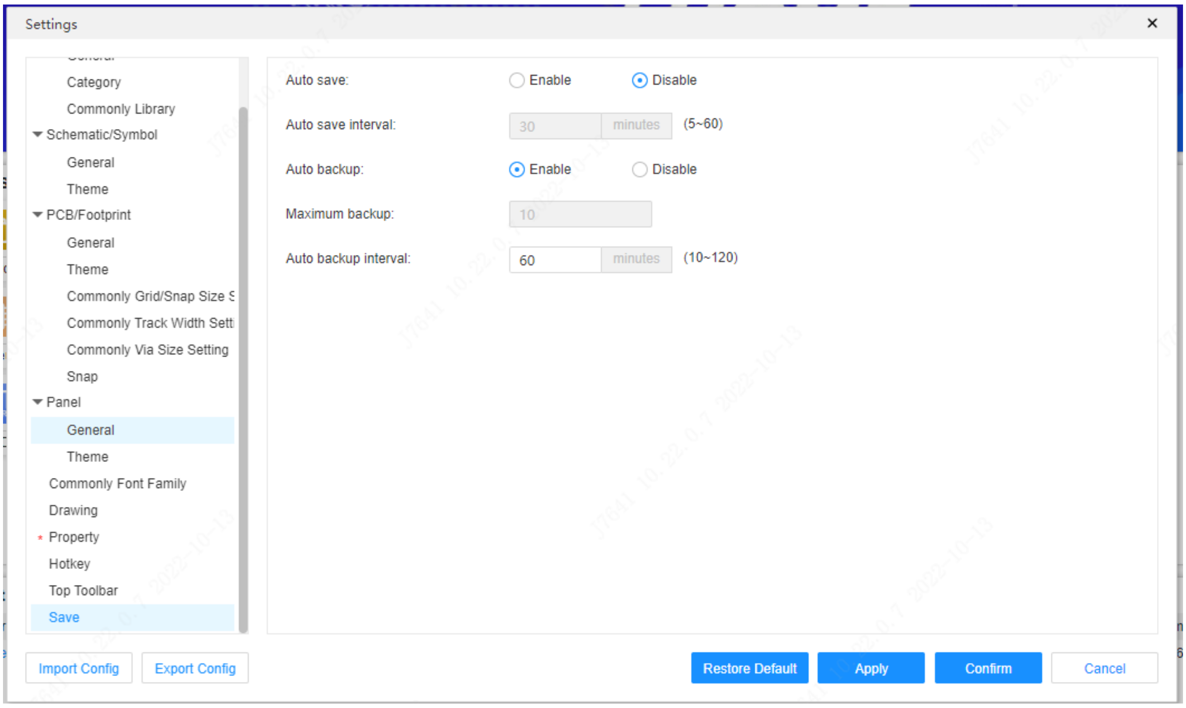
Top Toolbar

Adds and removes settings for quick Place on the top toolbar.



Save

You can set the automatic saving of documents, the time and quantity of saving, and the automatic backup parameter settings of the project.

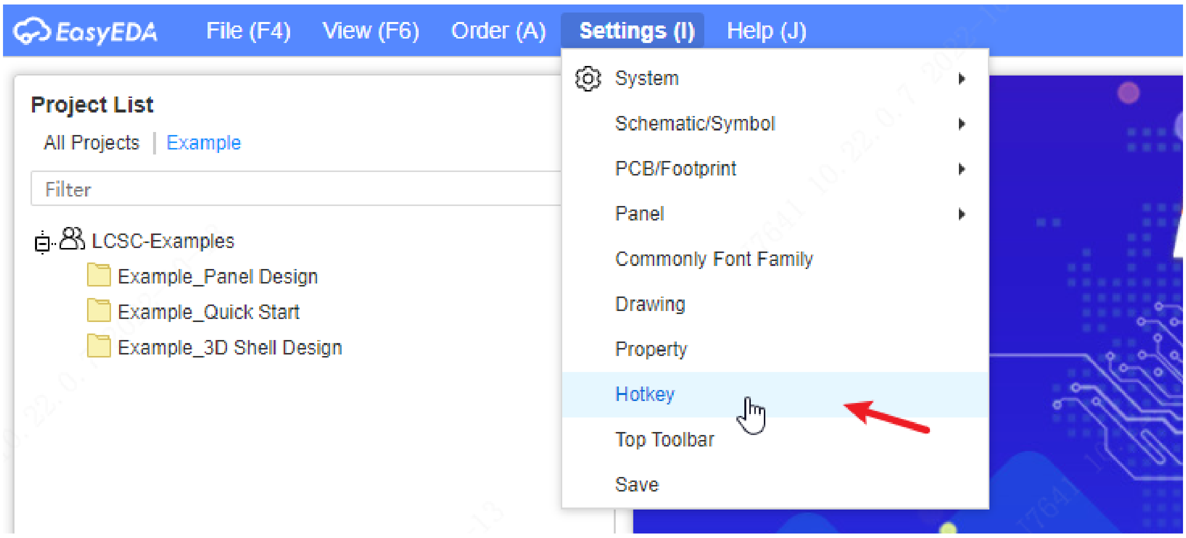


Shortcut Key

EasyEDA Pro provides many shortcut keys for users to use, and each shortcut key can be configured.

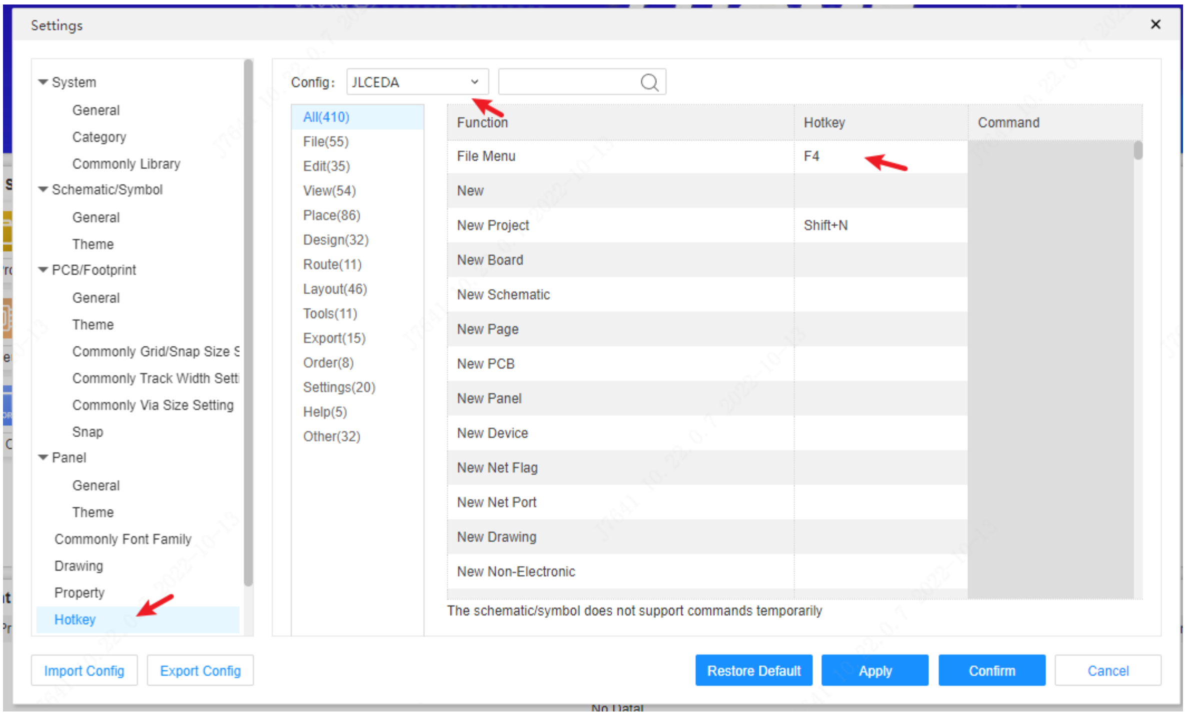
Enter the shortcut key configuration options:

Top Menu - Settings - Shortcuts



Modify shortcut keys

Double-click the shortcut key editor, then enter the corresponding modified shortcut key, and click OK to modify.



Common shortcut keys

The same shortcut keys can be used in different documents. The default shortcut keys are listed below.

Function	Shortcuts	Remarks
Cancel the drawing state (default) / close the popup	Right mouse button (click), ESC	
Open project	Ctrl+O	
New Project	Shift+N	
Save	Ctrl+Shift+S	
Save All	Ctrl+S	
Undo	Ctrl+Z	
Redo	Ctrl+Y	
Cut	Ctrl+X	
Copy	Ctrl+C	
Paste	Ctrl+V	
Cross selection	Shift+X	
Round Pass	Ctrl+Shift+X	
Move according to the center	M	
Delete selected	Delete	
Previous Page	Page Up	
Next page/New page	Page Down	
Previous widget	Page Up	
Next widget/New widget	Page Down	
Full Screen	F11	
Select All	Ctrl+A	
Find Replace	Ctrl+F	
Find Similar Objects	Ctrl+Shift+F	

Function	Shortcuts	Remarks
Zoom in	A	It is not set by default since v1.8, you need to set it yourself, reserve a single key for the top menu
Zoom out	Z	It is not set by default since v1.8, you need to set it yourself, reserve a single key for the top menu
Fit All	K	
View scroll left	Left	
View scroll right	Right	
View scroll up	Up	
View scroll down	Down	
Highlight Network	H	
Cancel highlight network	SHIFT+H	
Unit	Q	
Expand/Collapse Bottom Panel: Library, DRC, Find Structure, Log	S	
Rotate Left	Space	
Align Left	Ctrl+Shift+L	
Center Left and Right	Shift+Alt+E	
Align right	Ctrl+Shift+R	
Top Align	Ctrl+Shift+O	
Center up and down	Shift+Alt+H	
Align Bottom	Ctrl+Shift+B	
Snap to Grid	Ctrl+Shift+G	
Horizontal equidistant distribution	Ctrl+Shift+H	
Vertical equidistant distribution	Ctrl+Shift+E	
Move the selected graph to the left	Left	
Move the selected graph to the right	Right	
Move the selected graph up	Up	
Move the selected graph down	Down	

Function	Shortcuts	Remarks
Display properties dialog when placing element	Tab	
Switch selection range when an object is selected	Tab	
When drawing a rectangle, keep it as a square; when drawing a wire, the mouse moves horizontally or vertically	Shift (long press)	
Move the canvas (default)	Right mouse button (long press)	
Repeat to cursor	Ctrl+D	
Repeat to other layers	Ctrl+Shift+D	
Combined selection	Ctrl+G	
Ungroup	Shift+G	
Open a document in a new window	Ctrl+Shift+Left key	
Close current tab	Shift+W	
Close all tabs	Shift+Alt+W	
Open help document	F1	

Schematic/symbol shortcuts

Function	Shortcuts
Draw Wire	W
draw bus	B
draw pins	P
Draw Arc	Alt+A
draw rectangle	R
Draw a circle	Alt+C
Draw Polyline	Alt+L
draw text	T
Place Network Labels	N
Highlight/Un-Highlight Net Wire	H
Place Device Dialog	Shift+F
Display pin properties dialog when placing pin element	Tab
Show Net Label Information dialog when placing net labels	Tab
Display the Network Port Name Information dialog when placing a network port	Tab

PCB/package shortcuts

Function	Shortcuts
Wiring	W
Switch common line weights while routing	Shift+W
Toggle common vias when routing/placing vias	Shift+V
Differential Pair Routing	Alt+D
Equal Length Route	Shift+A
Increase the gap during isometric adjustment	Num+
Reduce the gap during isometric adjustment	Num-
Highlight/Un-Highlight Net Wire	H
Show/hide selected flylines	Ctrl+R
Unhighlight all	Shift+H
Toggle layer brightness	Shift+S
Flip the board	F
Place Vias	V
Place a single pad	P
Combined selection	Ctrl+G
Ungroup	Shift+G
Flip left and right	X
Upside down	Y
Single Wiring	W
Routing Corner	L
Distance	ALT+M
Command	C
Switch units	Q
switch to top level	T
switch to bottom layer	B
switch to inner layer 1	1
switch to inner layer 2	2
switch to inner layer 3	3
switch to inner layer 4	4
Switch to next copper layer	Num*

Function	Shortcuts
Switch to previous copper layer	Shift+Num*
Hide/Show copper area	Shift+M
Rebuild all coppers	Shift+B
Done	Enter
Cancel	Esc
Back	Backspace
Select overlapping entities	G
Flip path while drawing	Space
Show all layers	Ctrl+L
Temporarily show/hide network name	Ctrl+Q
Snap	Alt+S

Project

Create Project

When EasyEDA Pro creates a project, it will create a board, a schematic diagram and a PCB by default, so there is no need to create a project and then create a PCB.

Note: You need to create a project file and a board to create a drawing page.

Steps:

- Editor start page, create project directly
- Top Menu - New - Project. The new project will automatically create a board, which contains the schematic diagram and PCB

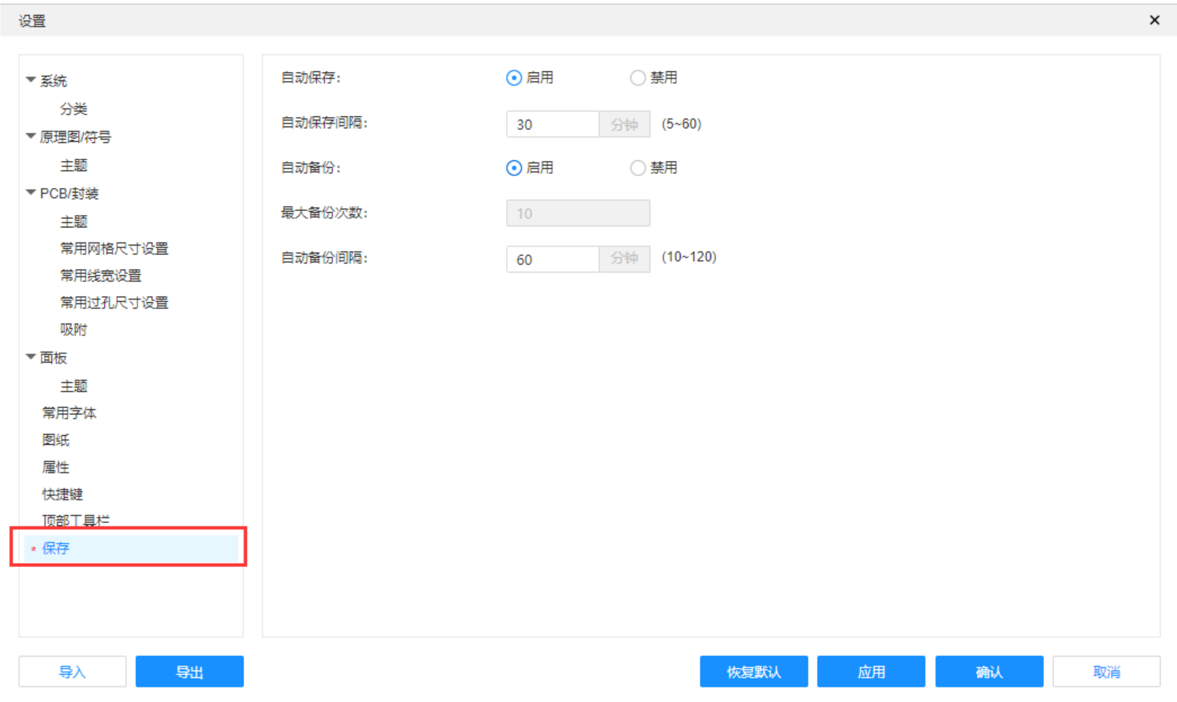
Create/Switch Project Version

Automatically create version

Back up the project to our EasyEDA Pro cloud, the backup can be divided into two types: automatic backup and manual backup.

Automatic backup needs to be turned on in the settings first. Check Enable to enable automatic backup. The maximum number of backups for automatic backup is 10 copies. If the number of backups exceeds 10 points, the old backup will be overwritten.

The automatic backup time can be set to 10-120 minutes.

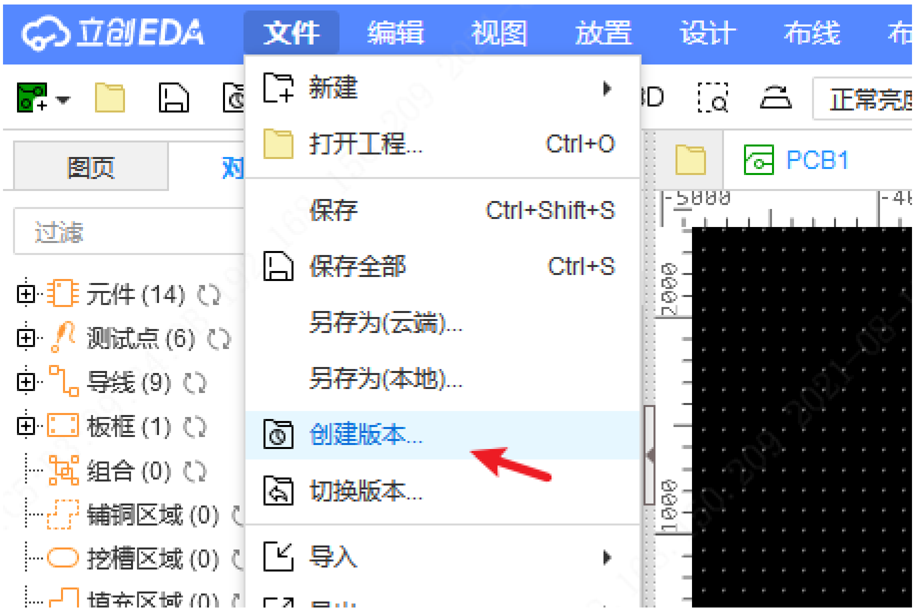


Manually create a version

For automatic backup, you need to manually back up the project to the server, and the number of manual backups can only be up to 15 copies.

Steps:

- Top Menu - File - Create Version



Enter the name and description in the pop-up window, and click OK to back it up in the cloud.

switch version

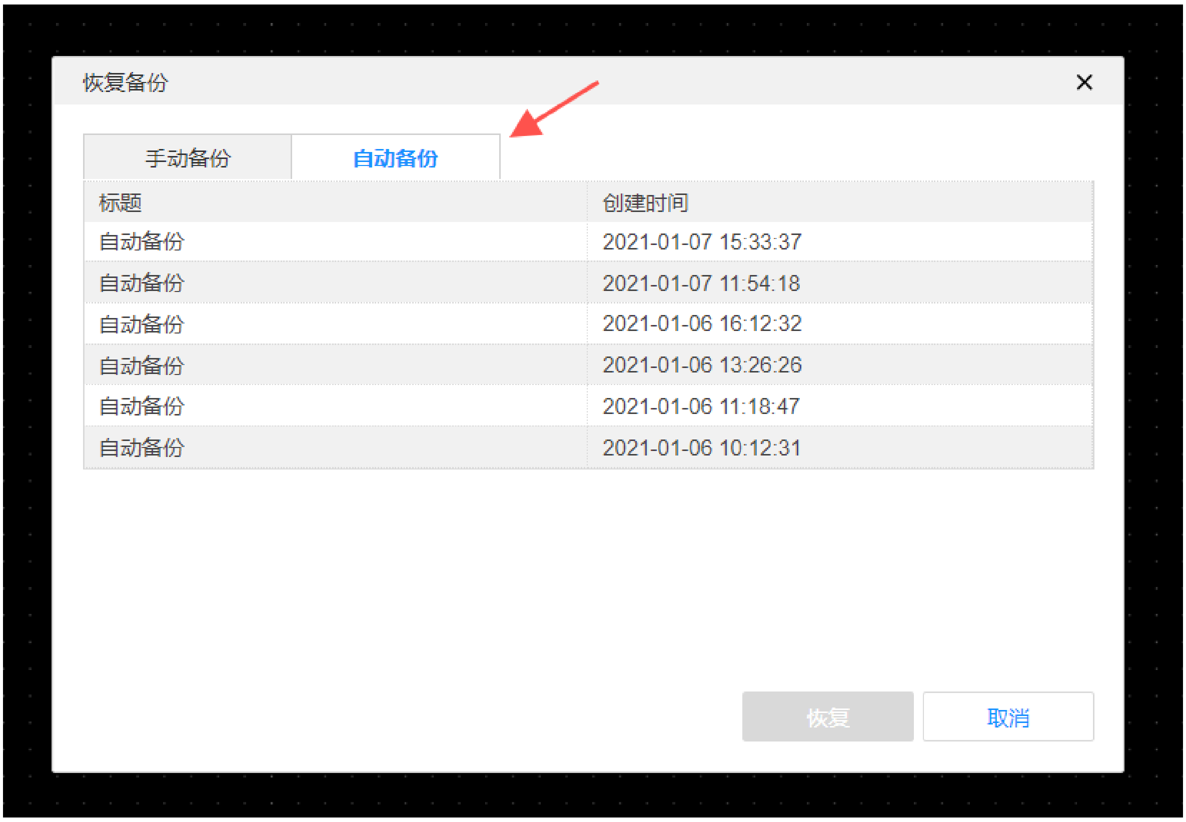
At present, EasyEDA Pro does not support switching versions directly on the project. The current switching version is implemented by creating a new project.

Restore projects designed in the cloud or before

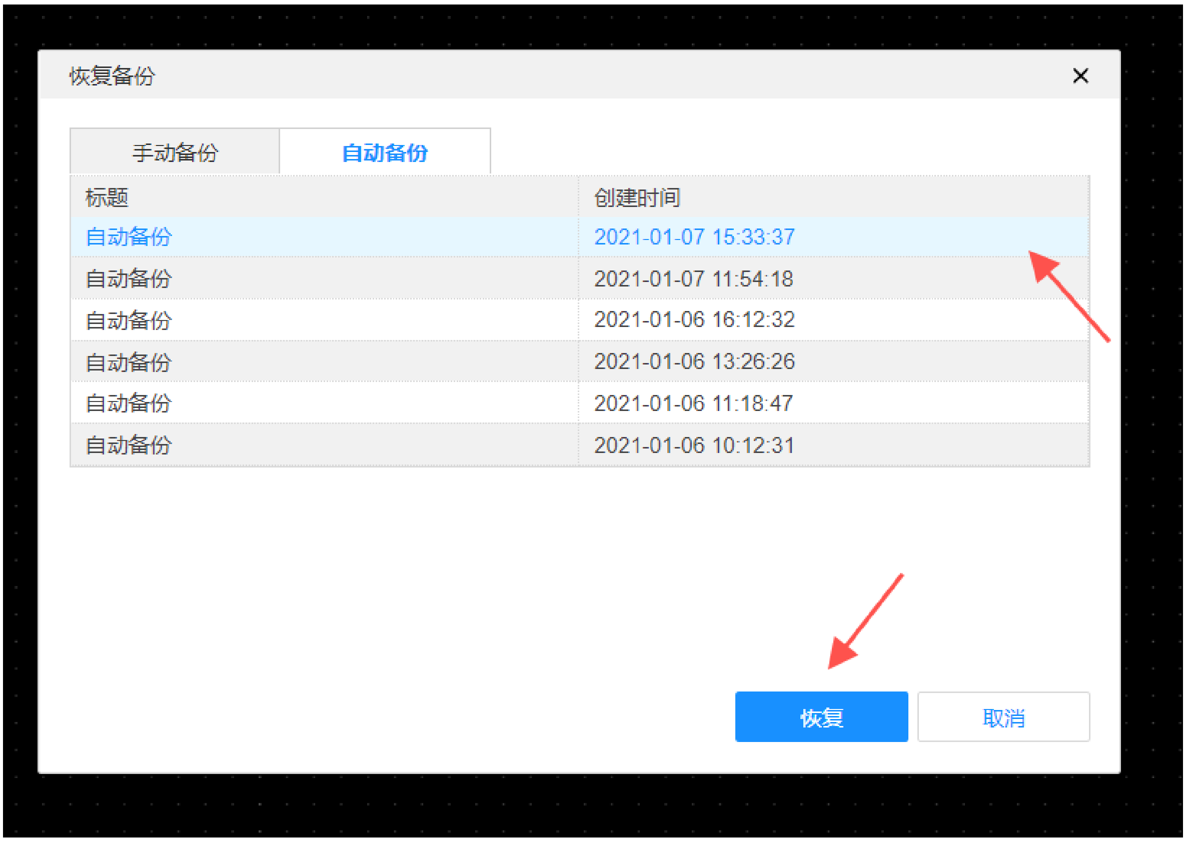
Create method:

- Ministry menu bar - File - Switch version

The pop-up window shows the project, creation time and description of automatic backup and manual backup.

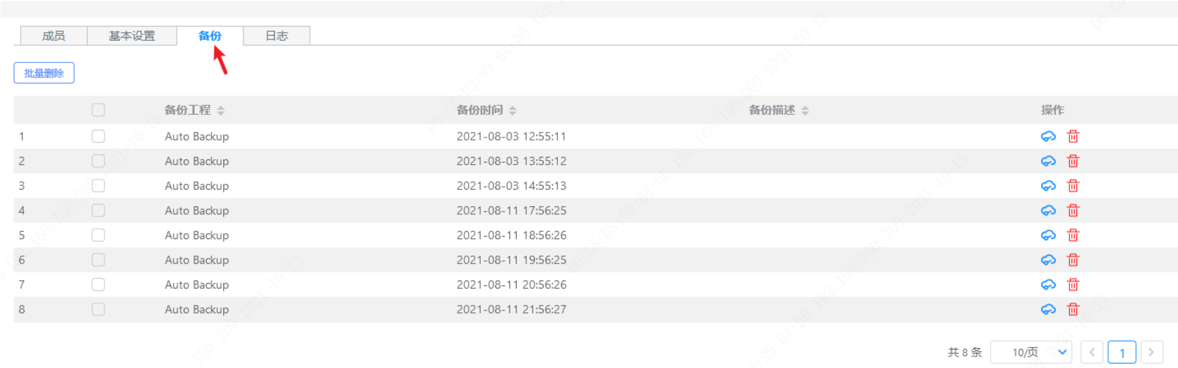


Select the project file to be restored and click Restore to re-import the backed-up project into the editor. The imported backed-up project will not conflict with the original project.



Backup management

Open the project details page in the workspace, and you can also see the backup list of the project:



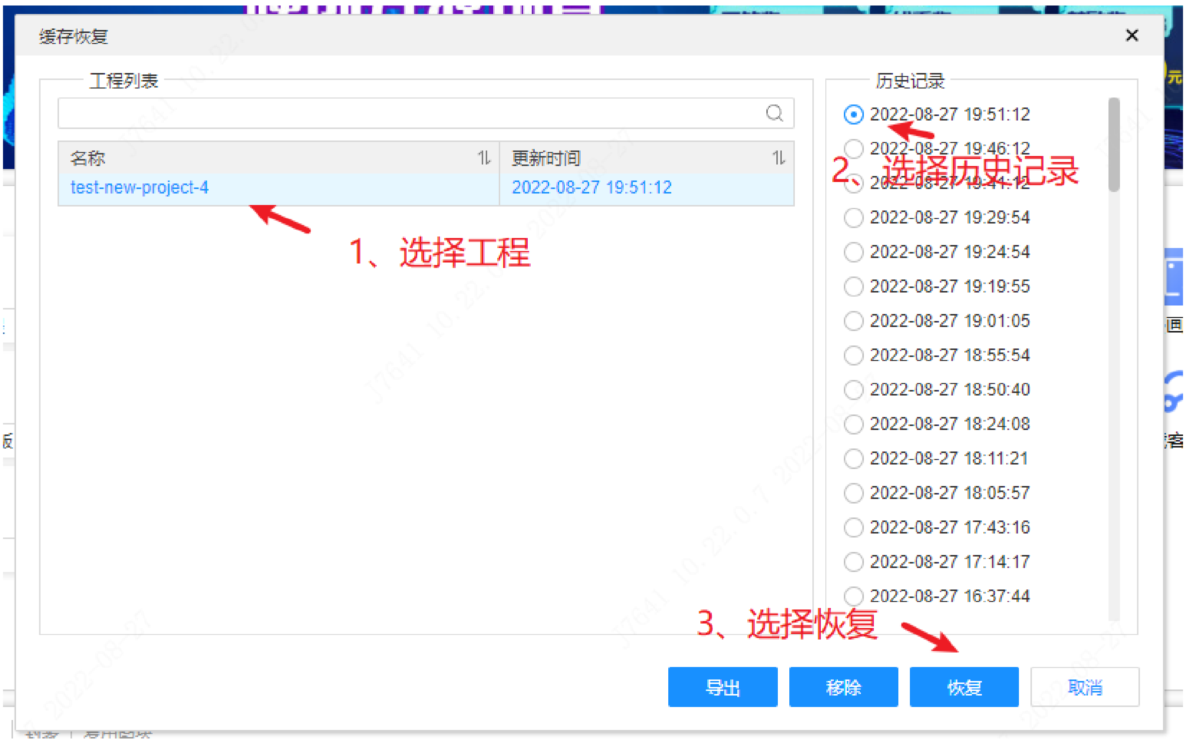
Delete operation is possible.

Cache Recovery

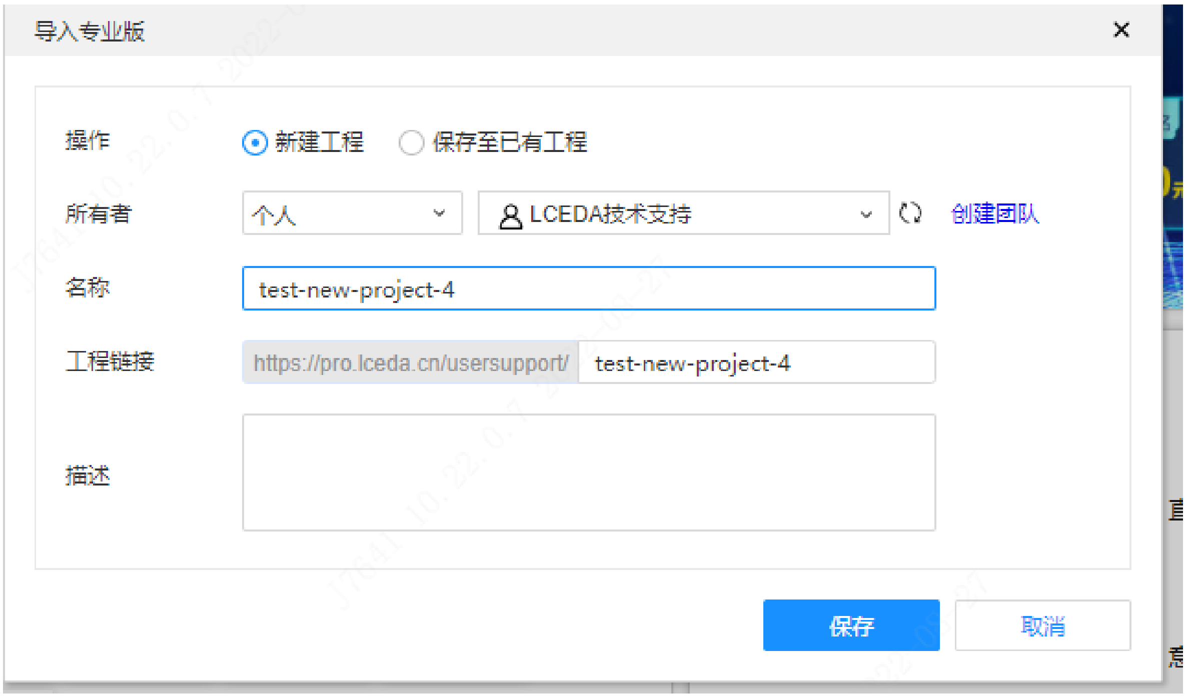
EasyEDA Professional Edition provides a cache recovery function, which can easily perform project recovery in case of accidental deletion or editor crash.

Entry: Top Menu - File - Cache Recovery.

Cache recovery can be turned on under the File menu after each document is opened.



After selecting Restore, the project logic will be imported, and you can select New Project to save.



You can also export the history to the local, which will be exported as a project archive. If retention is not required, history can be removed.

Notice:

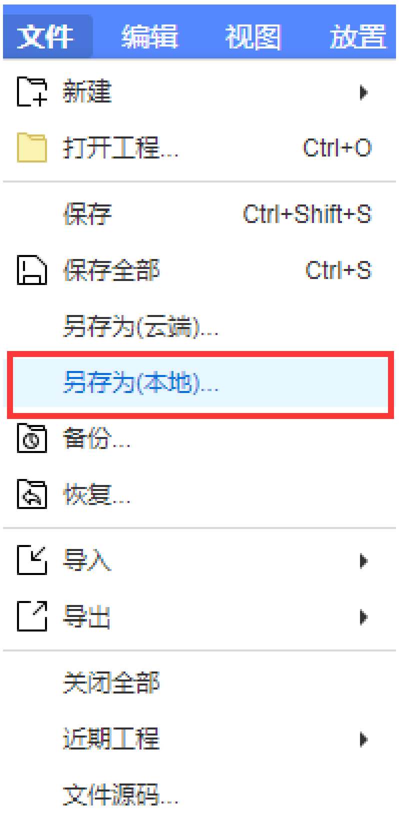
- Projects with editing rights will enter the cache recovery after opening, and there is no limit to the number; the history record of each project will only record the latest 50 times, and the excess will automatically remove the old records.
- Cache recovery is to cache the current project status to the local cache every 5 minutes. If the local cache data is cleared, the cache recovery will also be cleared.

Export Project to Local

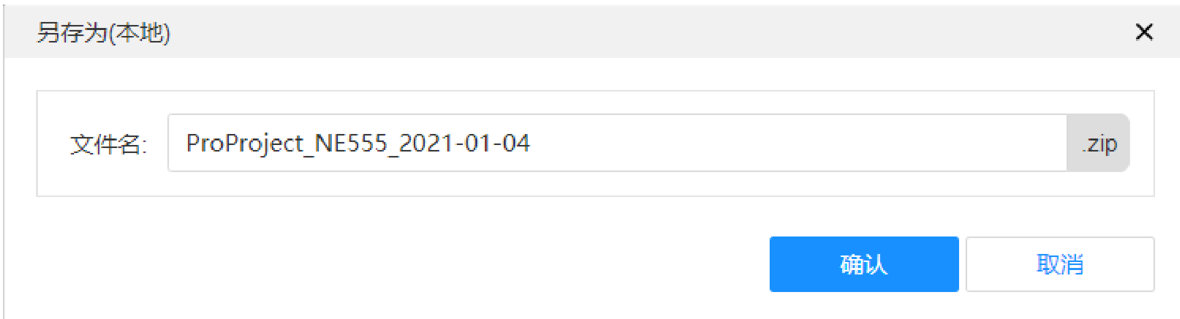
EasyEDA Pro supports saving project files to local files. The created project is saved in the cloud by default, and it needs to exist locally and needs to be operated by the user.

Export process:

- Top Menu - File - Save As (Local)



After clicking, the files in the project can be compressed to the local, and the compressed footprint includes the device library and footprint placed in the engineering schematic diagram.

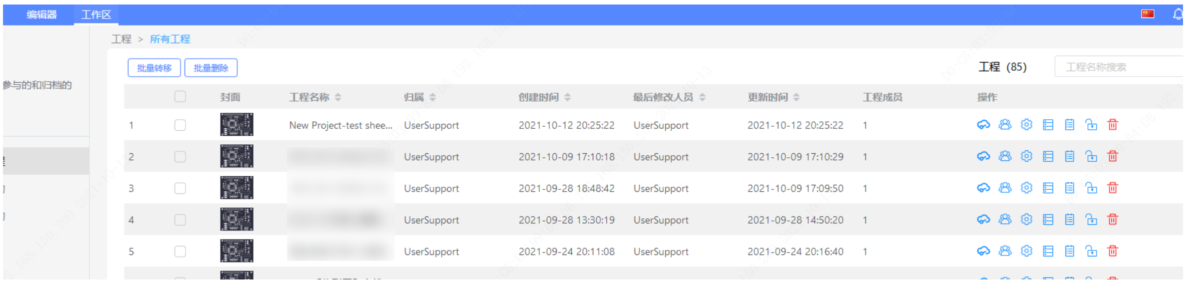


On the editor start page, you can import the project archive saved locally.

Modify Project Information

Open the project settings page in the workspace.

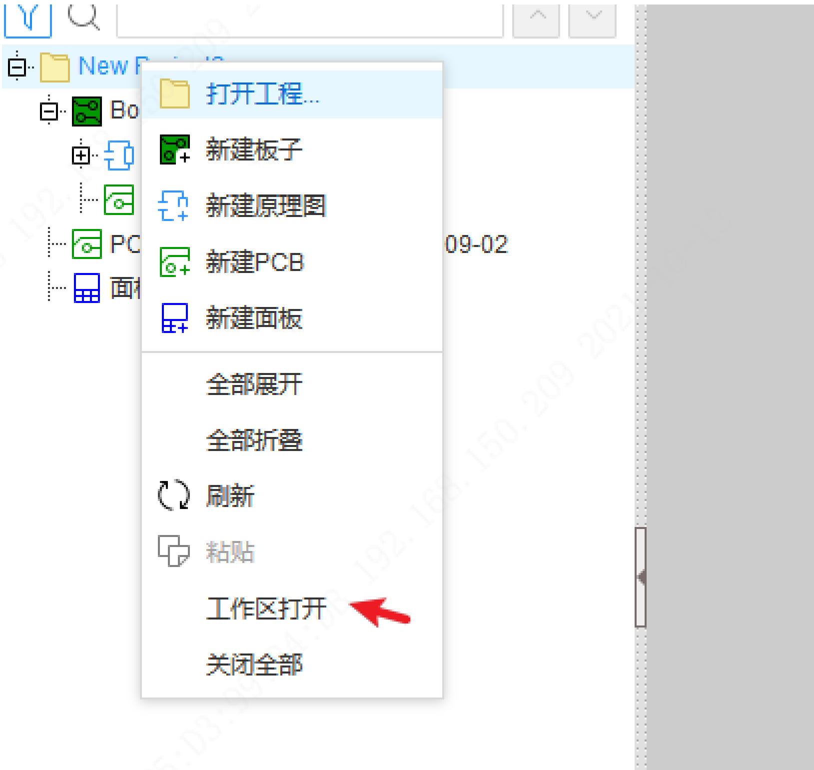
The entry is at the top of the start page - Workspace,



Bulk operations are supported.

or avatar dropdown:

Or after opening the project, the right-click menu of the project name:

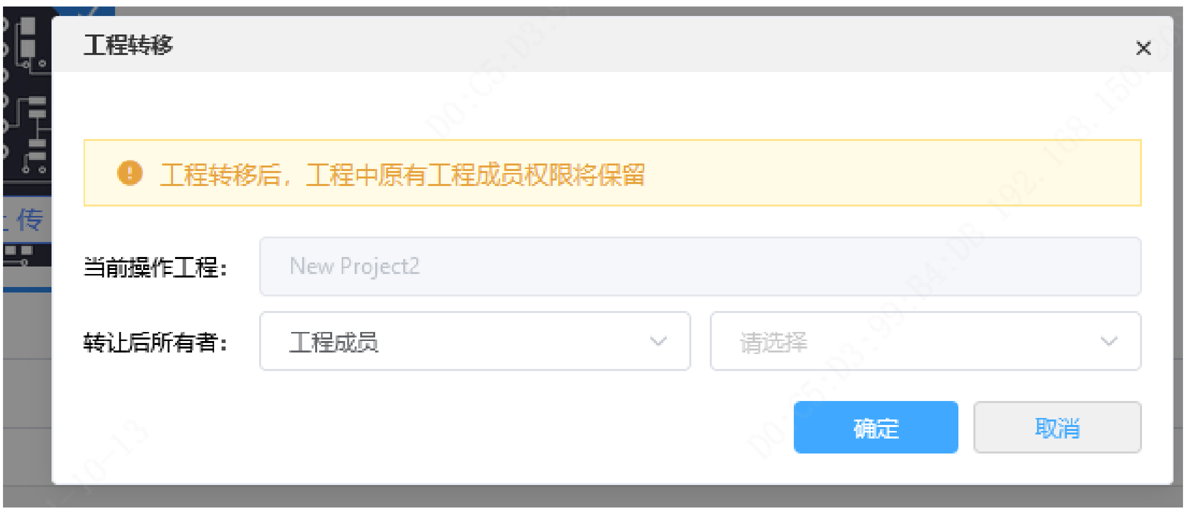


After opening, find the corresponding project setting entry:

Supports setting basic information, transferring, locking and deleting projects.

When the project is locked, they cannot do any editing and saving operations after opening the project, but can only view it.

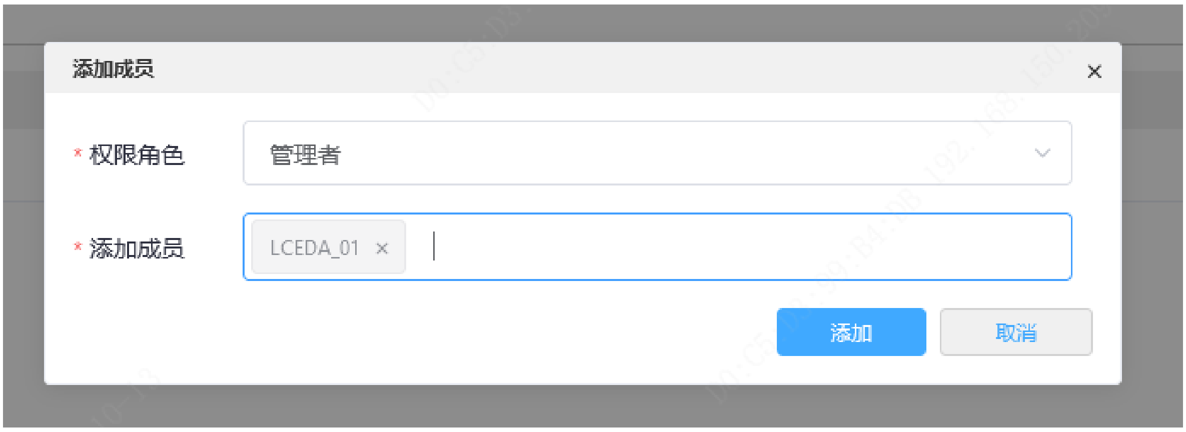
Transfer engineering support is transferred to engineering members, and to teams you join yourself.



Project Collaboration

Open the project settings page in the workspace, you can add project members and set the corresponding permissions, project members can read and write the project.

Add by entering the other party's username.



If you don't need the other party to participate, just remove them as a member.

Permission roles:

- **Manager**: You can edit project information, add or remove members, delete projects, delete documents, etc.
- **Developer**: You can create, edit and save documents.
- **Observer**: You can open and view the document as soon as possible. At present, the observer can export the project to the local and export the BOM and other operations.

Note: When the project is collaborating in a team, multiple people cannot edit the same document at the same time, and different documents need to be edited separately.

Project Log

On the project details page of the workspace, you can view the operation log of the project. Currently, only a few key operations are logged, and more fine-grained logging will be supported later.

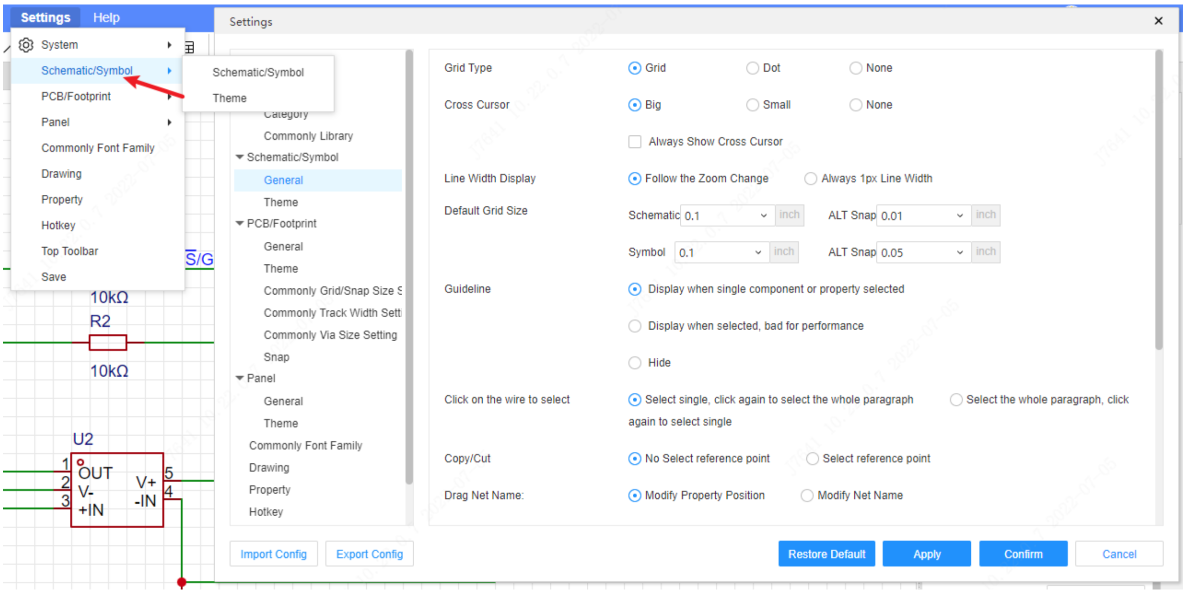
成员 基本设置 备份 日志

	工程内容	时间
1	工程New Project,UserSupport自动备份了工程	2021-08-11 21:56:27
2	工程New Project,UserSupport自动备份了工程	2021-08-11 20:56:26
3	工程New Project,UserSupport自动备份了工程	2021-08-11 19:56:25
4	工程New Project,UserSupport自动备份了工程	2021-08-11 18:56:26
5	工程New Project,UserSupport自动备份了工程	2021-08-11 17:56:25
6	工程New Project,UserSupport自动备份了工程	2021-08-03 14:55:13
7	工程New Project,UserSupport自动备份了工程	2021-08-03 13:55:12
8	工程New Project,UserSupport自动备份了工程	2021-08-03 12:55:11

Schematic Design

Schematic Settings

Operation steps: Top Menu - Settings - Schematic/Symbol

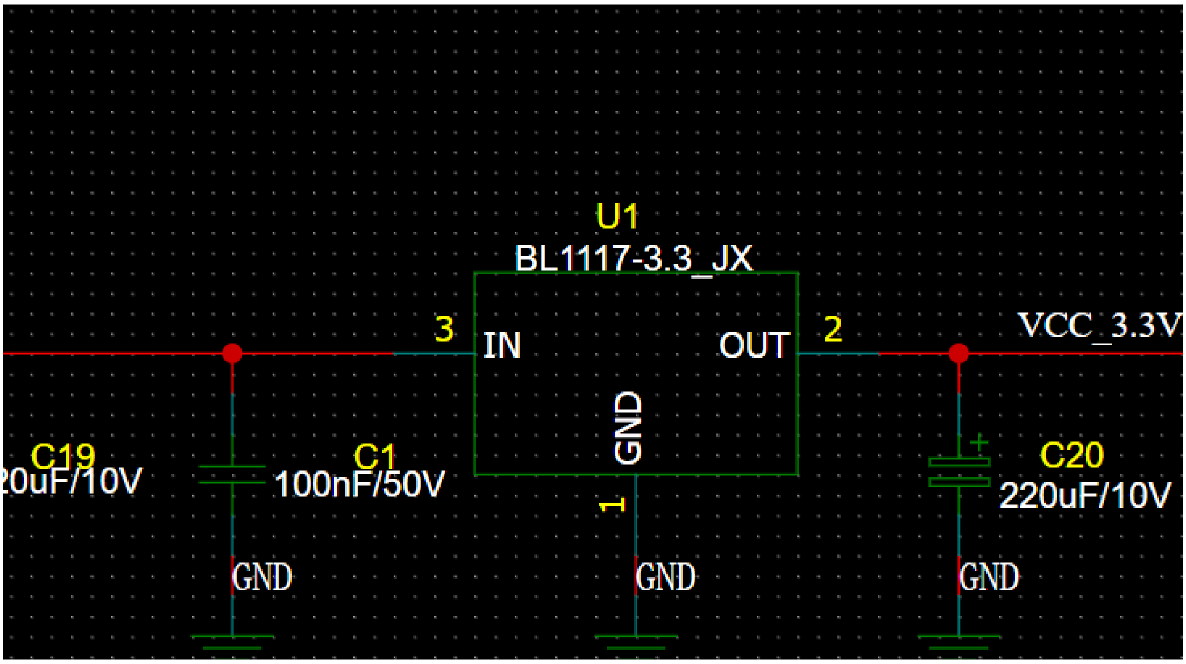


General

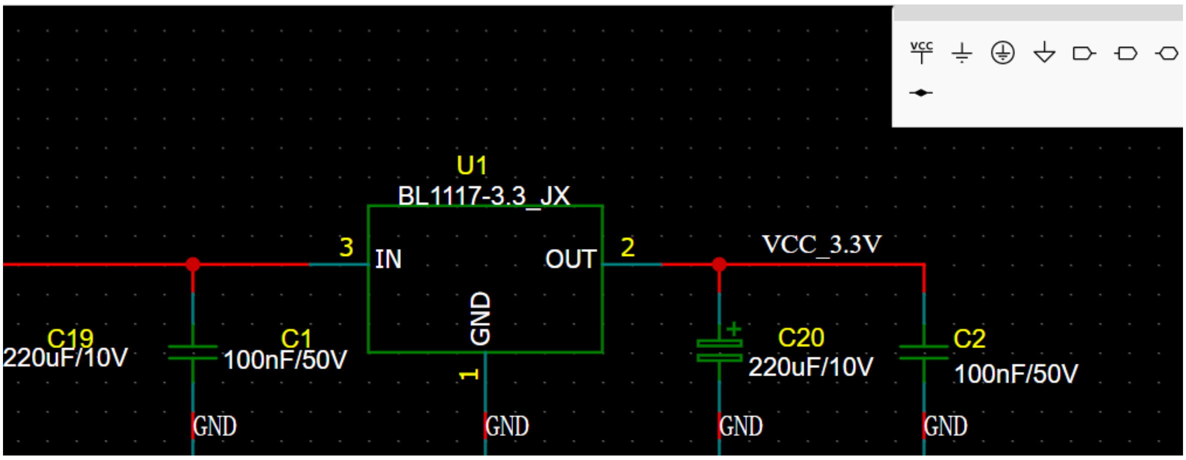
Grid Type: Set the default canvas. You can set **Dot**, **Grid**, **None**.

Cross Cursor: Set the cursor size of the schematic editor;

Line width display: Set the line width of the overall device line width;

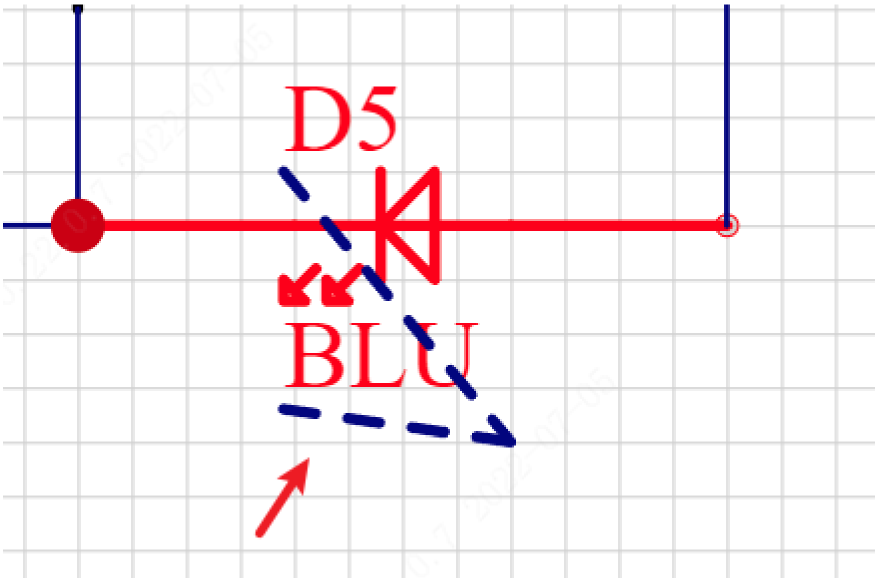


The line effect that follows zoom changes will become thicker.

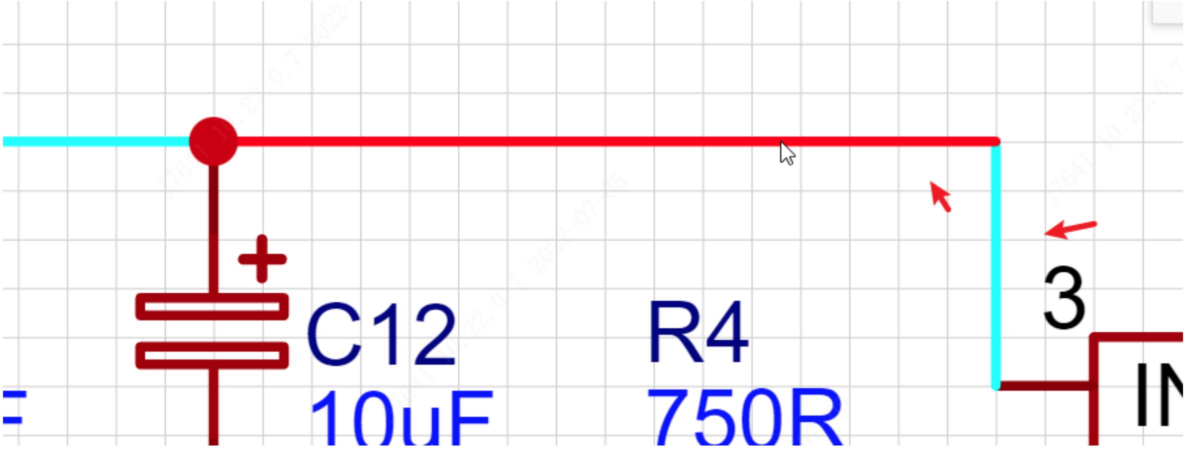


Default grid size: Set the grid size for opening or creating a schematic, which cannot be changed;

Guideline: Whether to display the indicator line between the component origin and component properties according to the setting.



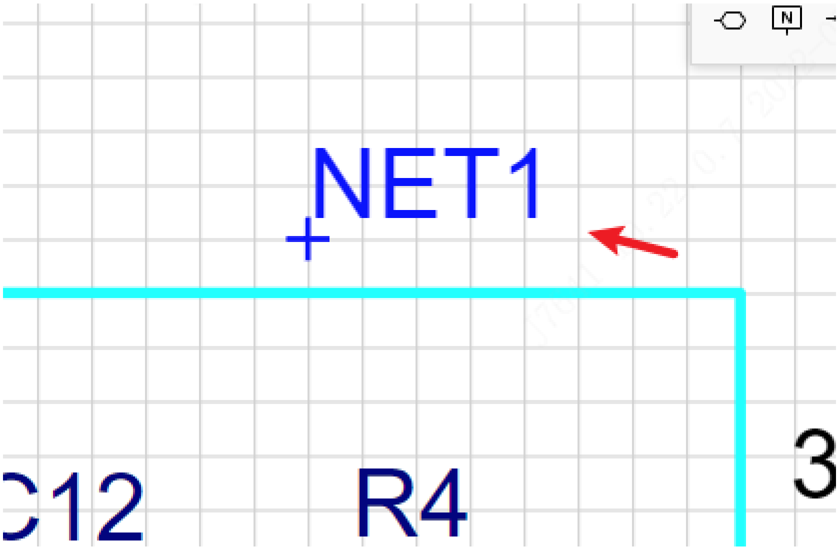
Click on the wire to select： According to your own usage habits, you can switch the selection range when you click the wire, select a single segment or select the entire segment.



Copy/Cut: Whether to select a reference point when copying/cutting.

Drag Net Name: How to handle when dragging a wire's net name off the wire.

- Adjust the attribute position: only move the position of the attribute name, without affecting the net name of the wire. The behavior before the professional version is similar to the behavior of the network tabs of PADS and Orcad.
- Modify net name: When the net name leaves the wire, the wire's net name will be cleared, similar to the Altium and Standard version's net label behavior.



Drag symbol, wire's following way: Set whether the wire moves with the component.

Other:

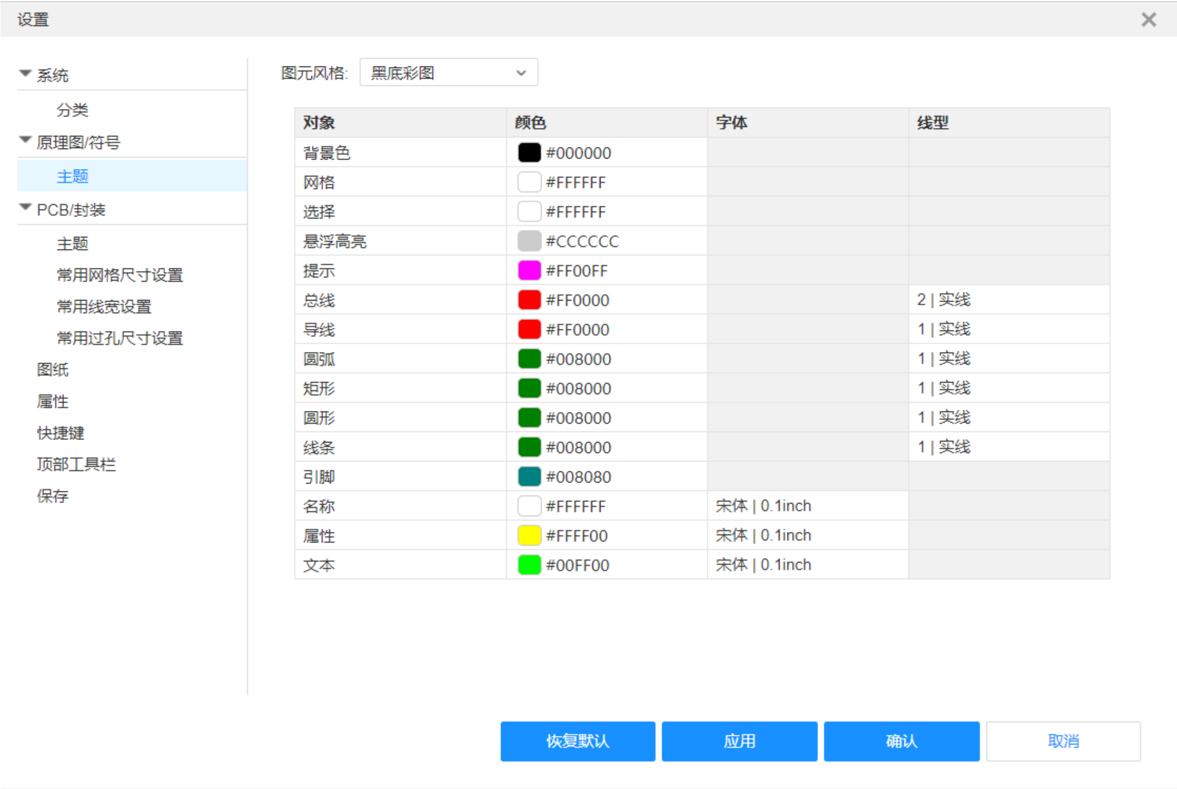
- Symbol Editor Display Scale Ruler
- Annotate designator automatically while placing or paste device(paste doesn't support multipart/subpart)
- Highlight entire network when hovering wire
- Adjust properties location automatically when components are rotated

Components placement for per page: At present, placing too many devices in the schematic diagram will cause lag, so the number detection is added. It is recommended to place less than 100 components on one page, and place other devices by creating paging.

Theme

Schematic theme settings

Here you can modify the background color of the schematic page or the color modification of the text.



Create Schematic/Page

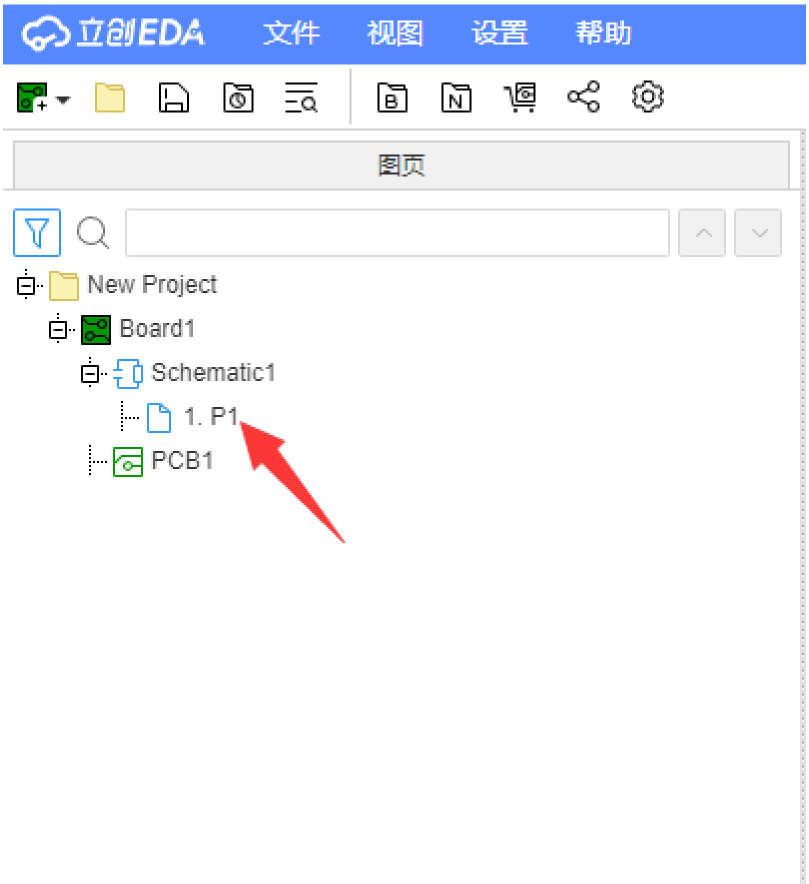
EasyEDA Pro supports multiple schematic diagrams of a project, even multi-page complex schematic design, there is no area limit, and multi-page schematics can be imported into the PCB interface at the same time.

Description: You need to create project files and boards to create drawing pages

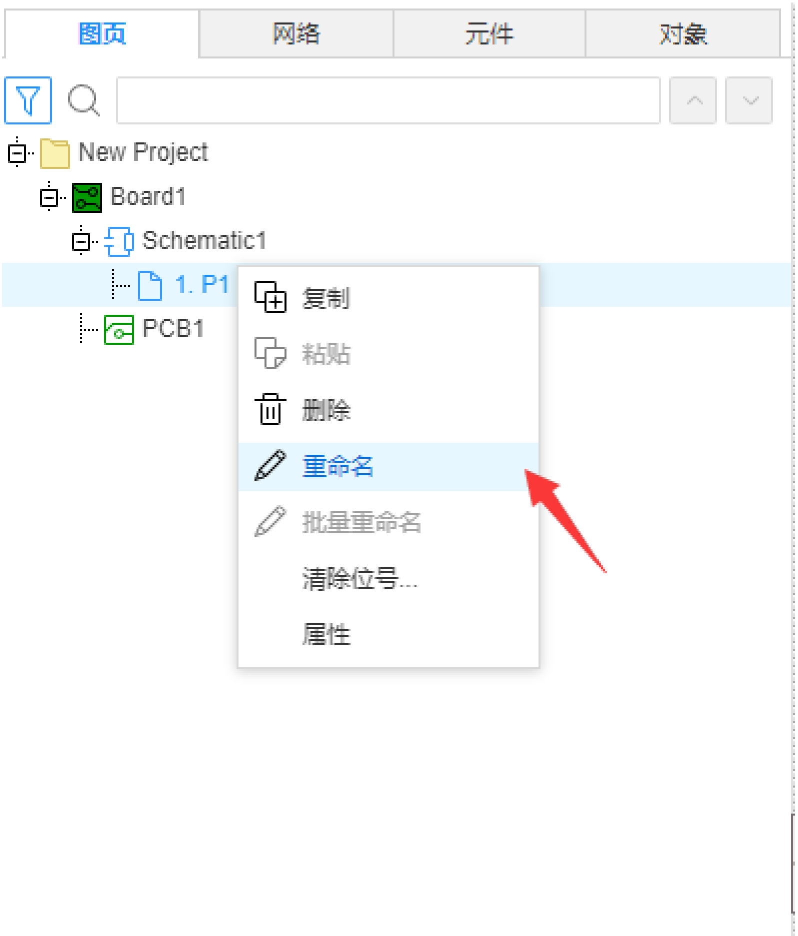
Steps:

- Top Menu - New - Page
- Right click on the project - New page

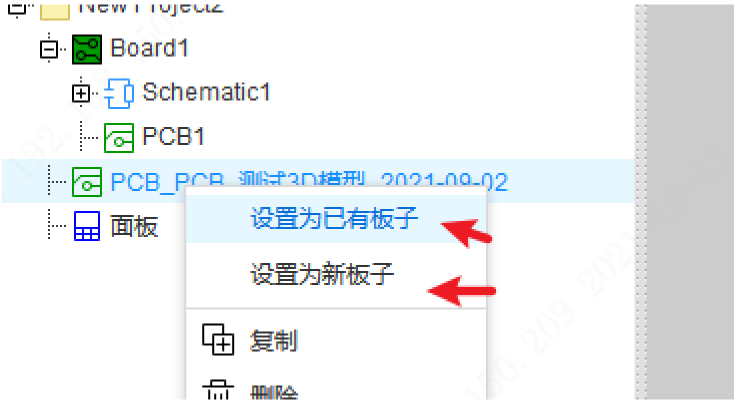
After the creation, the drawing page will be automatically added into your project file, and you can double-click to open the file.



If you want to modify your project information after creation, you can right-click on the board subfolder and select Rename to modify or delete it.



Note: The schematic diagram and the PCB must be under the same board before the schematic diagram can be transferred to the PCB, and the PCB can be imported and updated.
Schematic and PCB right-click can switch to different boards, or create new boards for association.

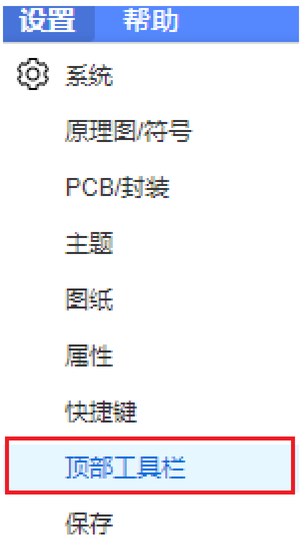


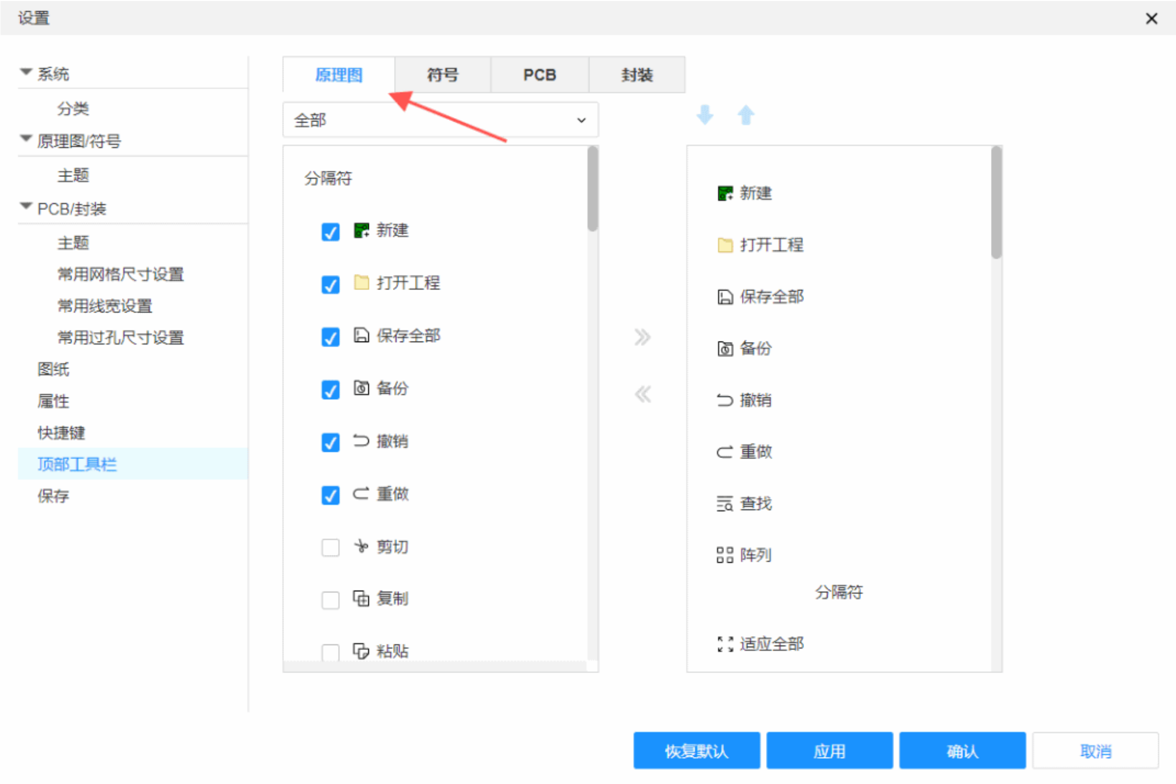
Top Panel - Toolbar

The function of the toolbar at the top of the schematic is similar to that of the PCB, and it is used to place some commonly used shortcuts.

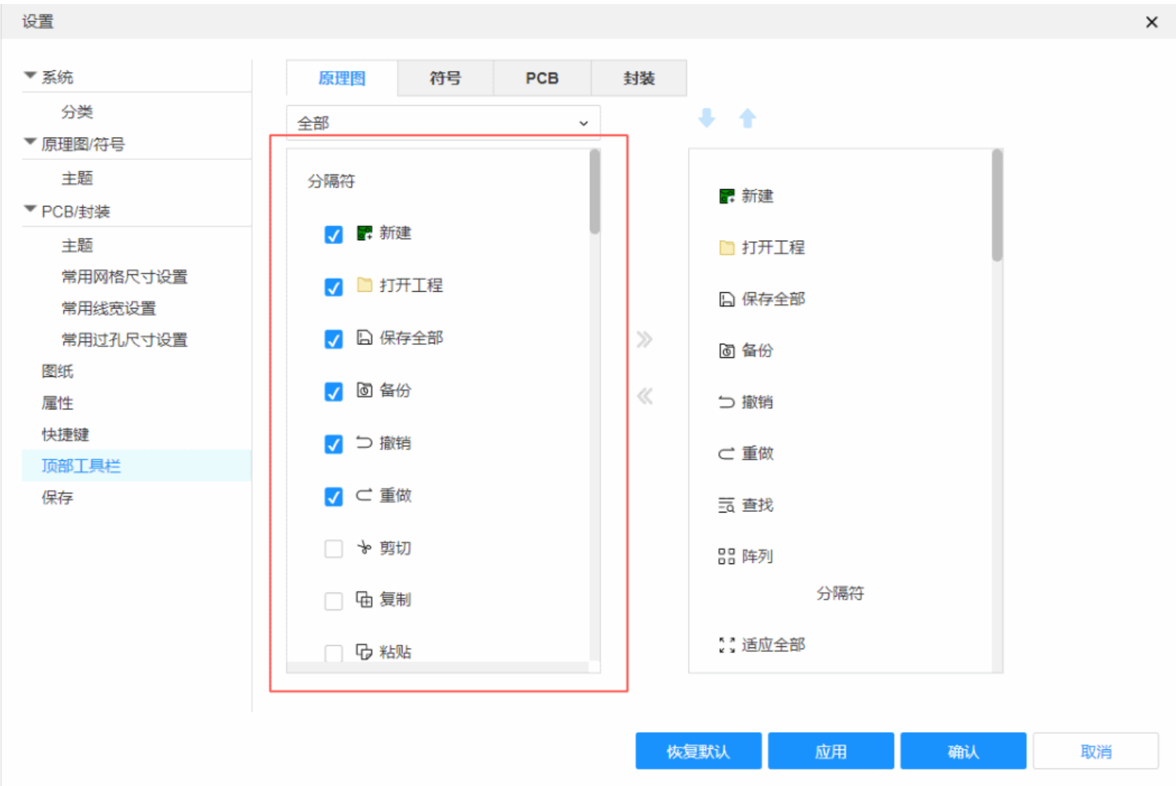
Modify the top toolbar

Top Menu - Settings - Top Toolbar - Select Schematic

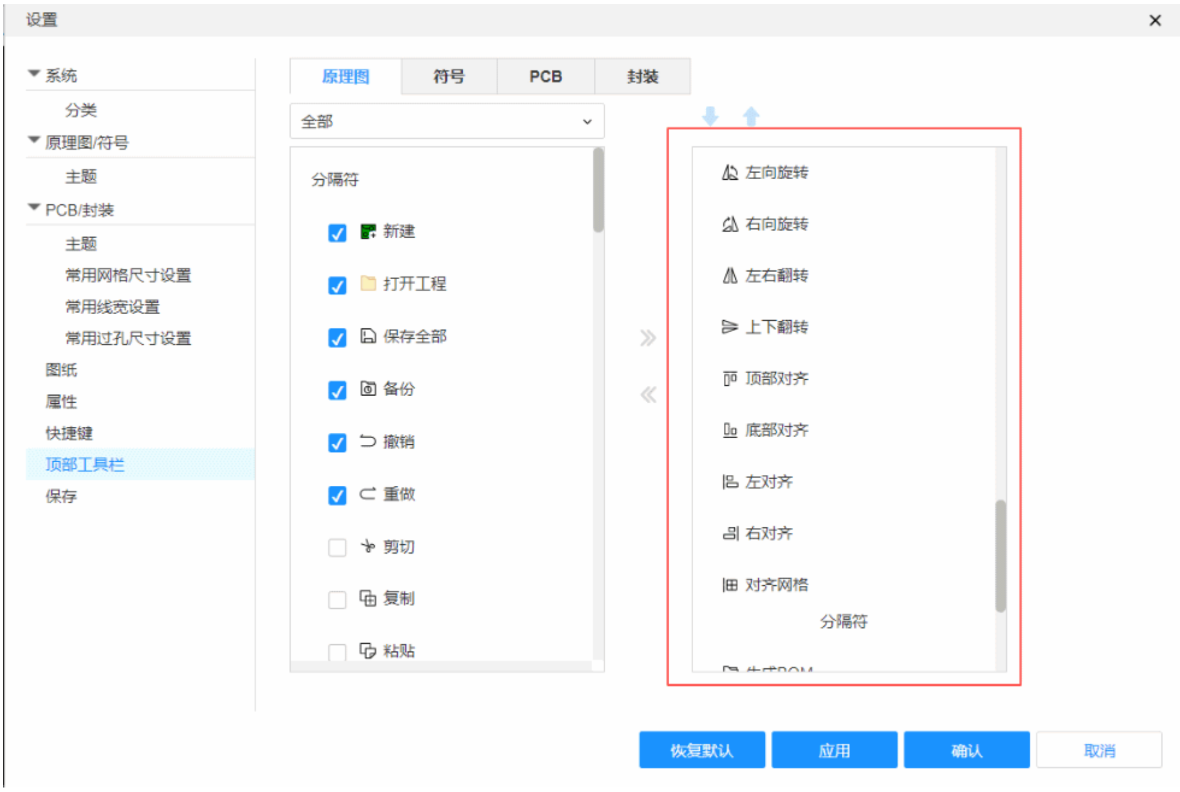




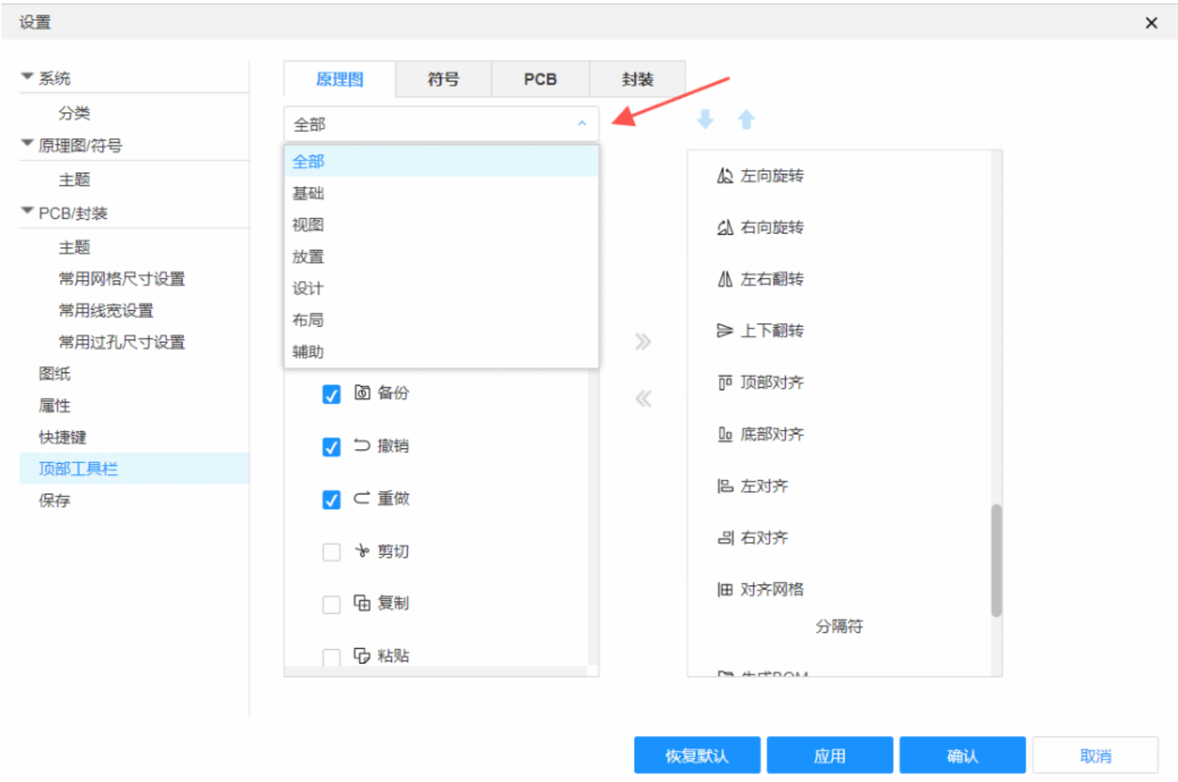
On the left side of the pop-up window are the shortcut functions that can be placed on the top toolbar. The checked ones are already in the top toolbar.



On the right are the shortcuts already placed in the top toolbar

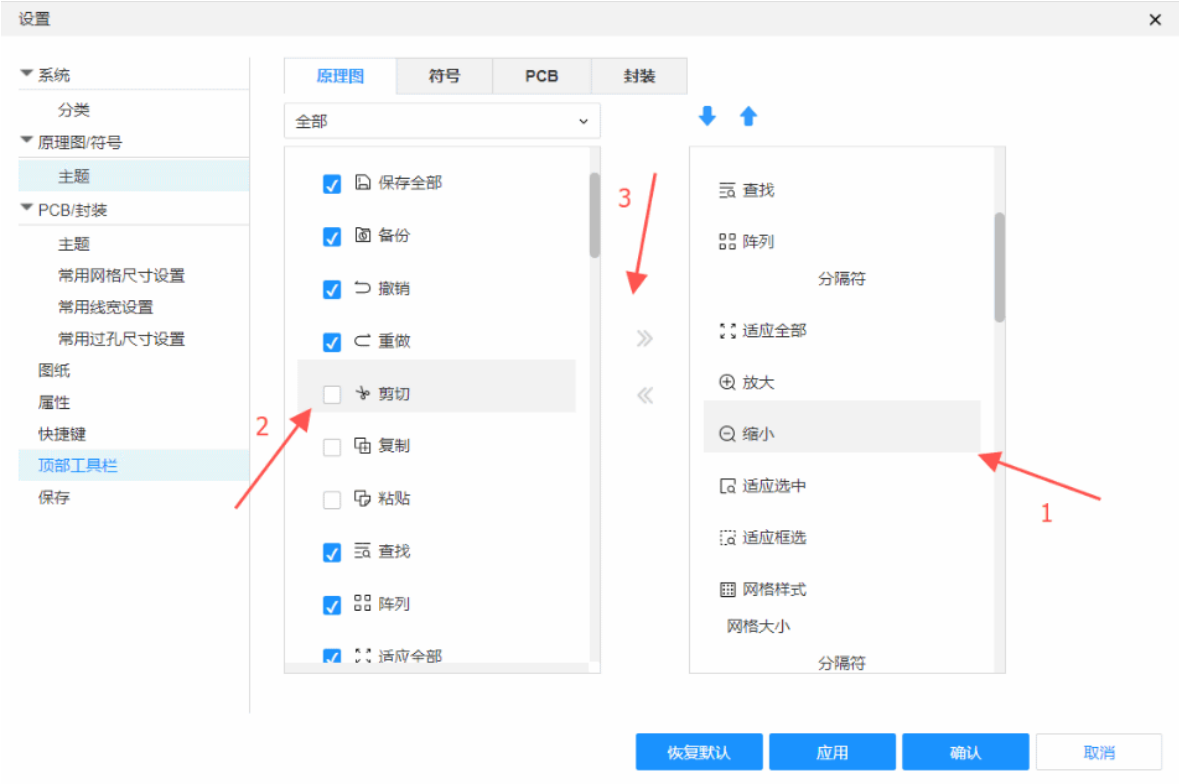


The top filter box of the pop-up window is to filter the function modules



Add shortcut

Add a shortcut to enter the top toolbar, the operation steps: first select the location to add the shortcut on the right, and then select the shortcut to be added, click the arrow in the middle or check Add to enter the top in the shortcut on the left side of the pop-up window Toolbar, click OK to save the current configuration, and the addition is complete.



Bottom Panel - Library

Use the shortcut key `S` to call up the bottom pop-up window and select the device path. Among the device libraries, there are the LCSC Mall library and the EasyEDA Pro system library.

Note: The system base library does not support customization!

Device library

The device library is a device library that includes symbols, footprints, 3D models, and pictures. The device library includes the system's basic library, personal device library, and team device library.

Use filters to quickly find the parts you want, for example, enter 0603 to quickly search for parts related to 0603.

You can double-click the device list or select it and click the "Place" button to place it. The bottom panel will be automatically collapsed, and it will automatically open again when you cancel the Place.

For the specific function of the device list, please check: "Device Library - Device List" chapter .

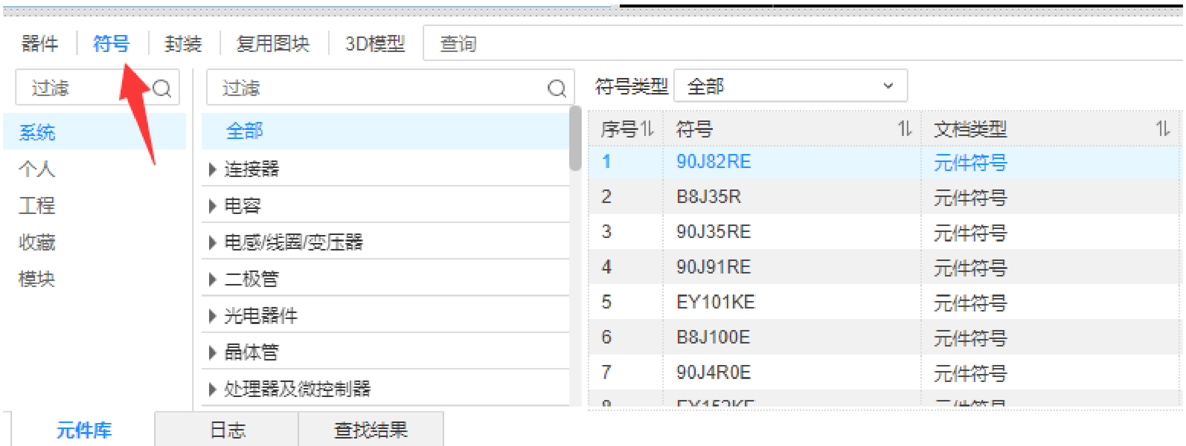
Place can also be opened via the Device dialog, SHIFT + F.

Note: The independent dialog box for placing a device can only place the device, but cannot edit the device.

Symbol library

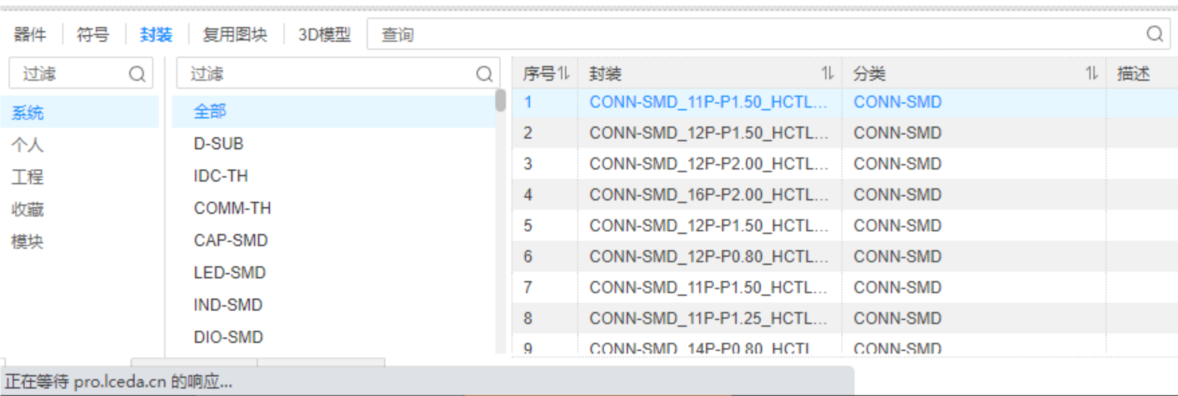
The symbol library only has one symbol, there is no footprint and 3D model, the symbol of the symbol library cannot be placed in the canvas of the schematic diagram, and the device needs to be bound to be allowed to be placed on the canvas. Symbol libraries include system libraries, personal libraries, and team symbol libraries.

Note: The symbol library of the system is not editable!



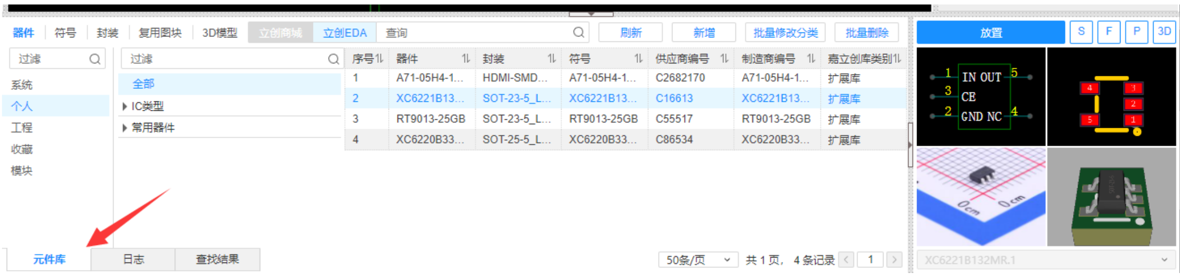
footprint library

The footprint library page has only one footprint without symbols and 3D models. The footprint cannot be placed in the PCB alone. footprint libraries also include system libraries, personal libraries, and team symbol libraries.



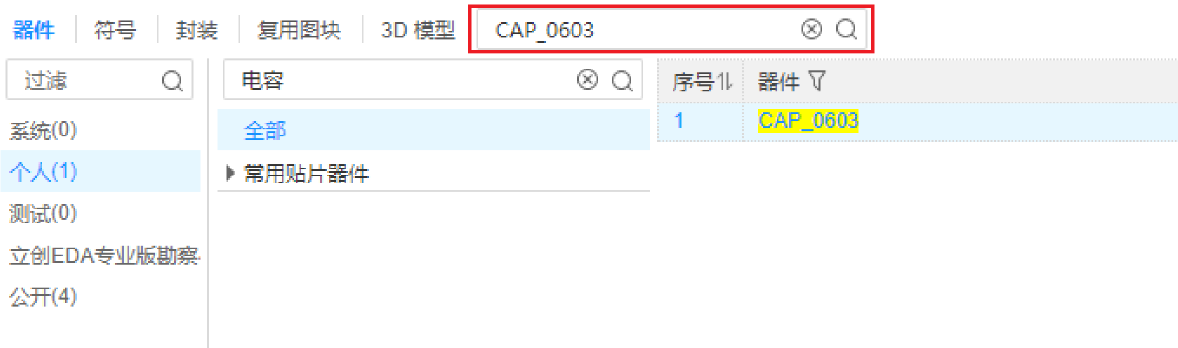
The component library is in the bottom panel of the PCB interface. The component library in the component library can directly place components in the PCB interface, but symbols, footprints and multiplexing blocks cannot be placed.

The component library includes system library, personal library, collection library and component library for joining the team. The component library also includes device library, symbol library, footprint library, and multiplexing blocks.



Device Search

The search at the top of the library is a global search that searches the entire library



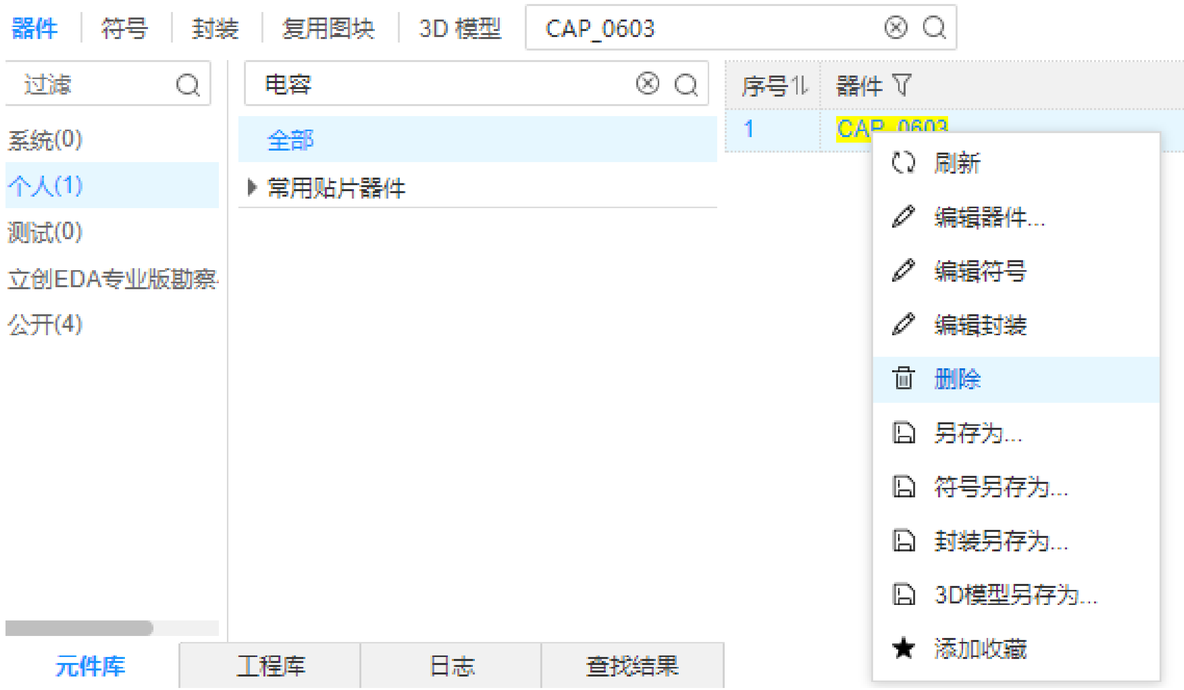
And the search box below is to search the system or personal device library by category

edit element

Select any device in the component library and click the right mouse button to modify the device.

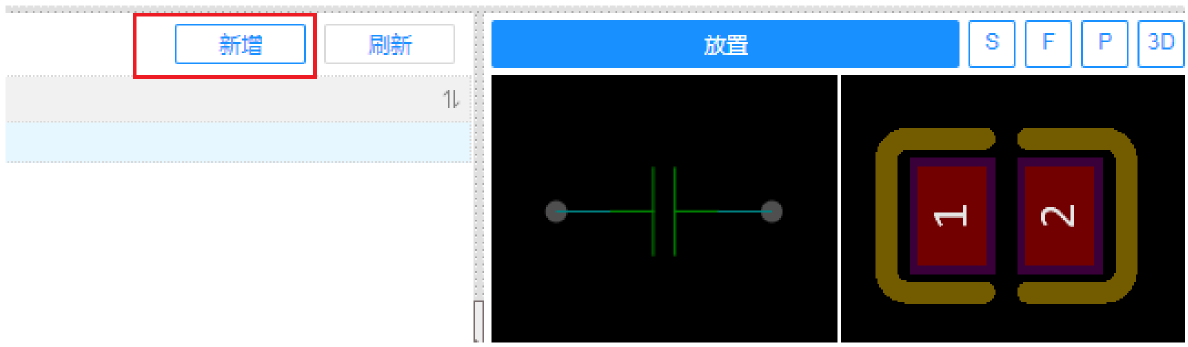
illustrate:

- System library does not support modification!



new device

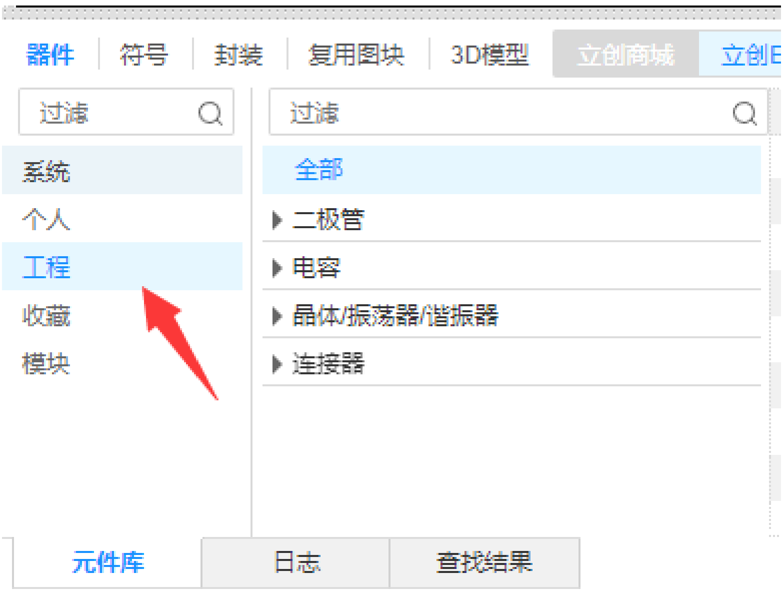
Click the new button on the upper right in the component library to quickly create a device. The symbols and packaging operations are the same.



Project Library

The project library is all the device libraries currently placed in the project, including symbols, footprints and multiplexed blocks. Adding devices and special symbols to the project can be displayed in the project library. The device project library deleted in the project will not be displayed. Following deletion, it will be kept in the project library, and a historical record will be made to record the devices placed in this project. At the same time, you can click to use or modify it again. The modified devices in the project library can only be applied to the current project. The component library has no effect, but the components that have been placed in the project cannot be deleted from the project library.

The project library of the PCB is the same as the schematic project library. The project library is all the device libraries currently placed in the project, including symbols, footprints and multiplexing blocks. Adding devices and special symbols to the project can be displayed in the project library. , the device project library deleted in the project will not be deleted, it will be kept in the project library, make a historical record, record the devices placed in this project, and can also click to use or modify again, and modify it in the project library The device can only be used in the current project and has no effect on the component library, but the device that has been placed in the project cannot be deleted from the project library.



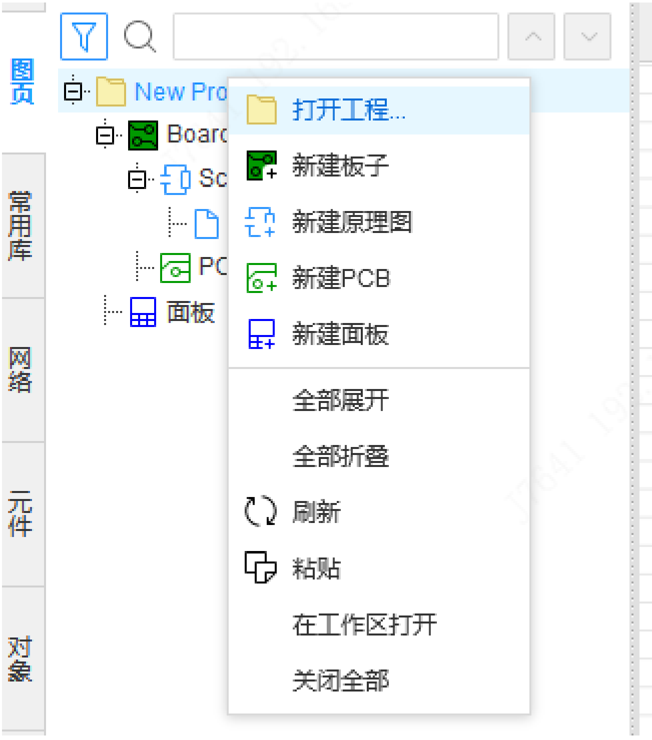
Left Panel - Page

The created project and drawing page can be viewed in the left panel - drawing page, double-click to open the created file and edit it.

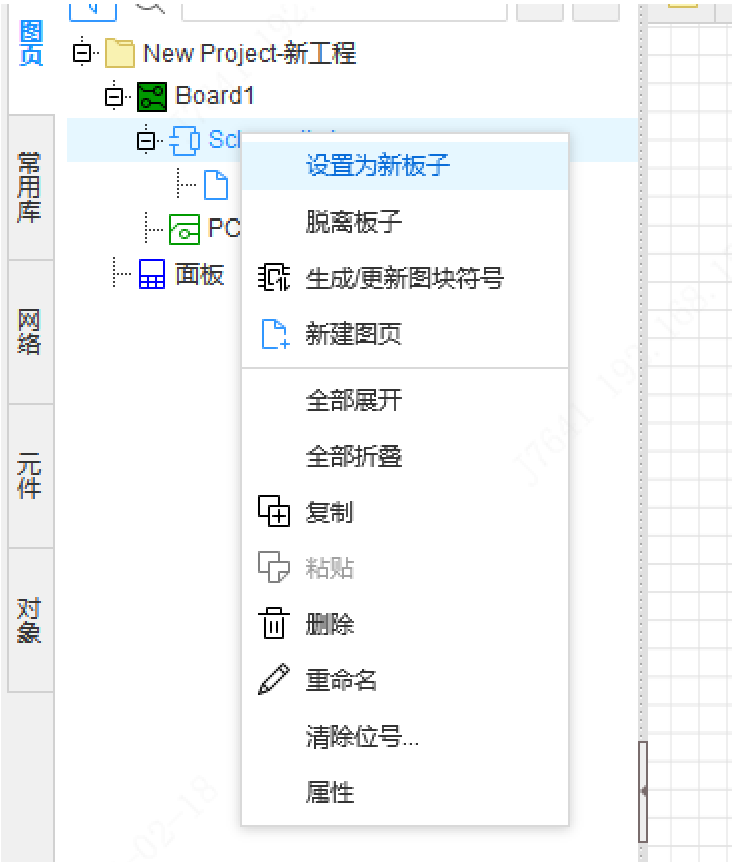
Figure 2

Select the file, click the right mouse button, and edit the created project or drawing page.

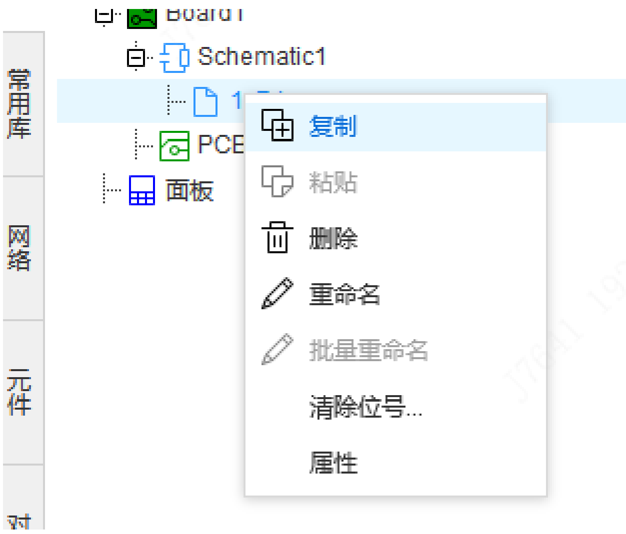
Right click on the project name:



Schematic right click:



Right click on the page:

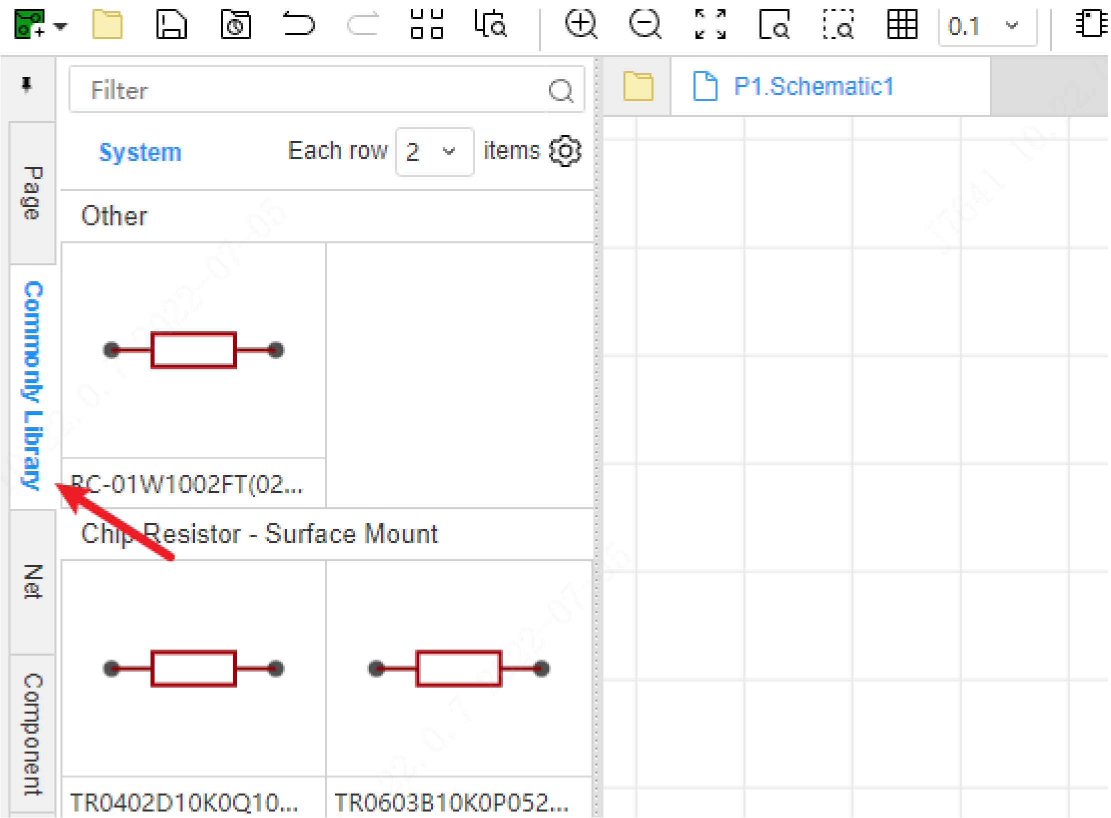


Menu Description:

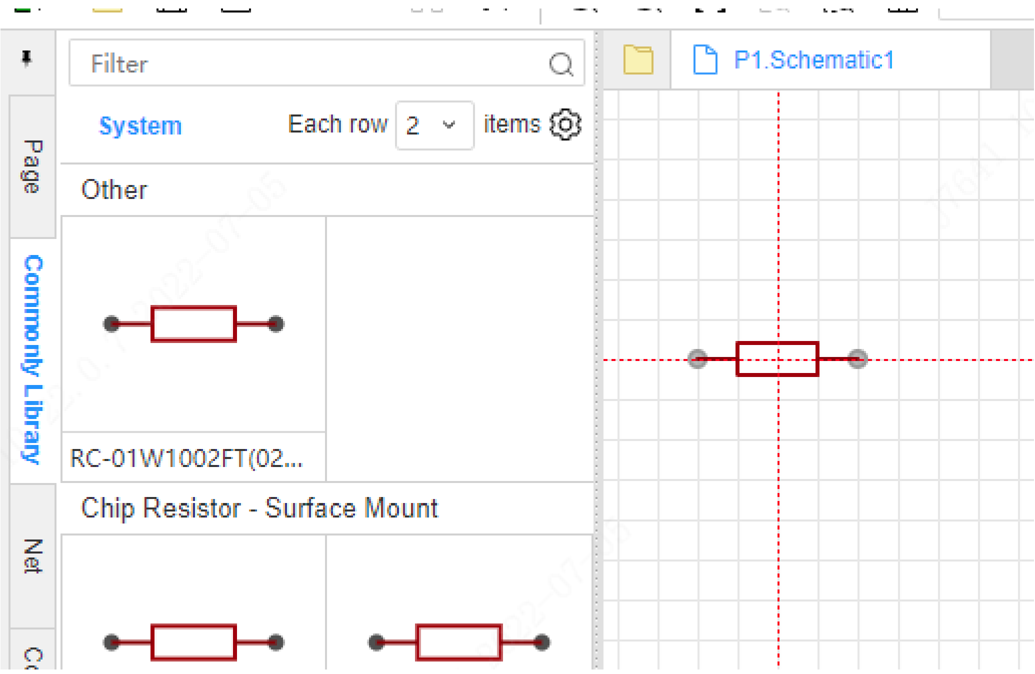
- Set as new board: Set the current schematic to the new board.
- Disconnect from the board: Take the schematic out of the board and become a free schematic.
- Generate/Update Block Symbols: Generate multiplexed block symbols based on the current schematic, or update block symbols. When generating block symbols, the current schematic diagram will become a hierarchical diagram template, which can be reused multiple times.
- Copy/Paste: Supports copying and pasting of schematics. You can also copy and paste documents across projects. The project needs to be opened in the same browser window.
- Rename: Modify the file name.
- Clear Tag: Clear the tag of the current schematic.
- Attributes: Display the corresponding attributes according to the right attribute panel such as the clicked page.

Left Panel - Commonly Library

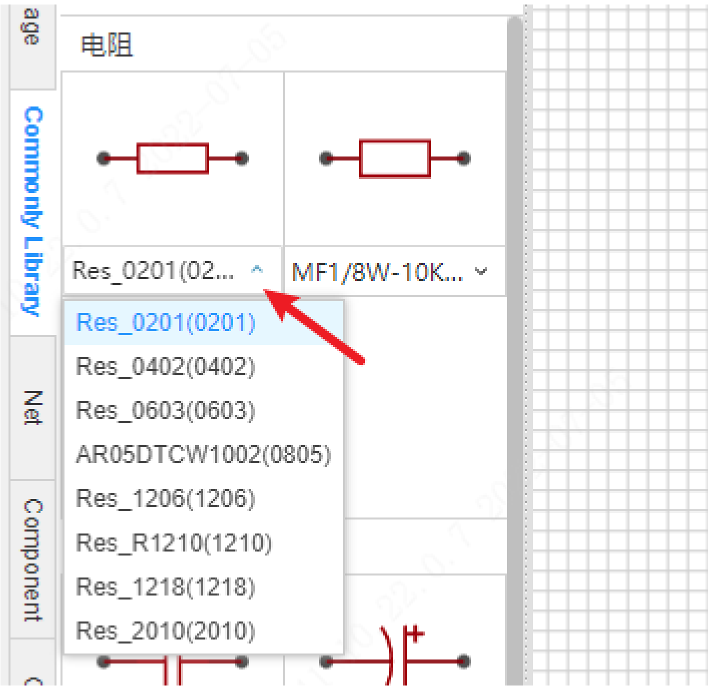
After opening the schematic diagram, you can see a common library Tab on the left panel, which supports displaying some common libraries built in the system, and also supports custom common libraries.



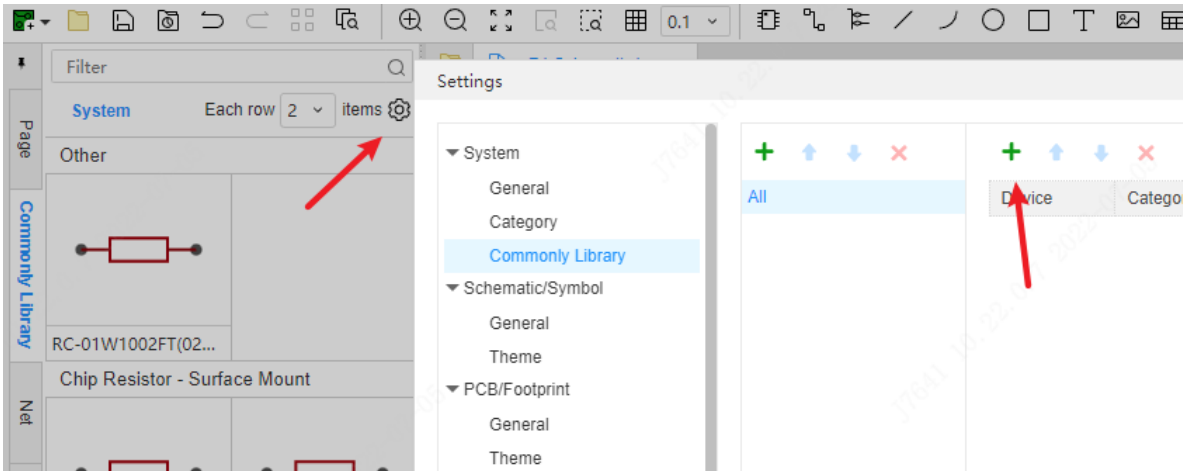
How to place: Click the component preview of the common library, move the mouse to the canvas, you can



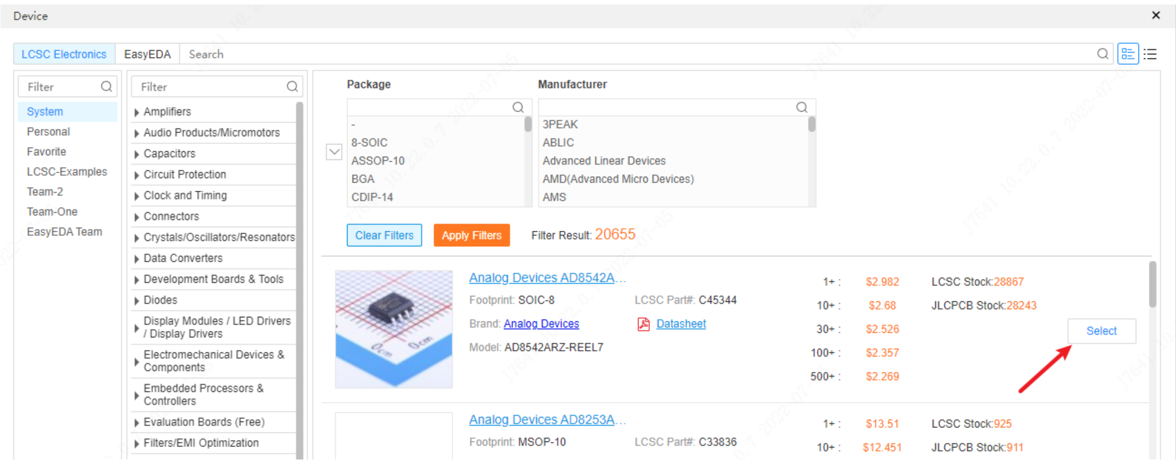
Switch component model: Click the drop-down of the component name to switch between different models.



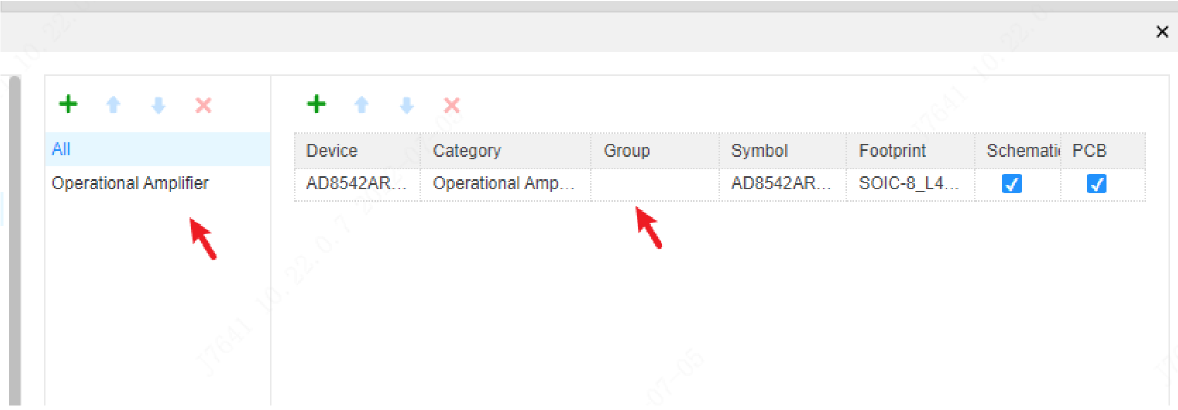
Custom common library: Click the gear icon in the upper right corner to open the settings dialog box, click the plus icon to add custom common library.



In the device dialog box, select the device to add to the common library.



You can double-click to modify the category name on the left, and the grouping of the basic library. The category is displayed in the category display of the common library, and the grouping is used to aggregate the devices under the same category. A drop-down list of devices appears under a thumbnail.



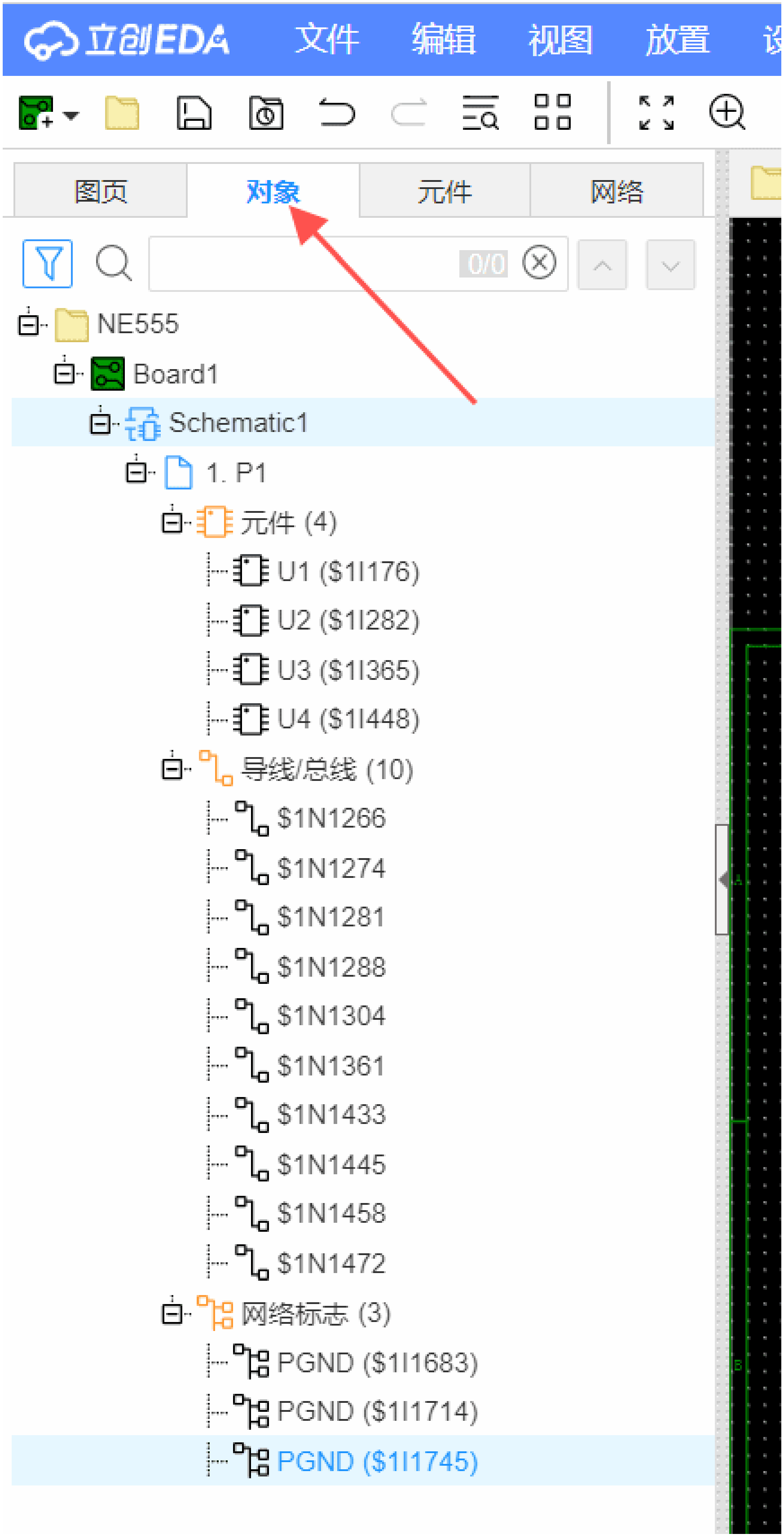
The Schematic and PCB tick box behind the device indicates whether the device is displayed in the Schematic or PCB common library.

Note: When the editor is opened for the first time, the thumbnails of the commonly used libraries will be automatically generated. When the computer performance is poor, the thumbnails of the commonly used libraries will be displayed slowly.

Left Panel - Object

Devices, wires, special symbols, and some graphics placed in the schematic can be viewed in - Objects in the left panel.

In the left panel, click a wire or a special symbol to highlight the selected element, and double-click to jump to the position of the current device in the schematic and highlight it.

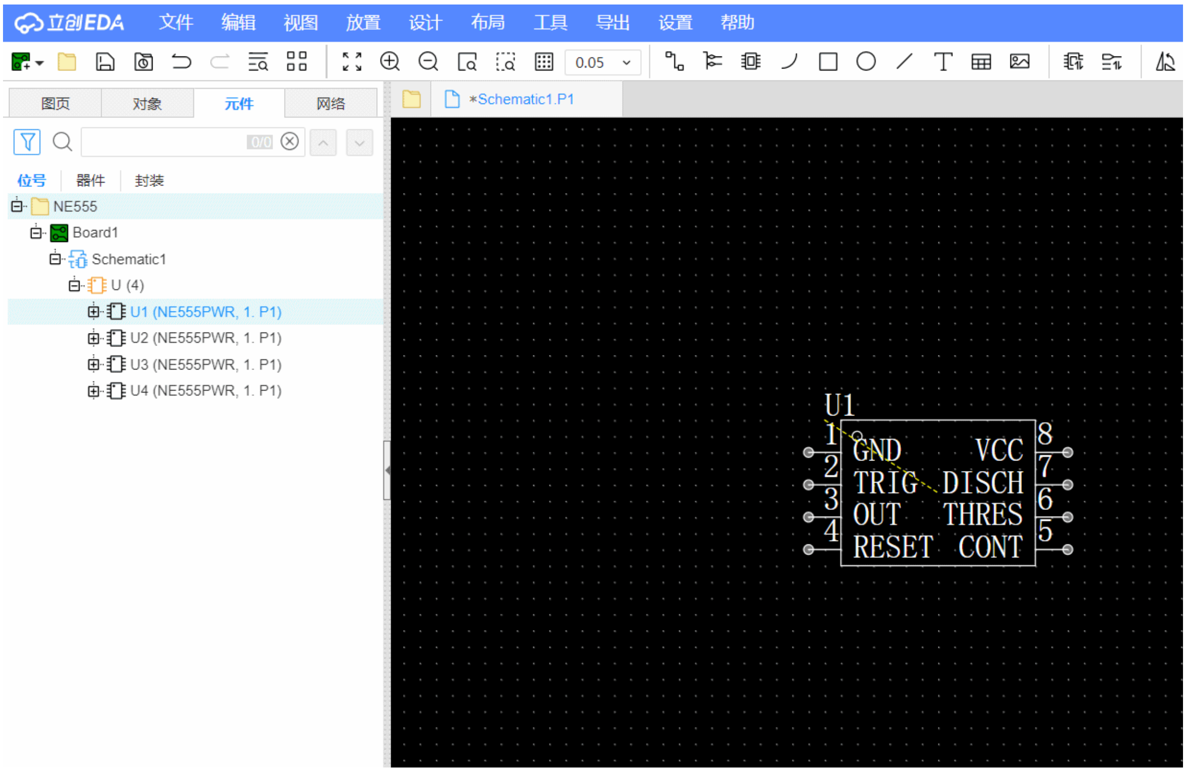


Left Panel - Components

The left panel of the Schematic Editor interface functions similarly to the left panel of the PCB interface and is used to display device information and quantities placed on the schematic.

image-20210107170901344

Click on the corresponding device to jump to the schematic and highlight it.



Left Panel - Net

The left panel nets in the schematic is to show the nets and the number of nets connected in the schematic, the pins connected in the nets.

Click to highlight the net and the pins connecting the net in the schematic.

image-20210107171650365

Right Panel - Property

After selecting a part, you can view or modify its property parameters in the property panel on the right.

Component Properties: You can modify the names and numbers of components, and set whether they are visible or not. Device information can also be modified. Please do not set the number in Chinese, the footprint number in the PCB does not support Chinese.

Here you can modify the component's supplier, supplier number, manufacturer, manufacturer number, footprint, etc.

You can also batch-select and then modify the properties in batches in the properties panel on the right.

image-20201216163341373

The property name and property value can be checked and displayed in the canvas.

Add custom properties: When you select a component or other primitives, you can add parameters to it,

If you don't need components to be in BOM or converted to PCB, you can set "Add to BOM" and "Convert to PCB" to No in the properties. When Convert to PCB is set to No, the device symbol will not be displayed in the Footprint Manager.

image-20201216163507688

When selecting the overall circuit, you can individually select the object property values to be modified in the right panel - Properties.

image-20210108101851944

Right Panel - Filter

filter

When filtering other elements of the device you want to select, if you check it, you can select it in the schematic, and if you uncheck it, you cannot select it in the schematic.

Figure 1

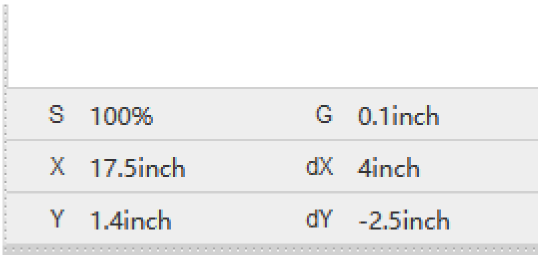
Drawing properties: You can select the property names and property values of drawings such as A3.

Pin Pair: The wire between two pins will be selected

Net: All wires with the same name will be selected based on their name

Right Panel - Status

After opening the document, the lower right area of the right panel displays the current canvas and cursor status.

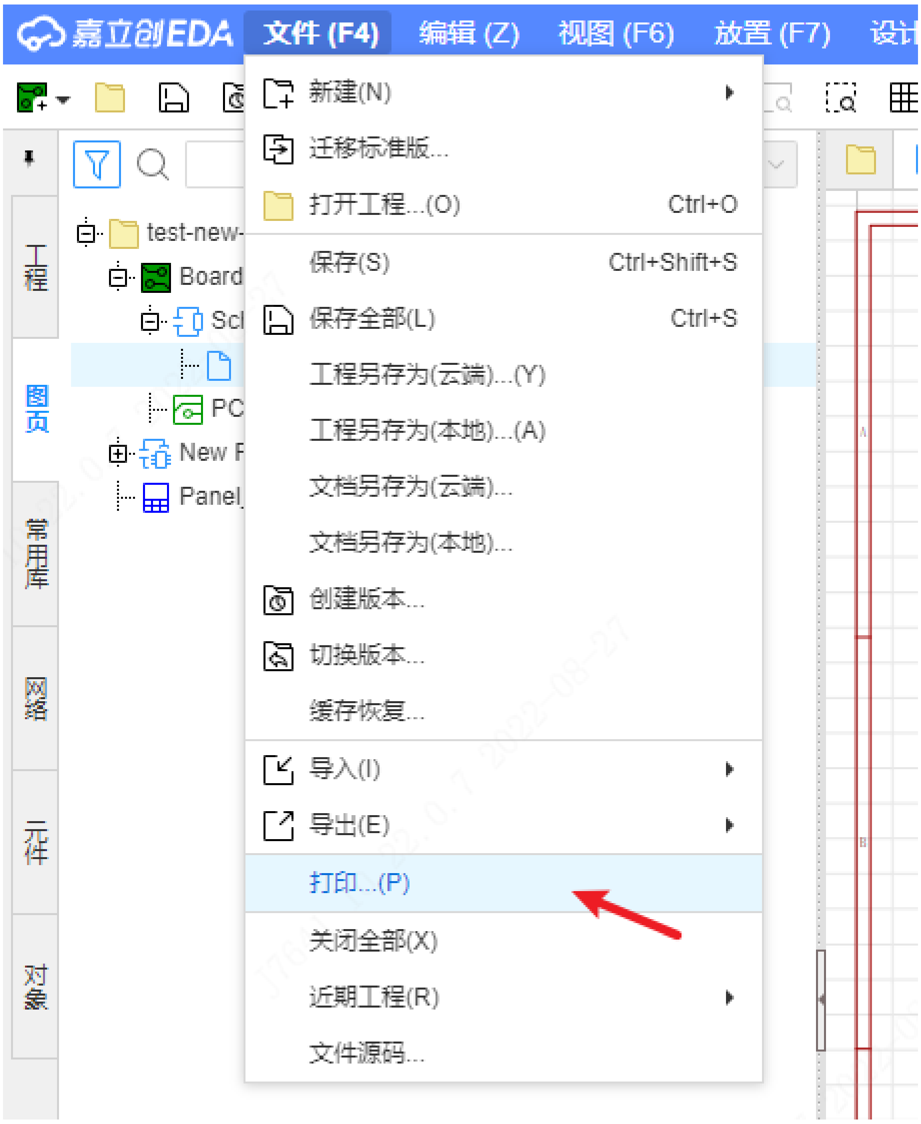


Parameter Description:
S (Scale): canvas zoom factor;
G (Grid): The X and Y dimensions of the canvas grid point, the schematic has only one value;
X (X): the current coordinate X of the cursor;
Y (Y): the current coordinate Y of the cursor;
dX (Distance X): the distance the cursor moves on the X axis;
dY (Distance Y): The distance the cursor moves on the Y axis.

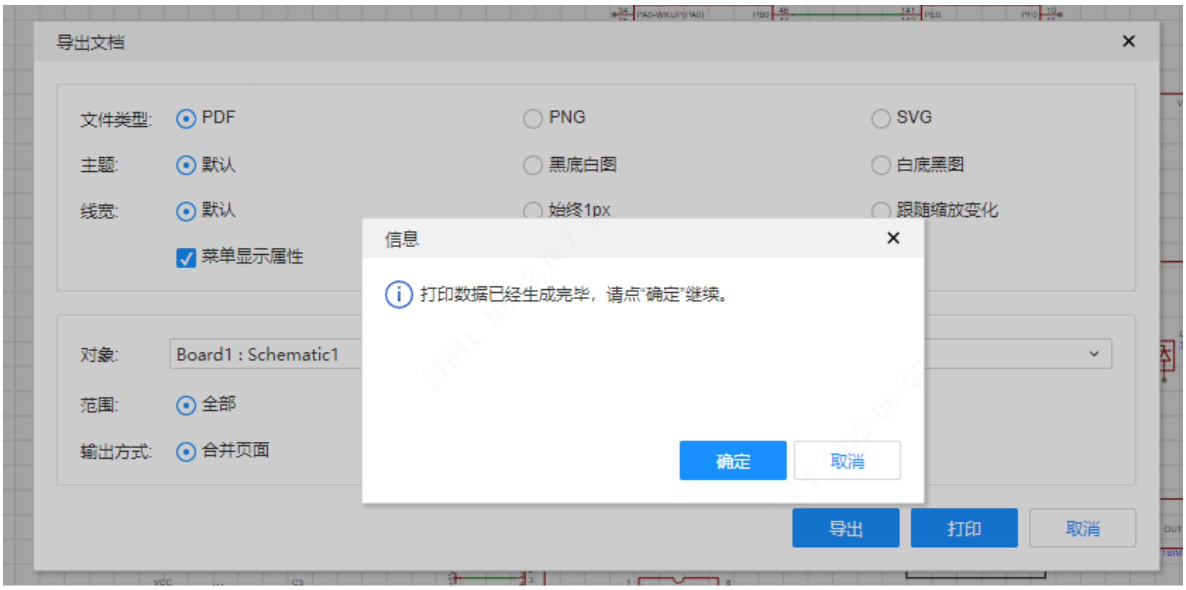
Print

After opening a schematic or PCB, the current document can be printed from the File menu.

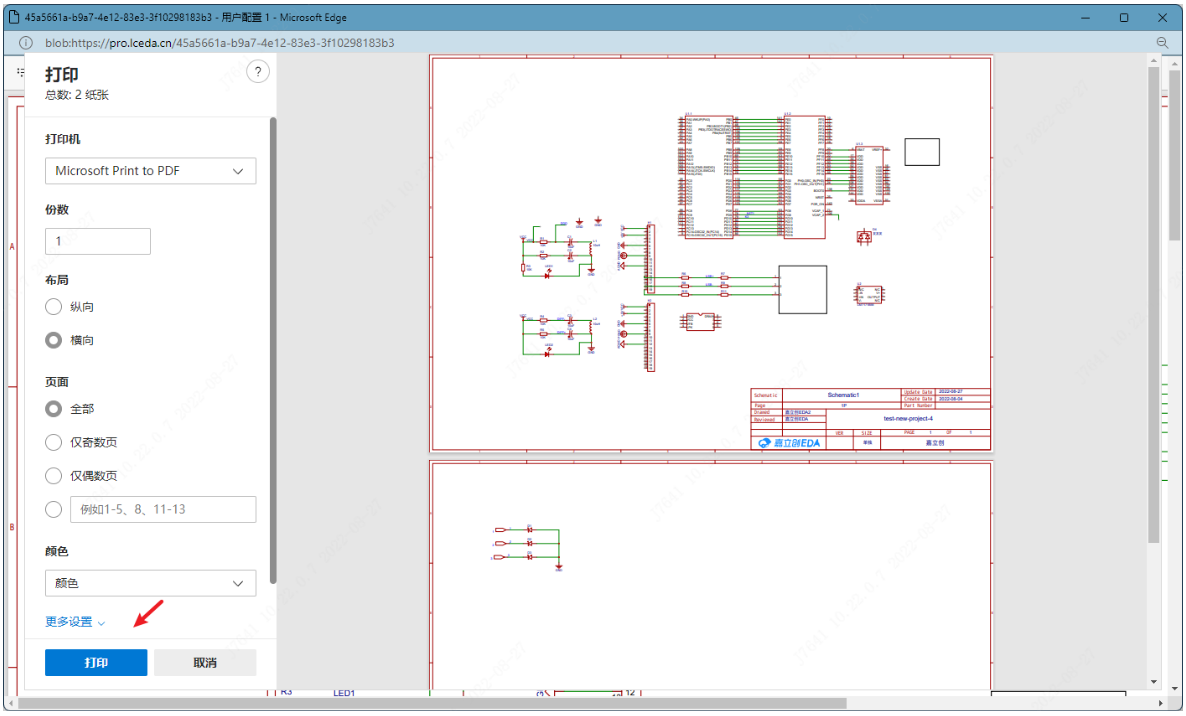
Entry: Top Menu - File - Print.



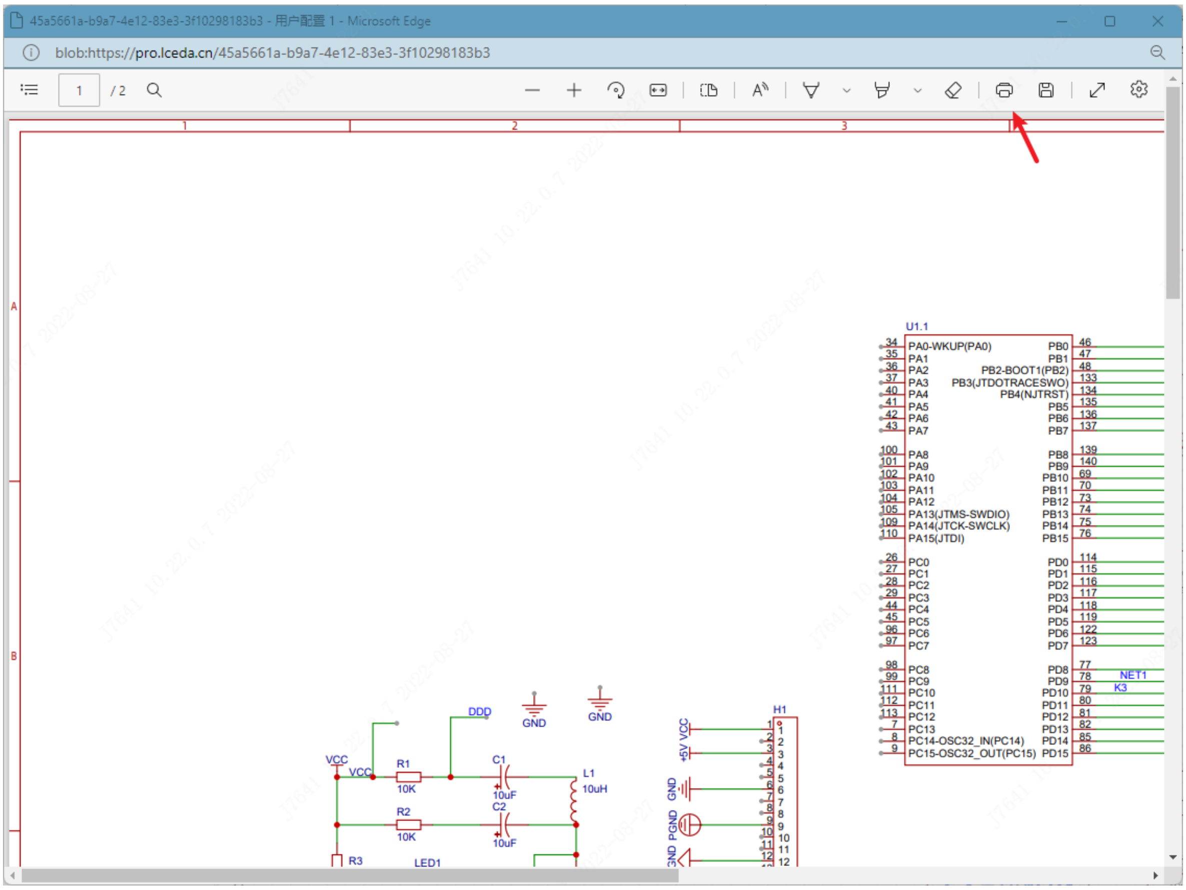
After clicking the print menu, the export file dialog box will open first, and then the print data will be generated.



After clicking OK, the document preview interface will be opened, and the system print dialog box will be opened. After setting, you can start printing.



If the system print dialog doesn't open automatically, you can press CTRL+P, or click the print icon at the top of the preview window to open it. For example, when printing the schematic diagram in the Edge browser:



hint:

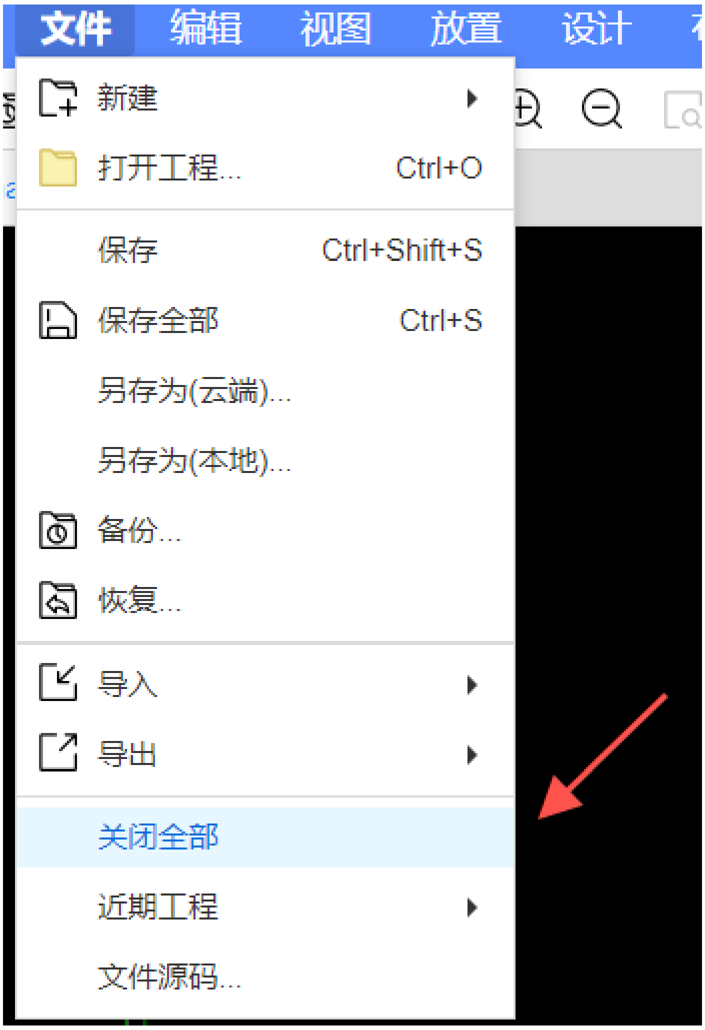
- Because printing is the way to export PDF, the export PDF pop-up window will pop up first. You can also set the parameters to be exported in the export PDF pop-up window, and then press the print button to print.
- The printing function of PCB is basically the same.

close all

Close all open project file drawing pages, and jump to the main page of the editor after closing.

Steps:

- Top Menu - File - Close All

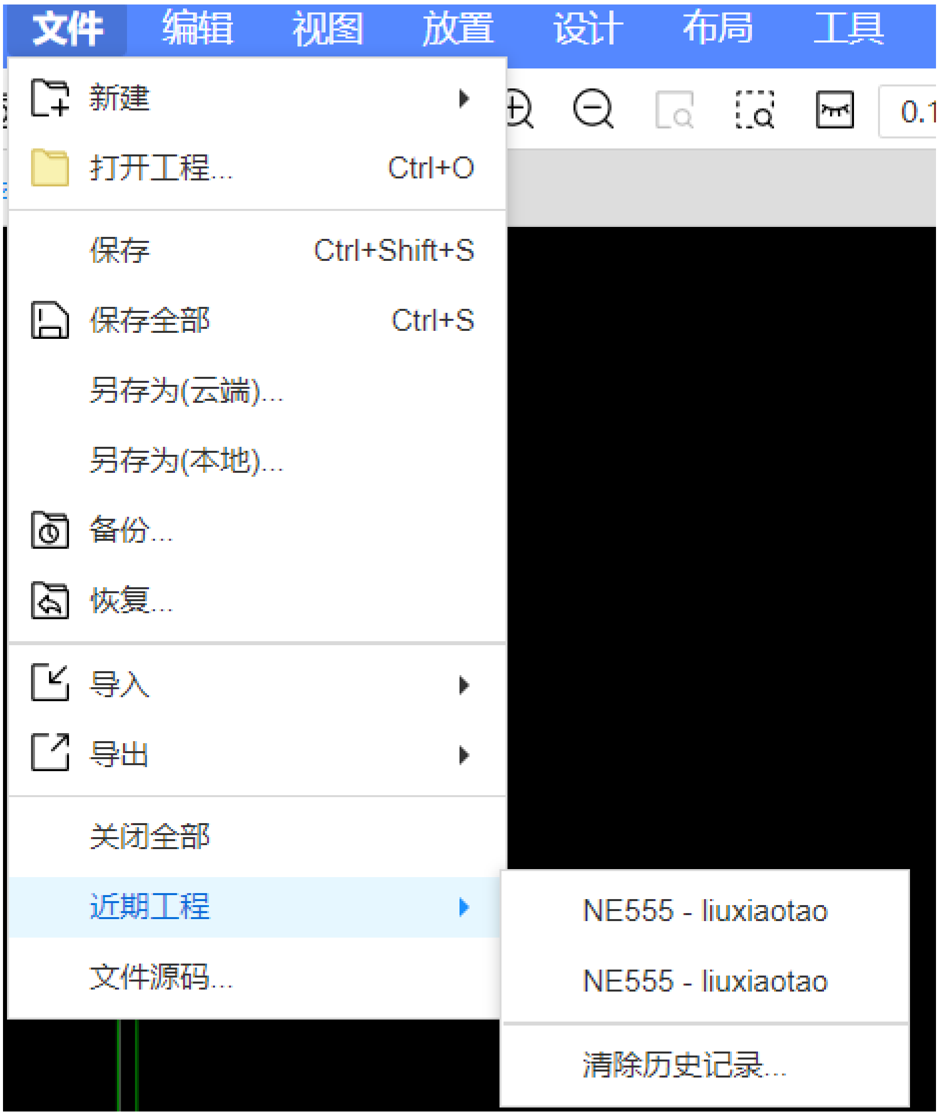


Recent Projects

Show recently edited projects

Steps:

- Top Menu - File - Recent Projects



undo/redo

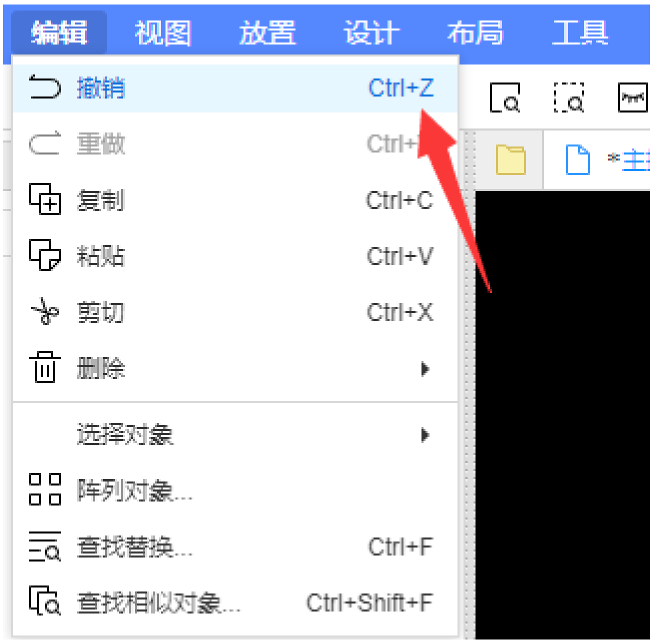
It can record a series of operation processes of the user, and can restore to the current state according to the record. After the user performs an operation, he can return to the state before the operation by undoing. After the user undoes, he can also restore the previous state by redoing Undo operation.

undo

Undo the current operation and roll back the previous operation.

Steps:

- Top Menu - Edit - Undo
- Shortcut `Ctrl` + `Z`

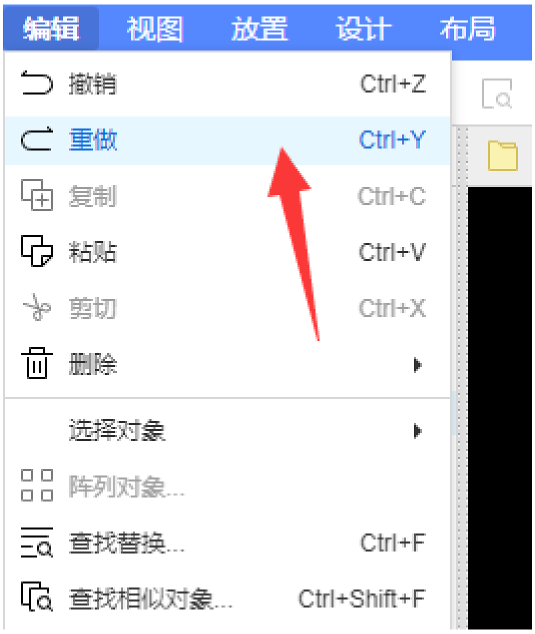


redo

Redo restores a previously undone operation.

Steps:

- Top Menu - Edit - Redo
- Shortcut `Ctrl` + `Y`



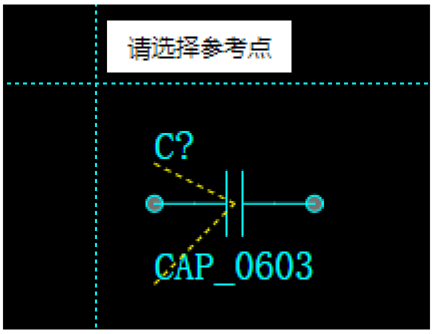
copy/paste

At present, the copy/paste of EasyEDA Pro can only be applied to copy/paste under the same browser, and cannot be copied/pasted across browsers.

copy

Steps:

- Select the content to be copied - Top menu - Edit - Copy - Select copy reference point
- Shortcut keys `Ctrl` + `C`;
- Select device - Press and hold `Ctrl` + left mouse button and drag to copy;

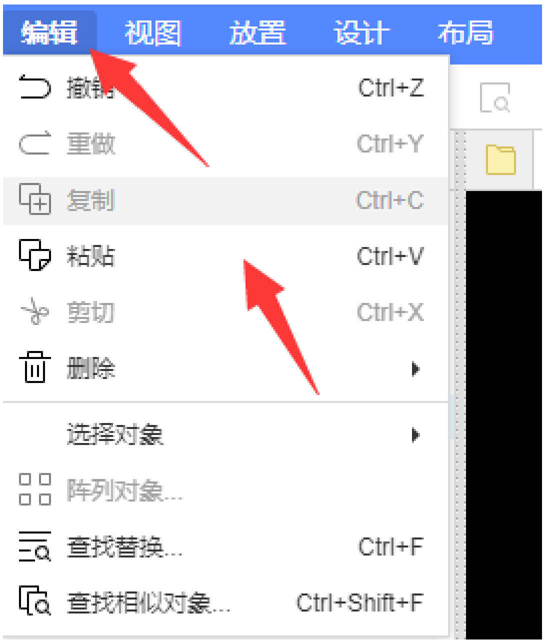


paste

Copy the selection to where it needs to be placed.

Steps:

- Top Menu - Edit - Paste
- Shortcut: `Ctrl` + `V`

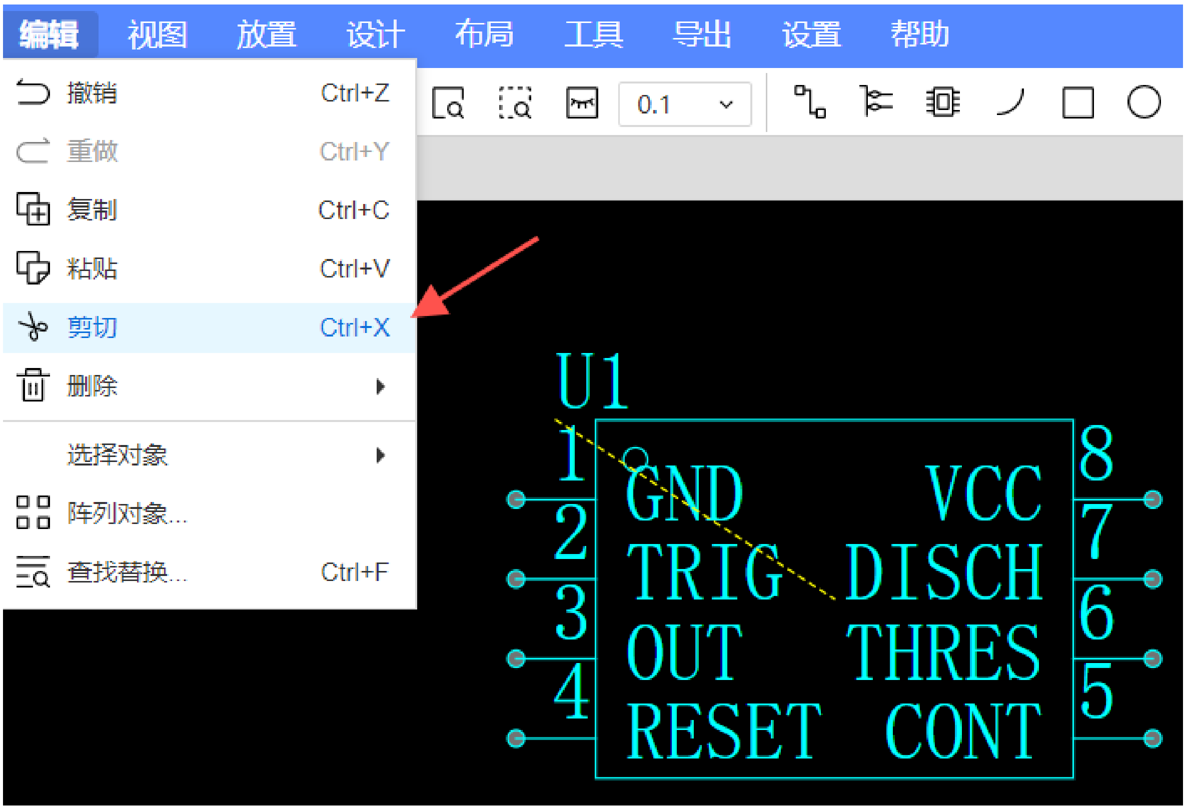


cut

Cut refers to cutting the device you selected into the clipboard. The subsequent operation is generally “paste”. After the “paste” step is completed in a new place, the information in the original place will be automatically deleted, and the information in the clipboard will be Appear in new places.

Steps:

- Select the device to be cut - Top menu - - Edit - Cut
- Shortcut: Ctrl + X



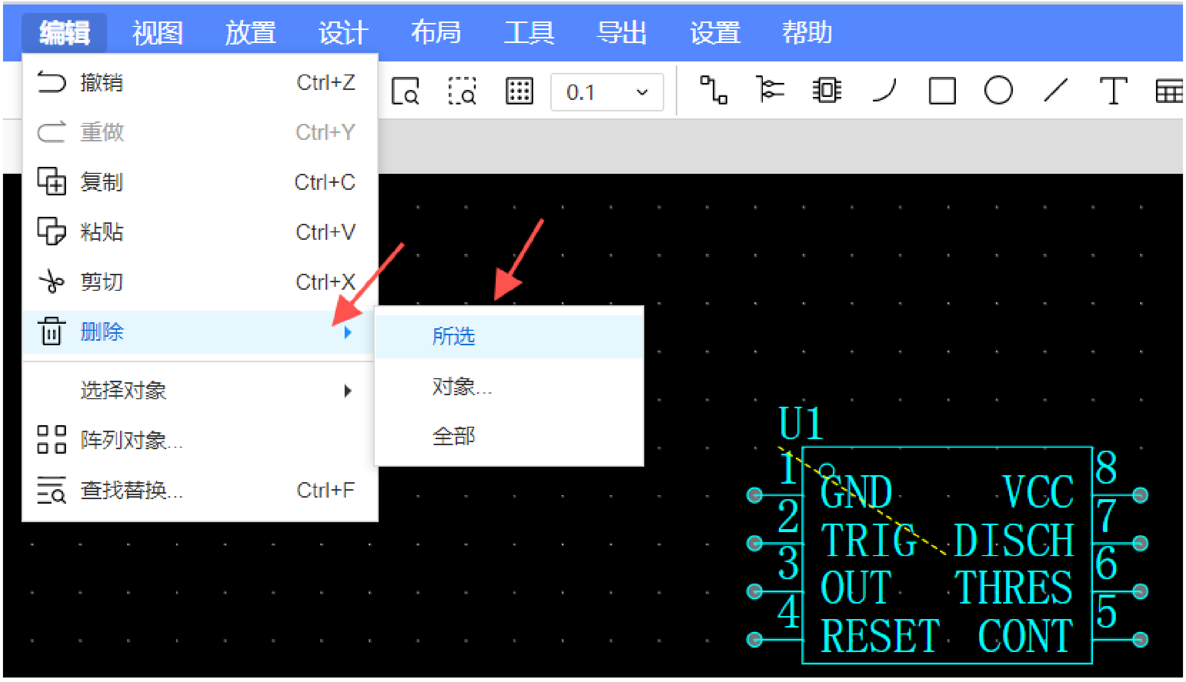
delete

Elements selected in the schematic are deleted

delete selected

Steps:

- Select the object you want to delete - Top menu - Edit - Delete - Selected
- Select the object to delete - Shortcut `Delete`

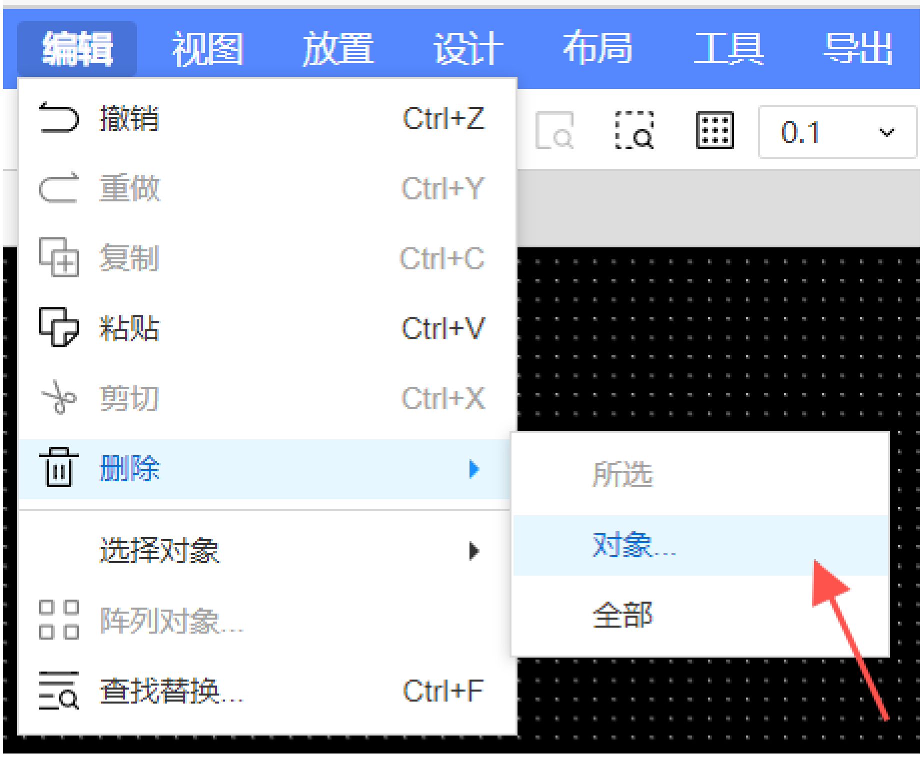


delete object

When deleting objects, you can select the content that needs to be deleted, and this deletion is deleted as a whole in the schematic diagram.

Steps:

- Top Menu - Edit - Delete - Object



In the pop-up window, select the pop-up window, check the objects to be deleted, and click OK to delete all the checked objects in the schematic diagram.

delete all

Remove all elements placed on the schematic.

Steps:

- Top Menu - Edit - Delete - All;

- Shortcut `Ctrl` + `A` Select all - Click to delete shortcut `Delete`

select object

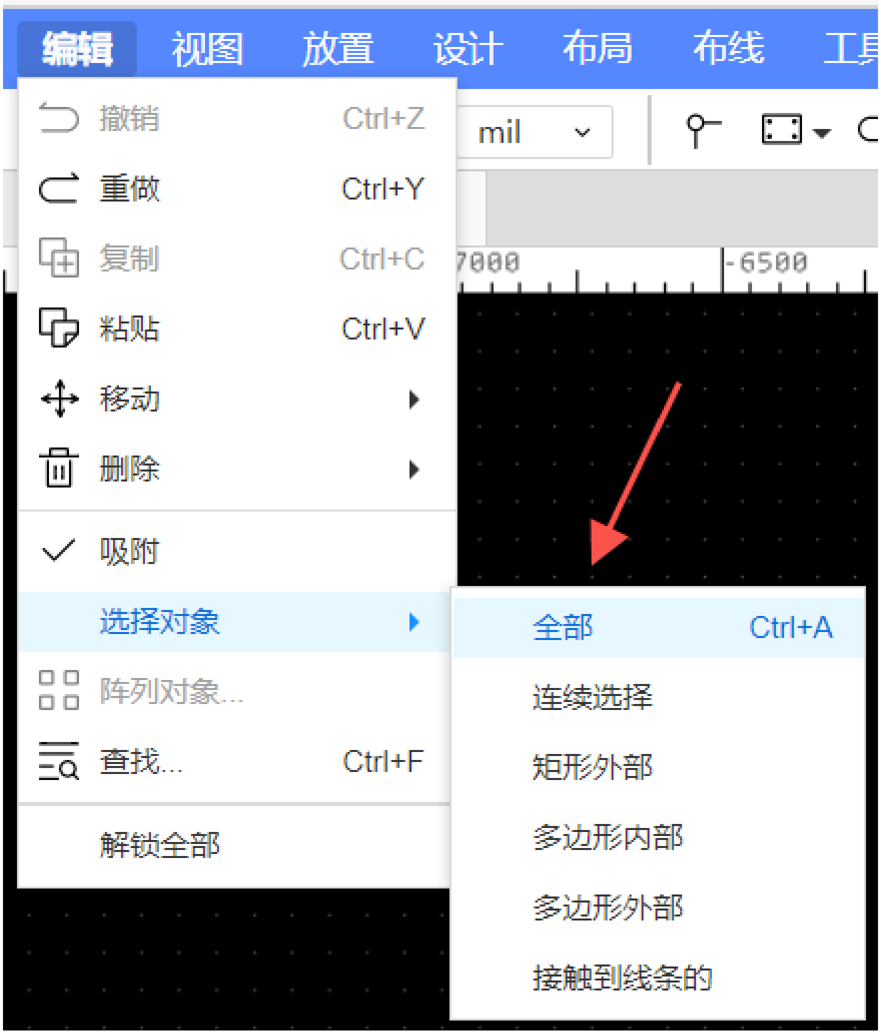
All Selection

Select all elements in the schematic.

Steps:

- Top Menu - Select Object - All
- Shortcut `Ctrl` + `A`

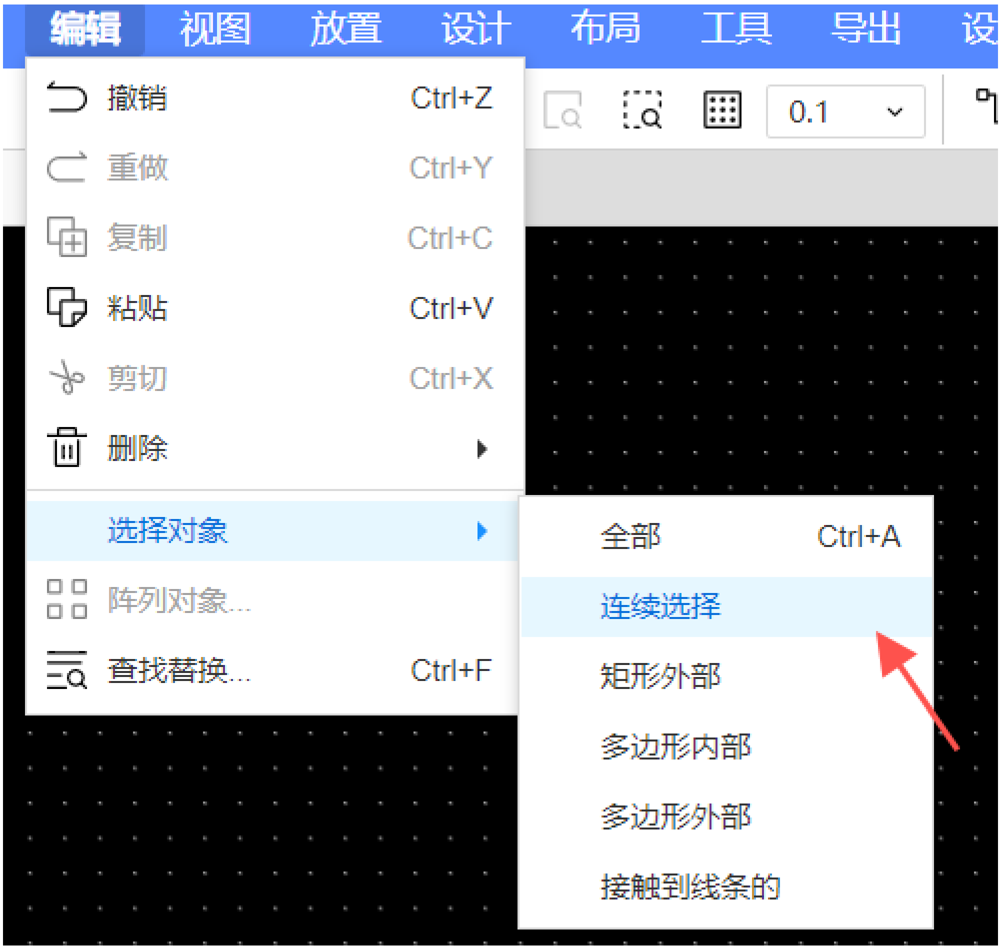
All elements in the schematic can be selected.

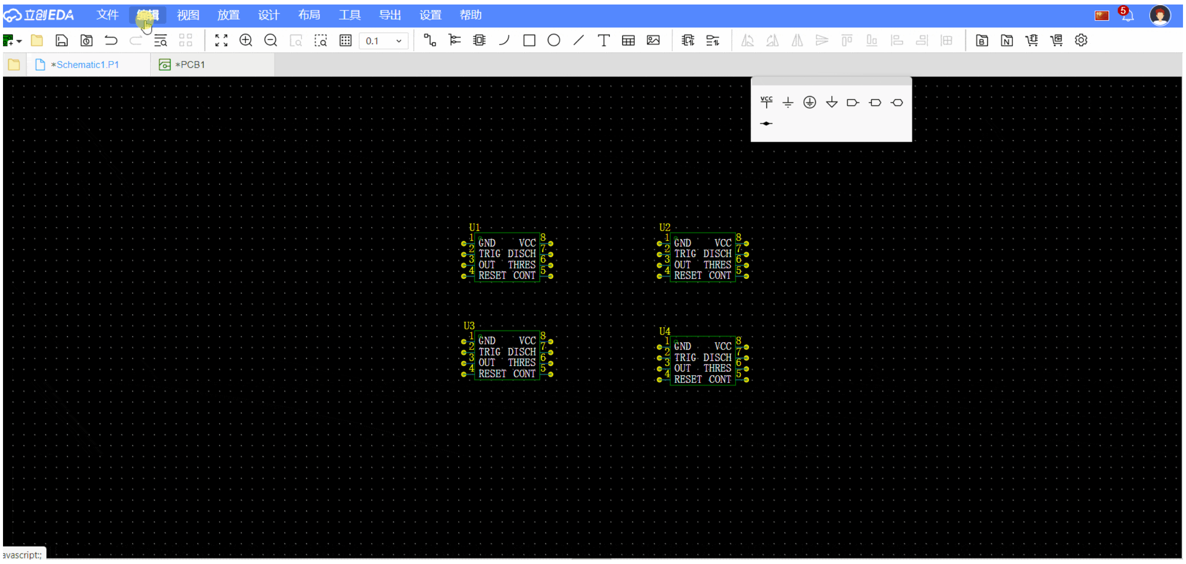


Continuous selection

Steps:

- Top Menu - Edit - Select Object - Continuous Selection



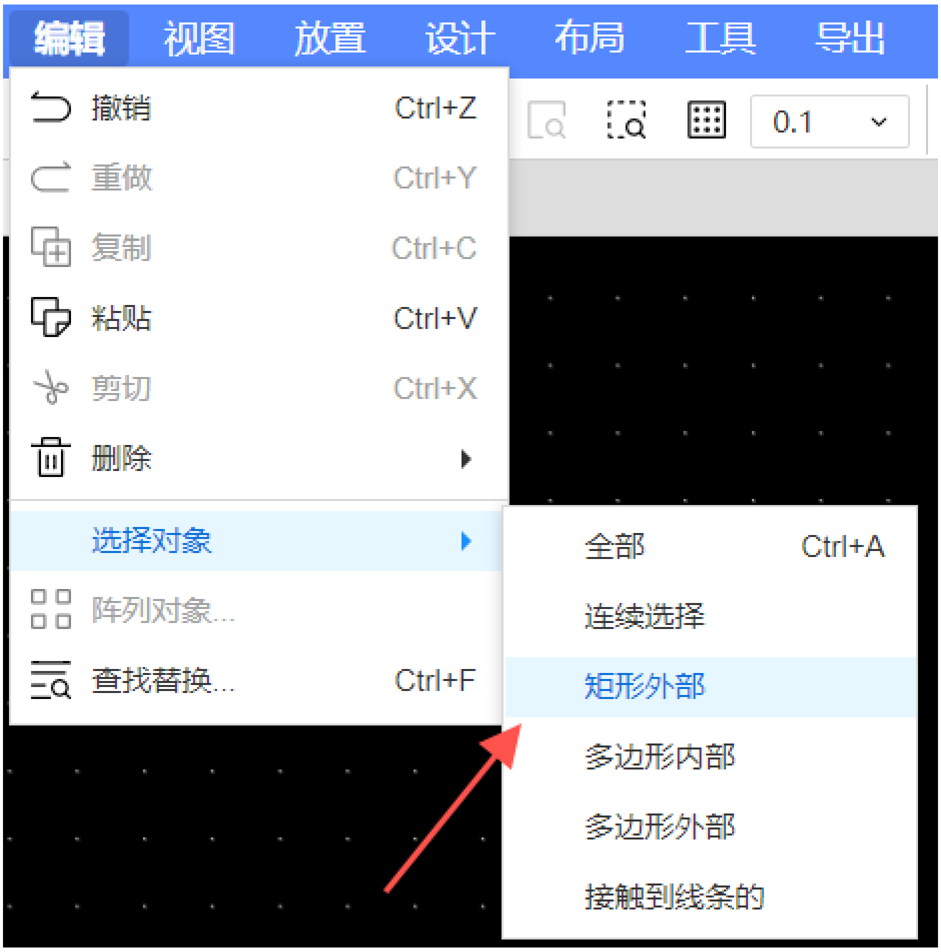


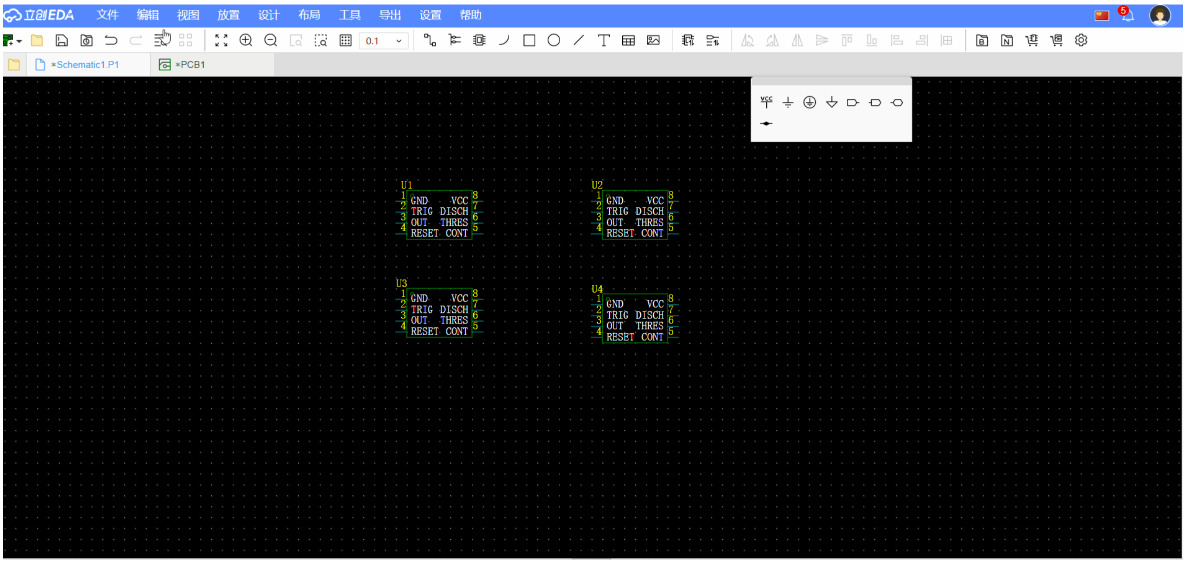
outside the rectangle

Elements outside the rectangle are selected, and elements inside the rectangle are not.

Steps

- Top Menu - Edit - Select Object - Outside Rectangle



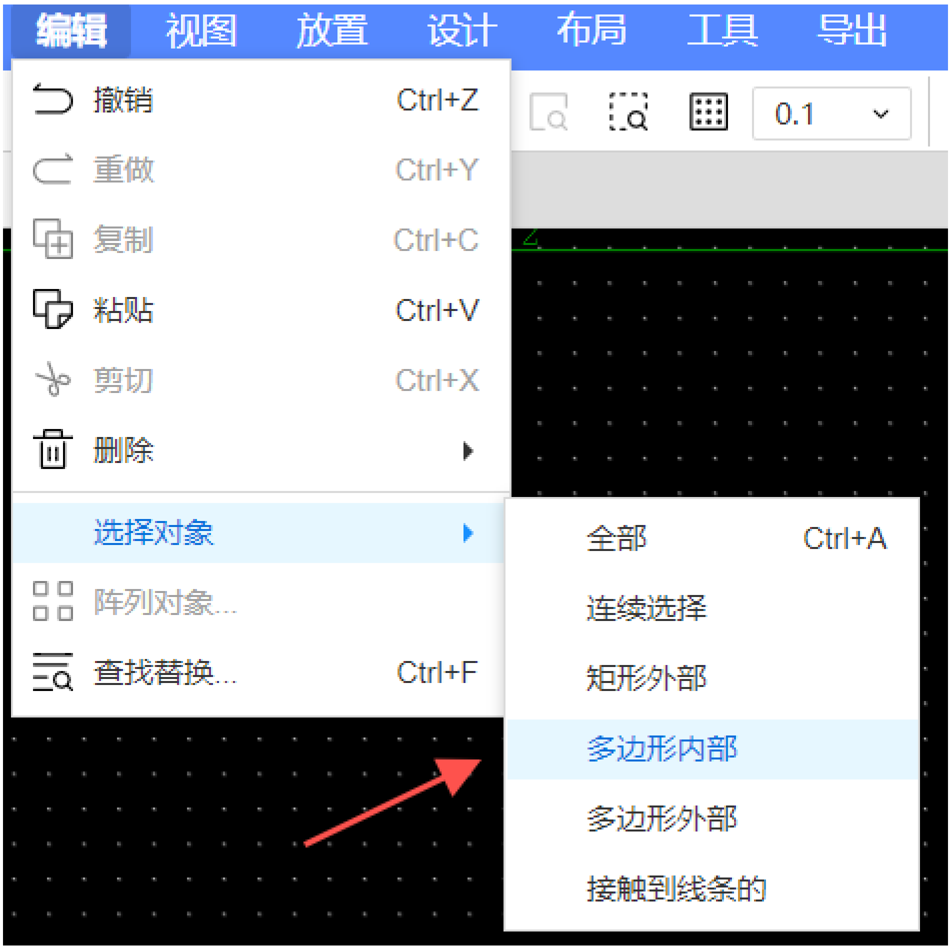


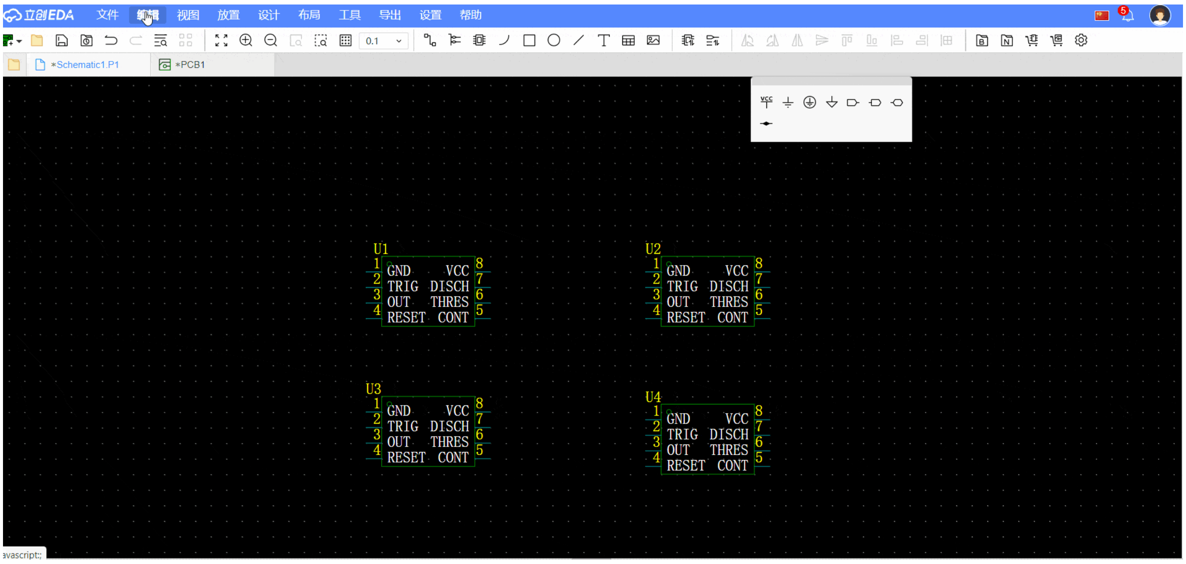
inside the polygon

Elements inside the drawn polygon are selected, and those outside are not.

Steps:

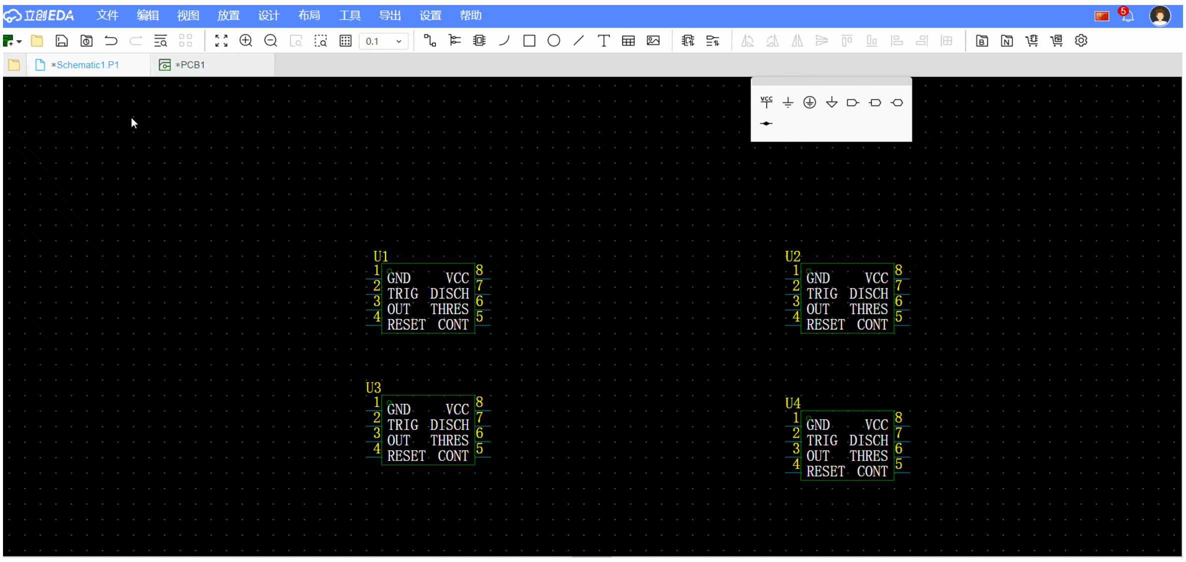
- Top Menu - Edit - Select Object - Inside Polygon





outside the polygon

Elements inside the drawn polygon are deselected, and those outside are selected.

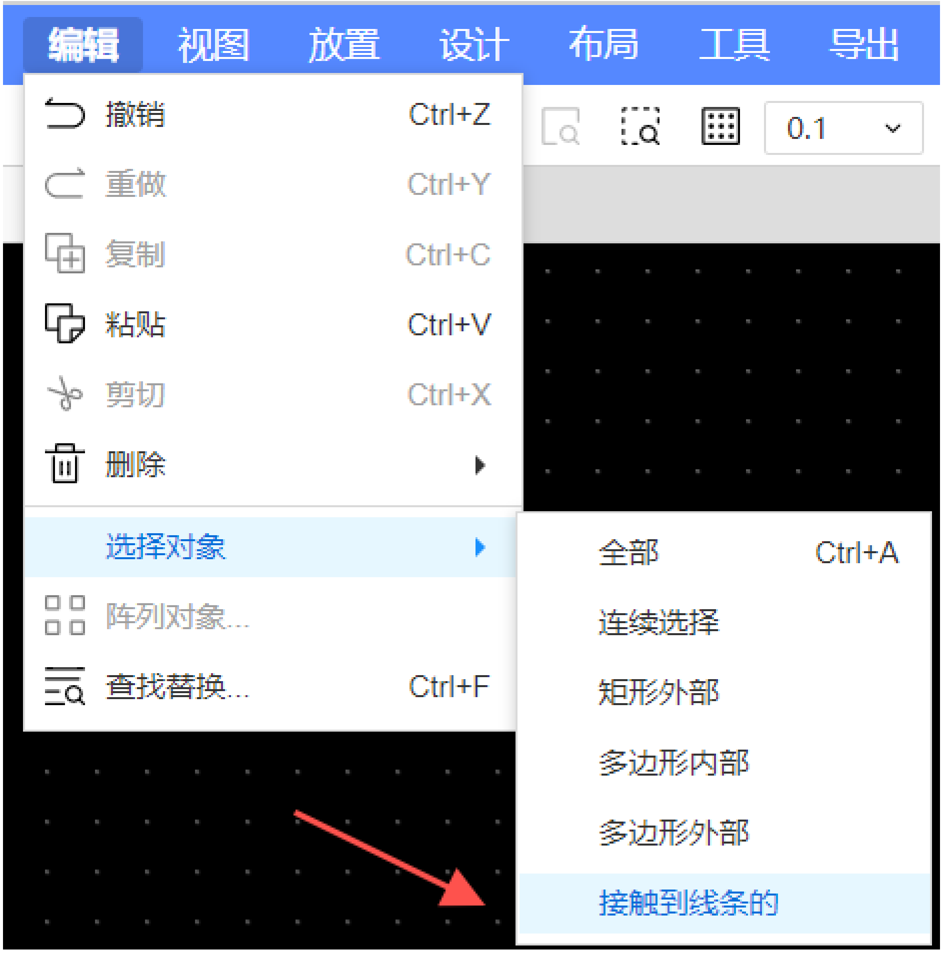


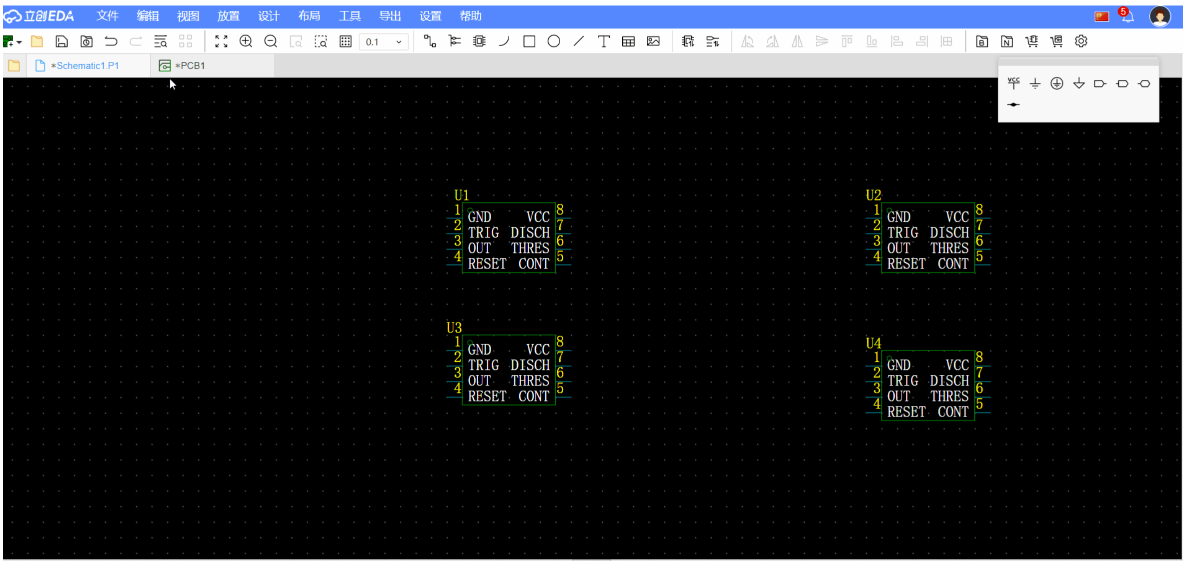
touch the line

Touch the selection of the line.

Steps:

- Top Menu - Edit - Select Object - Touch the Line's

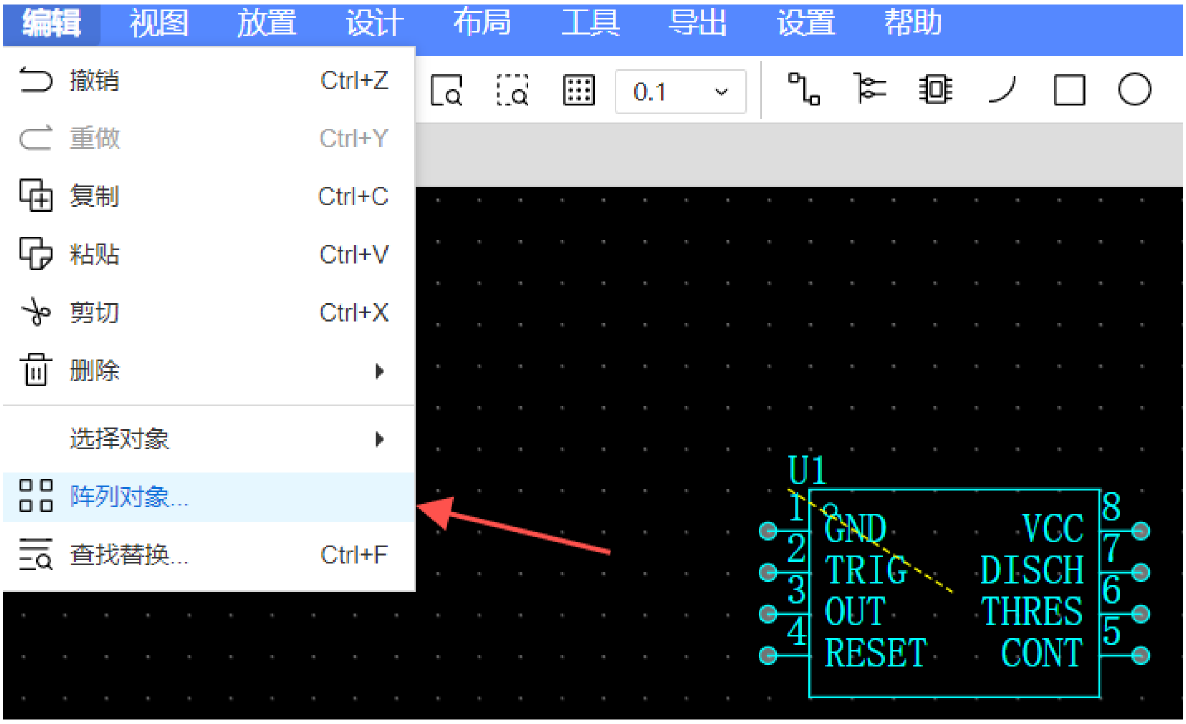




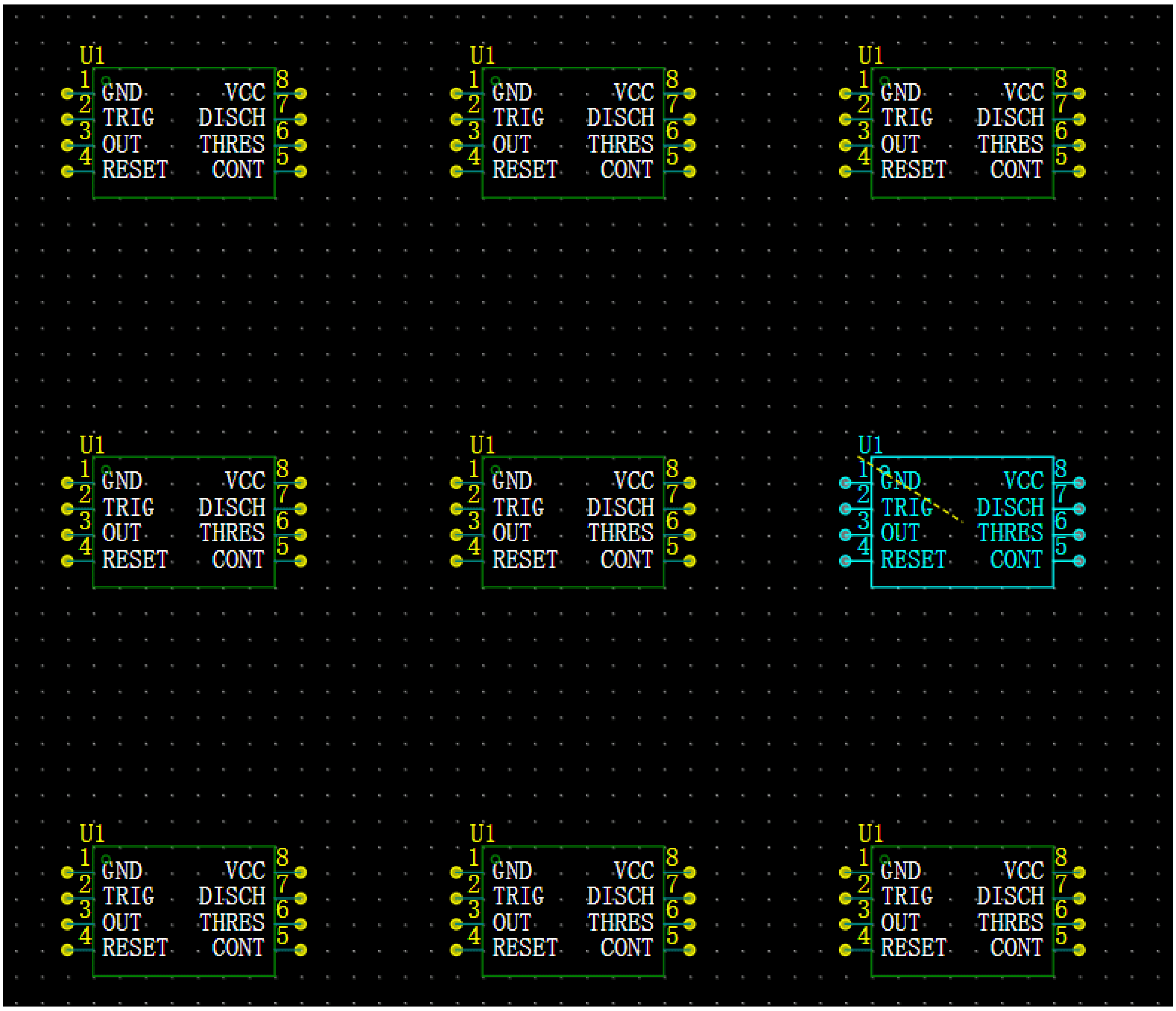
array object

Steps:

- Select Device - Top Menu - Edit - Array Object



In the pop-up window, enter the row and column of the desired array, as well as 1 row spacing and column spacing. The default unit of row spacing here is inch, which cannot be modified. After filling in the information, click OK to sort the symbols in an array according to the filled values.



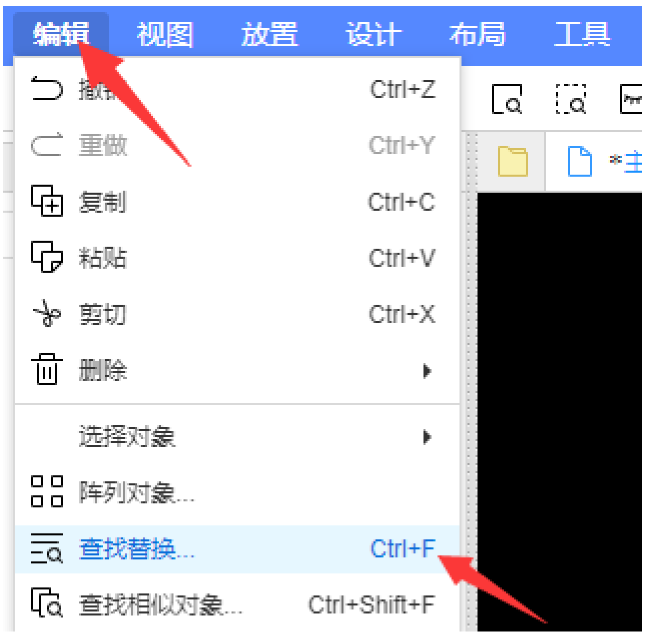
Find and Replace

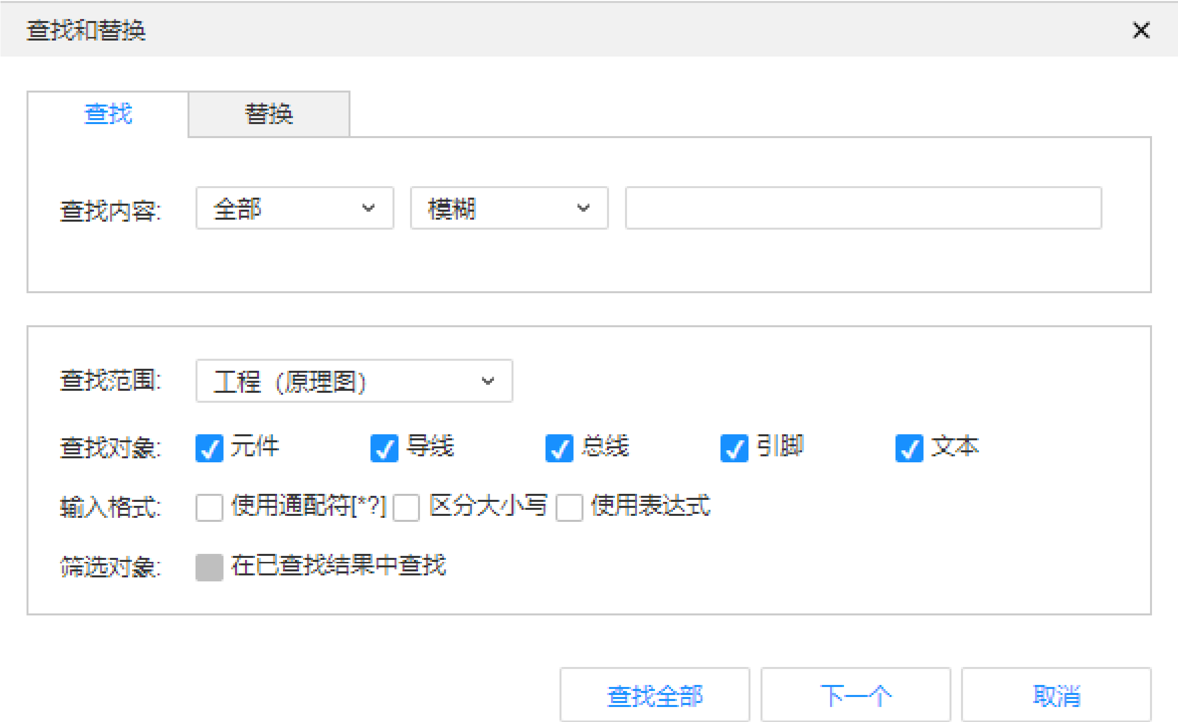
find

Find components, wires, buses, symbols, text, pins, and more in schematics,

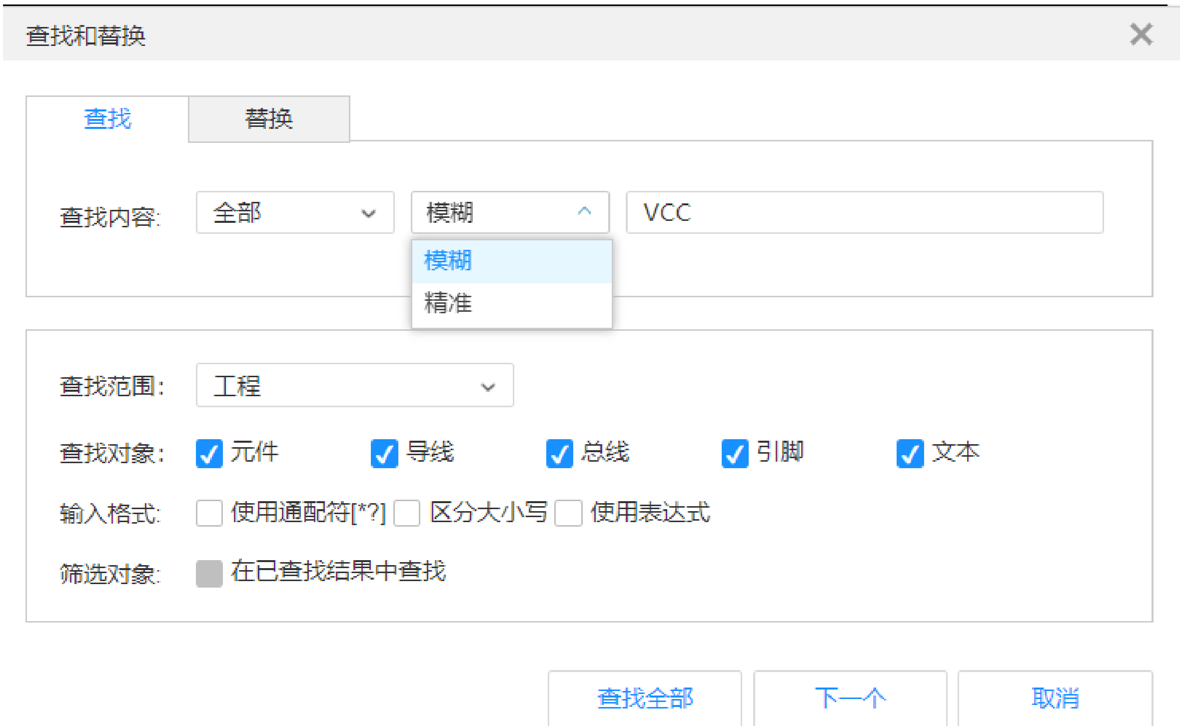
Steps:

- Top Menu - Edit - Find or
- Shortcut CTRL + F

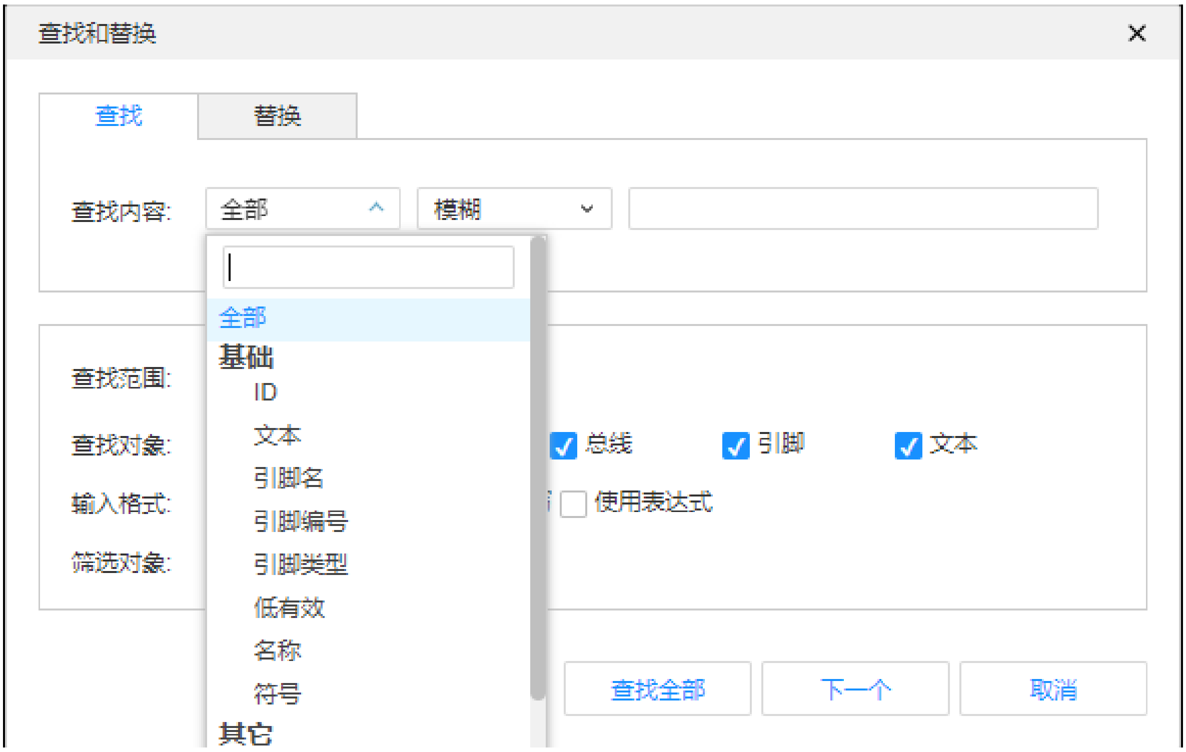




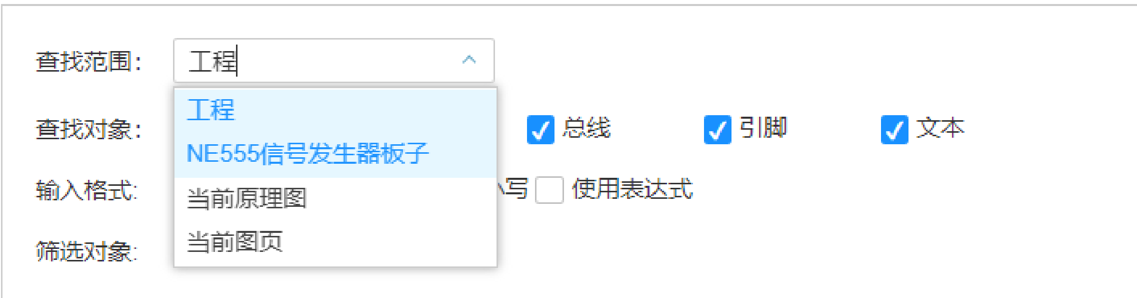
2. Enter the content to be searched, select exact, search or fuzzy search, and the project to be searched.



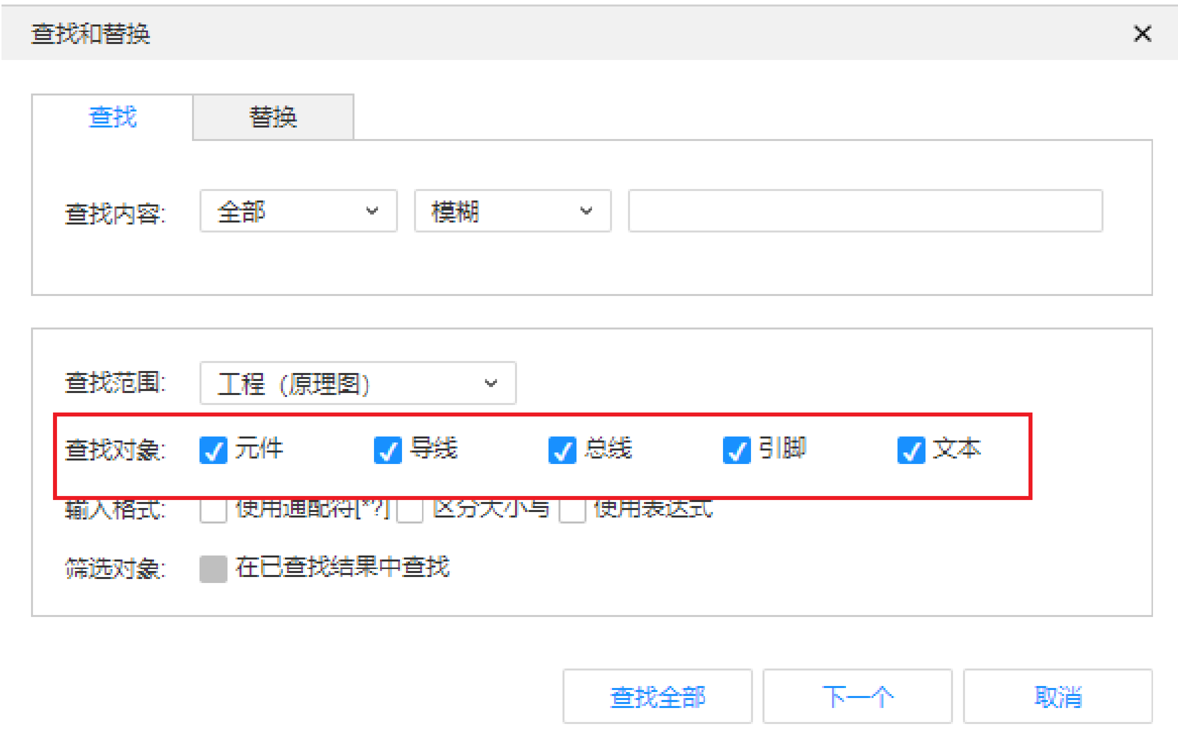
3. Select what to find.



4. Search range.



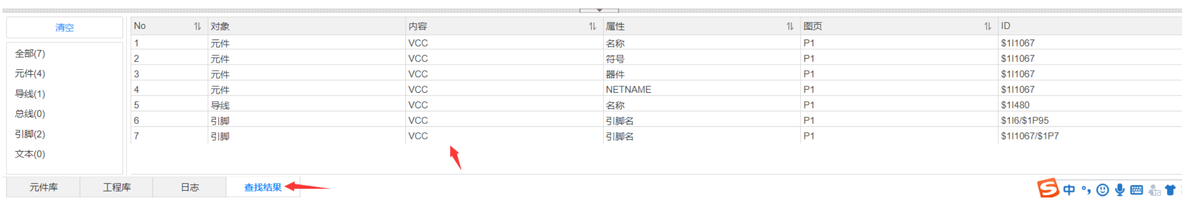
5. The object and format of the search.



- Use wildcards. When checked, all text input boxes support wildcard characters `* or ?`. For example, to search for tags: `BAT?` or `BAT*`, you can search for all tags starting with BTA. `*` can be used in place of zero, one or more characters, while `?` can only be used in place of one character.
- Case sensitive. Search by case. For example query: `max232` and `MAX232` are two different search terms.
- Use expressions. Supports inputting regular expressions in the text input box for character matching. For example, search: `/u/`, you can find all attributes with u.
  Please check the usage of specific regular expressions: https://www.w3school.com.cn/js/js_regexp.asp

6. Click Find All.

You can see the search results in the bottom menu bar

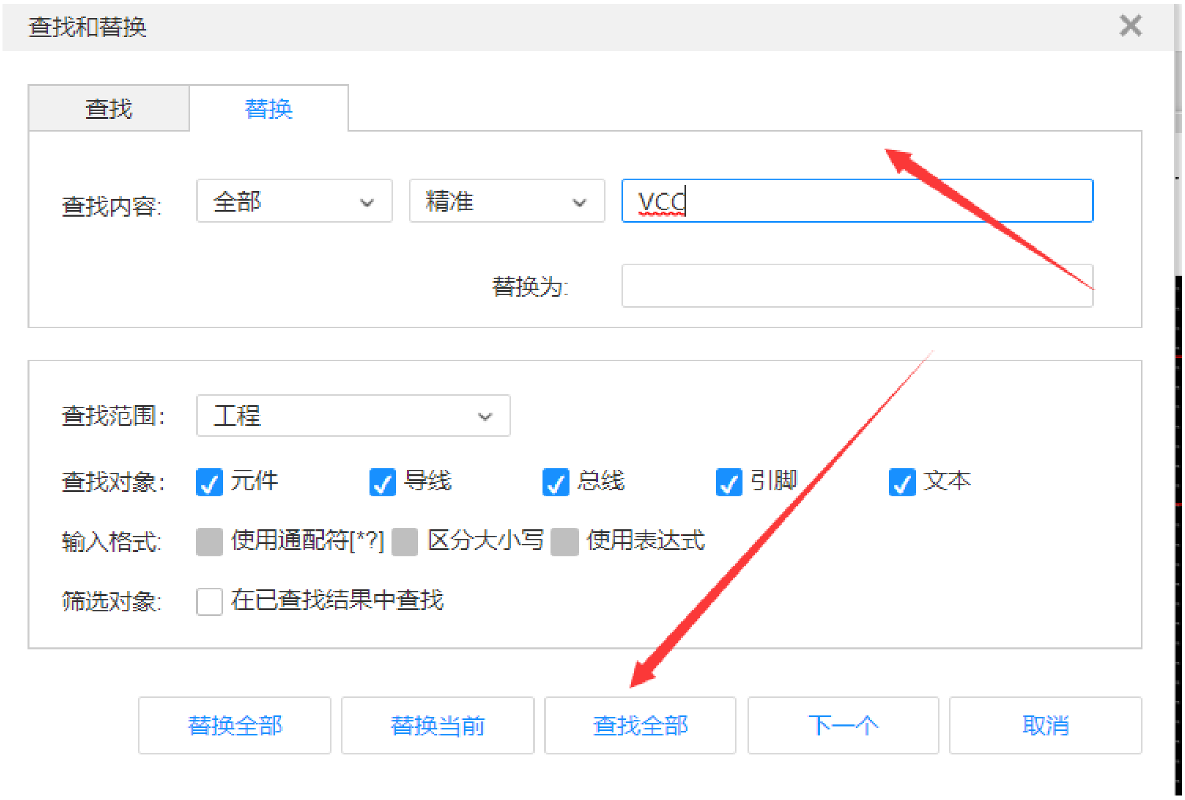


replace

Replace the result of the search, the operation is basically similar to the search.

The sequence logic of rePlace is **: find and then replace** .

1. Enter the content you want to find.



2. Enter the content to be replaced.

Replace all: Replace all the searched content under the project;

Replace Current: Only replace the currently selected content;

Find Similar Objects

EasyEDA Pro provides two search tools to find and find similar objects, which can be very convenient to find objects.

Entry: Top Menu - Edit - Find Similar Objects. Select the element on the canvas, right-click menu - Find Similar Objects. Shortcut CTRL + SHIFT + F

The Find Similar Objects opened by right-clicking will automatically select the primitive type and fill in the corresponding attribute parameters.

Type: Supports all the primitive types in the schematic and the properties of the components

Scope: Supports setting the search range, current page, current schematic, all schematics, search in the selected element, etc.

Properties: Set the search conditions and rules.

- Use wildcards. When checked, all text input boxes support wildcard characters `* or ?`. For example, to search for tags: `BAT?` or `BAT*`, you can search for all tags starting with BTA. `*` can be used in place of zero, one or more characters, while `?` can only be used in place of one character.
- Case sensitive. Search by case. For example query: `max232` and `MAX232` are two different search terms.
- Use expressions. Supports inputting regular expressions in the text input box for character matching. For example, search: `/u/`, you can find all attributes with u.
  Please check the usage of specific regular expressions: https://www.w3school.com.cn/js/js_regexp.asp

View Zoom

Interface for zooming schematics.

view zoom out

Steps:

1. Top Toolbar - View - Zoom Out
2. Shortcut key "Z"
3. Mouse wheel back zoom

view zoom

Steps:

1. Top Toolbar - View - Zoom In
2. Shortcut key "A"
3. Roll the mouse wheel forward to zoom in

fit all

Adapt to the schematic design interface, follow the page border of the schematic to adapt the view.

Steps:

1. Top Toolbar - View - Fit All
2. Shortcut key "K"

Adapt to selected

Make the selected element larger

Steps:

1. Select Device - Top Menu - View - Fit Selected
2. Select device - right mouse button - adapt to selected

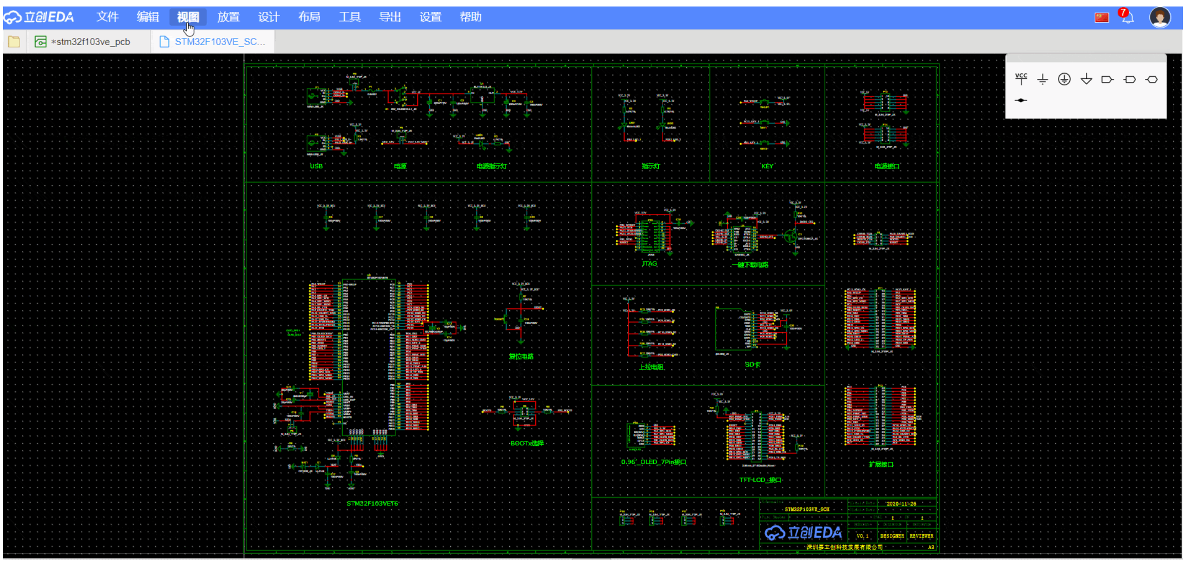
image-20210106160524051

Adapt to box selection

Enlarge the selected area

Operation steps: Top Menu - Edit - Fit Frame Selection

image-20210106160726338



full screen

Enlarge the editor as a whole according to the size of the screen.

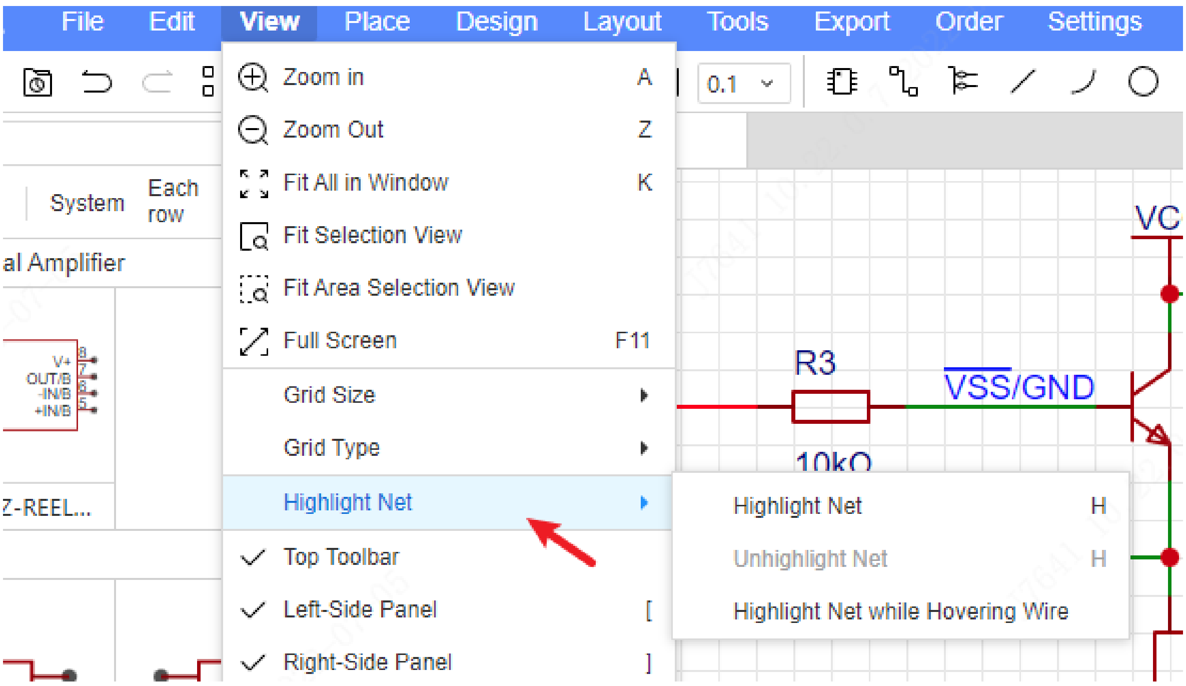
Steps:

1. Top Menu - View - Full Screen
2. Shortcut key "F11"

view - hightlight net

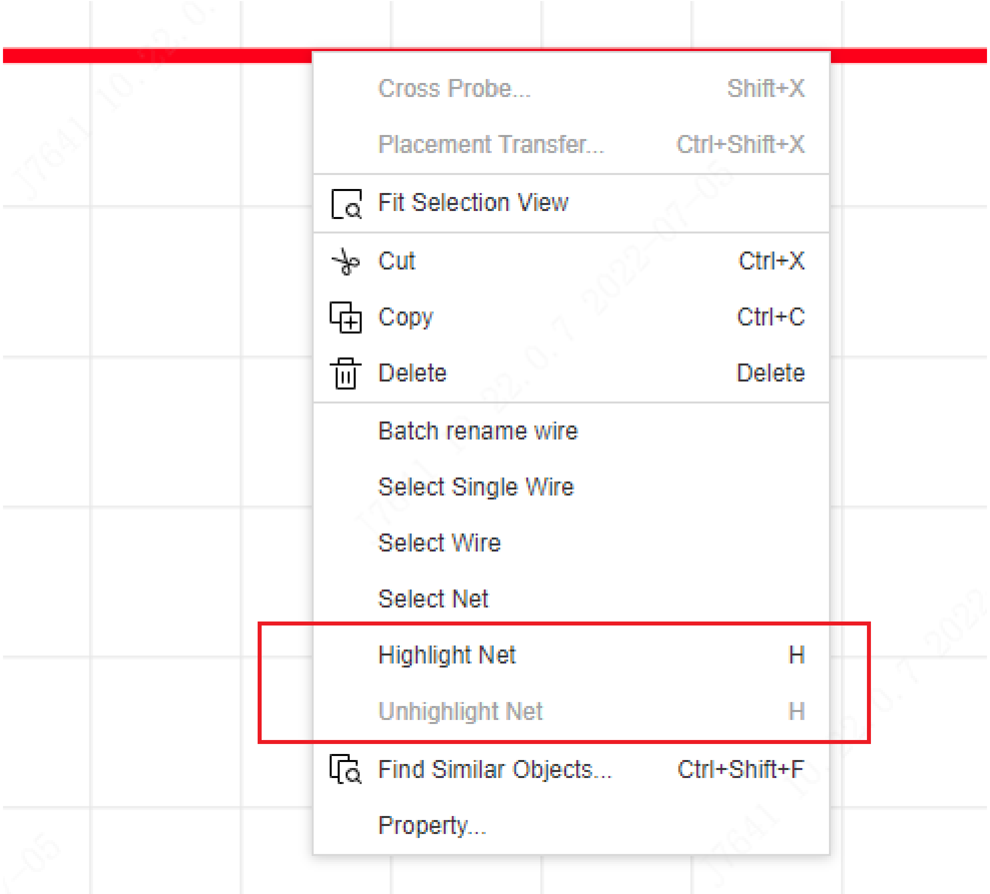
In the design process of the schematic diagram, you need to check the connection relationship of the network frequently. You can highlight the network through the highlight network function in the top menu.

Entrance: Top Menu - View - Highlight Network.

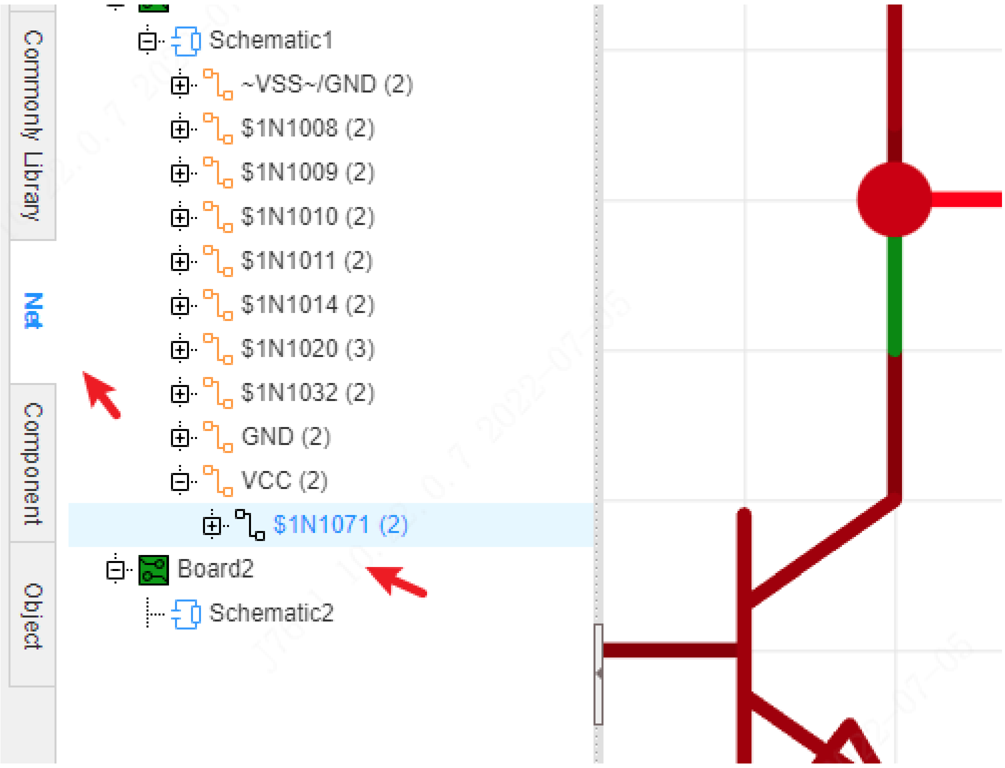


You can also use the shortcut key H to highlight. Before highlighting, you need to select the wires you want to highlight on the canvas.

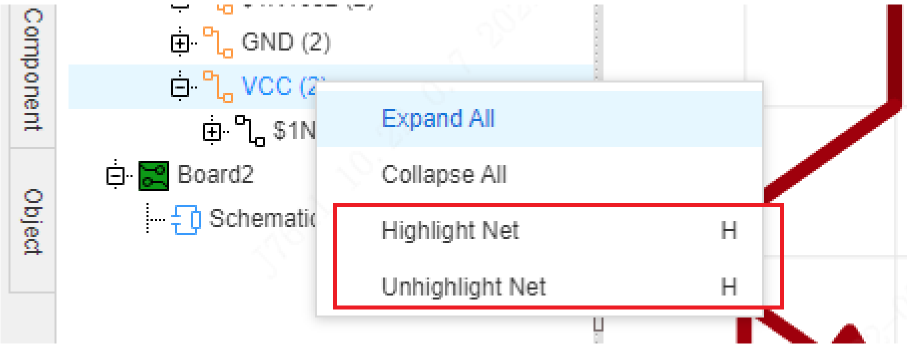
Or right-click the wire to highlight or cancel the highlight.



Or in the network tab on the left, click the wire or network name to highlight, but the highlight of the entry is temporarily selected and highlighted, and the previous highlight menu is continuously highlighted.



You can also right-click the network list on the left to highlight the network



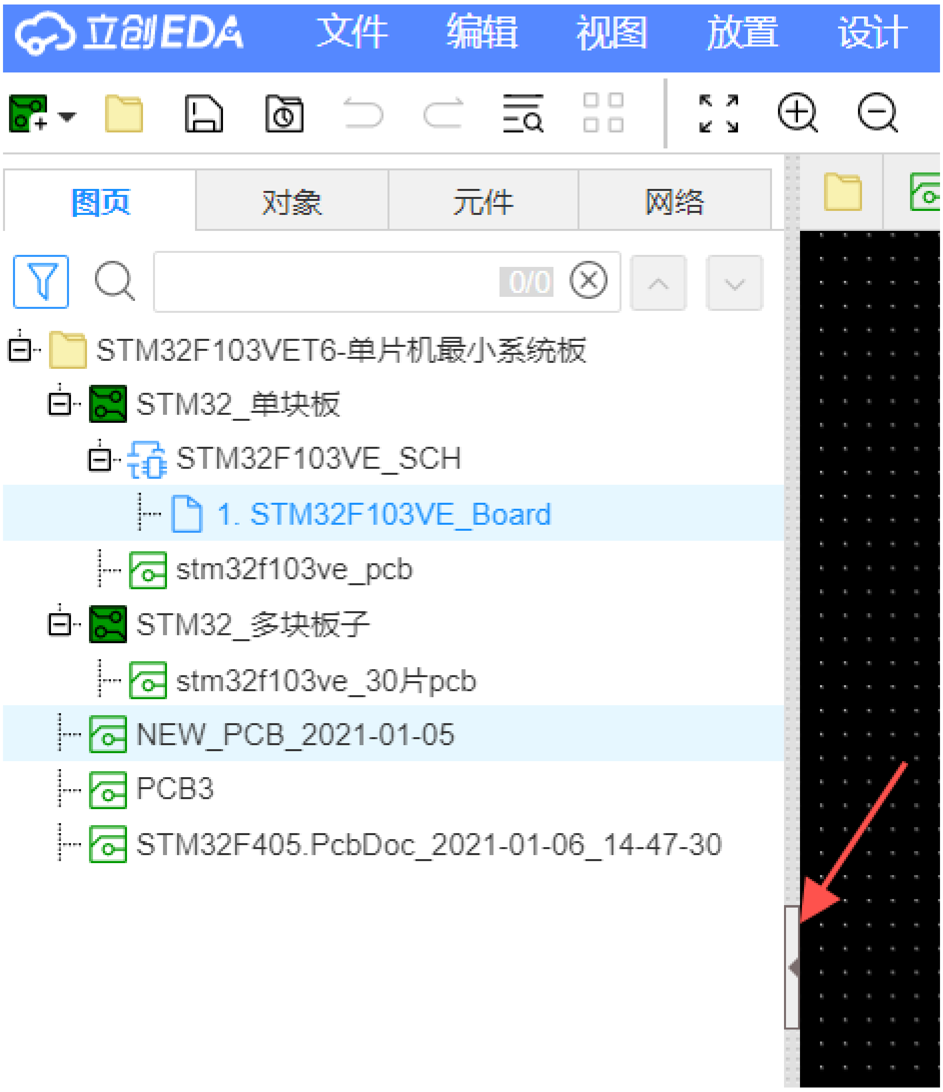
Note: The highlight network function is to highlight the entire schematic diagram. When you highlight a network, open a new diagram page, and the wires of the corresponding network in the diagram page will also be highlighted.

view layout

The view layout is to fold the left and right panels and the top and bottom panels to make the editor interface larger and more concise.

Collapse the panel

Click the arrow next to the panel to collapse the panel.

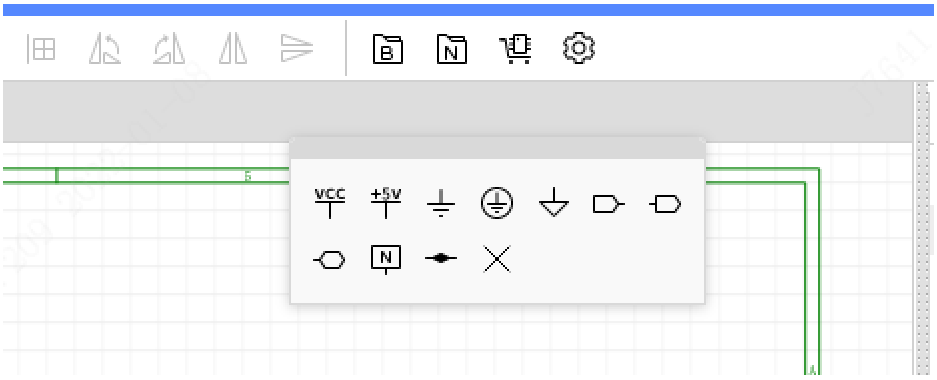


open panel

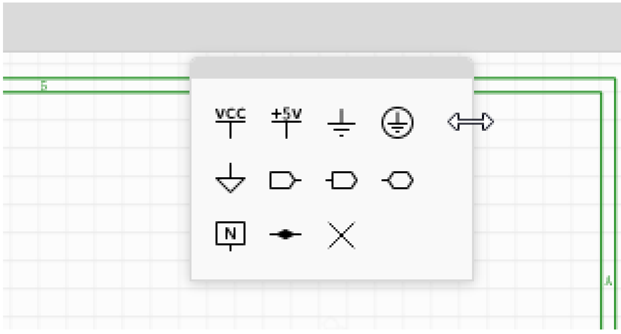
The collapsed panel will be placed at the very edge of the editor. You need to open the arrow by clicking the edge to open the collapsed panel. The bottom panel can be opened by using the shortcut key "S" or in the top menu - View Select Open Panel.

Special Symbols

After opening, the schematic diagram will display a floating toolbar of special symbols, which is convenient to quickly place commonly used symbols.



Support drag the width of the toolbar, adjust the size you need, you can move to any position.



device

Devices are called components after the canvas on which they are placed.

Both the schematic diagram and PCB of EasyEDA Pro use the template mechanism. After placing a device on the canvas, the device will enter the project library as the template of the project. When placing the same device in the future, the template of the project library will be used first, and will not be used by the device. Affected by the update of the library.

If you want to update the project library template, you can go to: Top Menu - Design - Update Symbol, or Canvas Component, right-click menu - Update Symbol to update the device, symbol or footprint of the project.

Device dialog

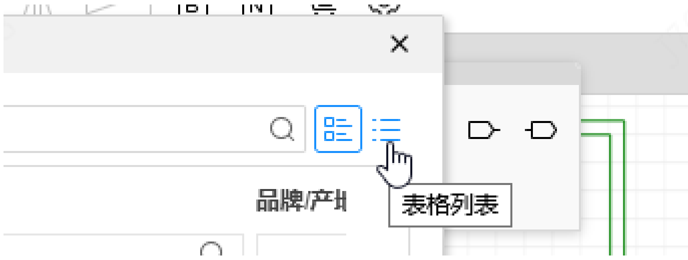
Entry: Top Menu - Place - Devices. or top toolbar - place device icons. Shortcut key SHIFT + F.

Fig.25

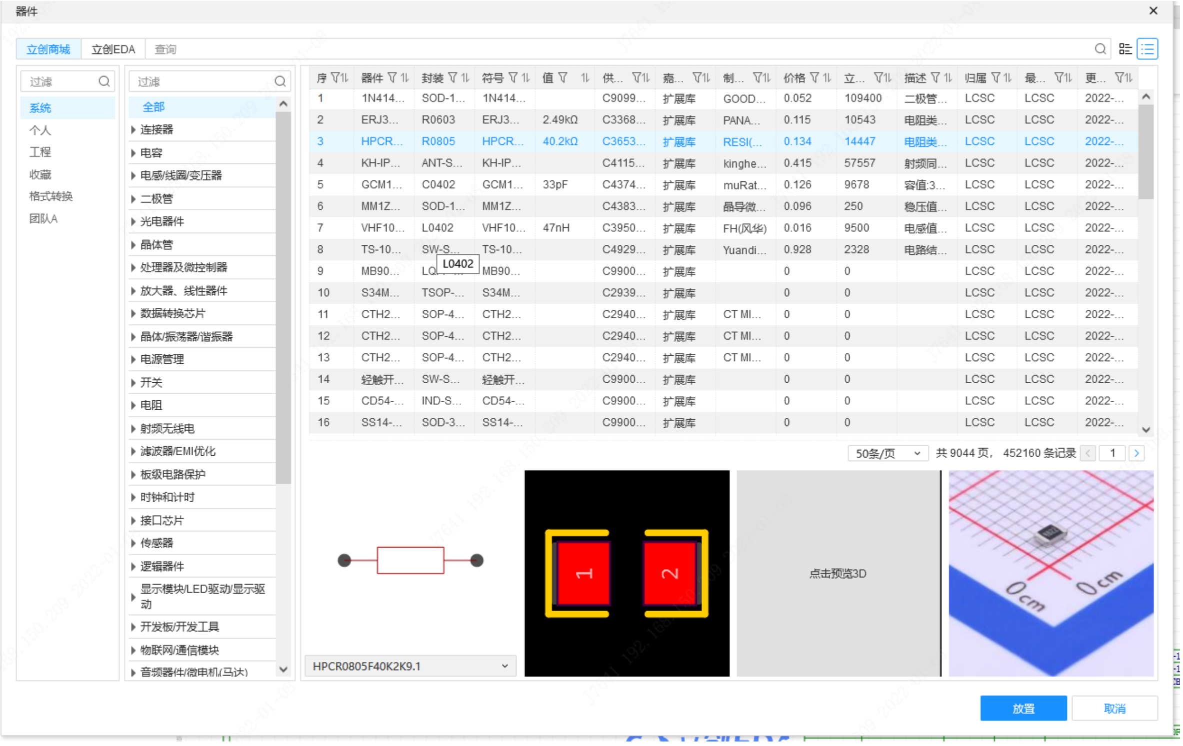
The search engine supports the dual engines of LCSC Mall and EasyEDA Pro. When one of the engines does not meet the search expectations, it can be switched. The LCSC Mall engine can only search for devices in the system library.

Fig.26

The default is the details list, which supports parameter filtering. You can also switch to the table list.



The table list supports preview of physical images, symbols, footprints, and 3D model preview images.



When you click to place a device, the device dialog box will disappear automatically, and when you cancel the Place, it will be displayed waiting for a new device to be selected. In the table list mode, you can also double-click the library list to place it.

Tip: Use the ESC key to close the drop dialog.

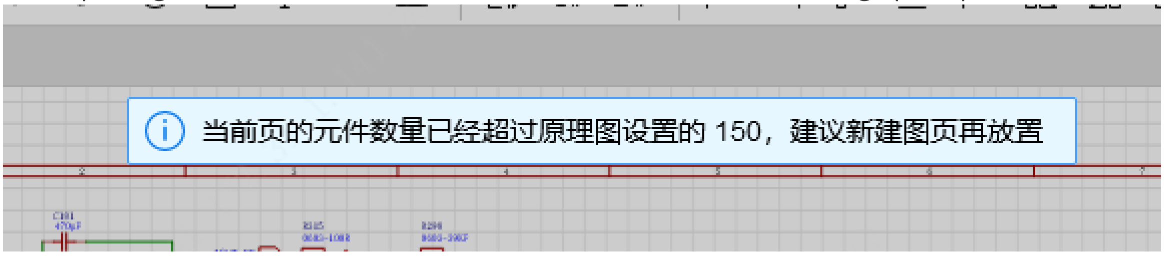
Bottom component library

At the same time, it supports another entry to place devices, the bottom panel, and the shortcut key "S"

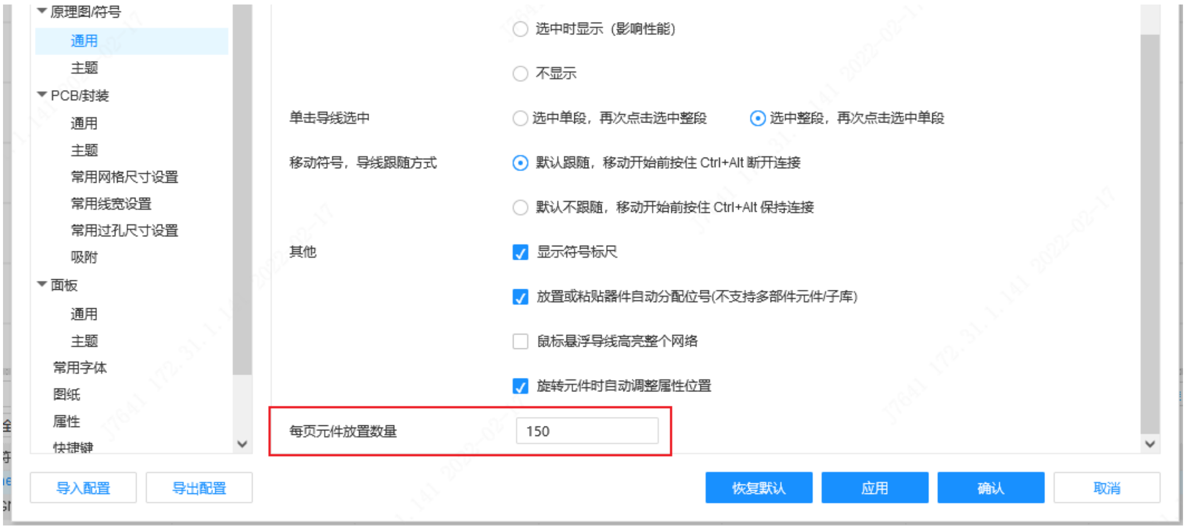
Select a device and click the Place button, or double-click to place it.

In the project library, you can edit the device, symbol, and footprint template content of the current project. After the update, the corresponding drawing page and PCB will be automatically refreshed.

When placing more devices than the set number, there will be a message prompt.



At present, the schematic diagram of EasyEDA Pro does not use the same new engine as the PCB, so it is not recommended to place more than 150 components per page, otherwise it will be stuck. If the number of components is large, it is recommended to create a new drawing page.



Component properties

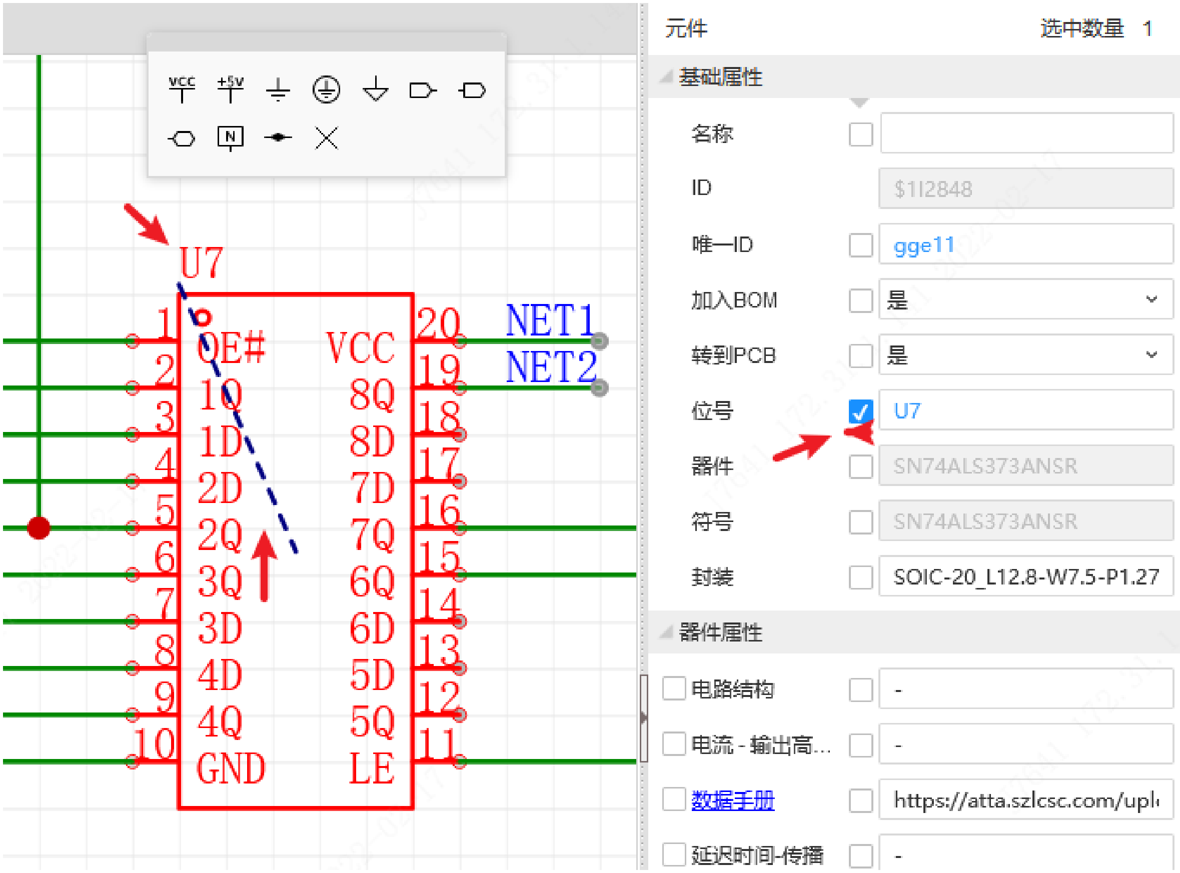
After clicking the component, the properties of the component can be modified and added in the property panel on the right.

Figure 3

Attribute name description:

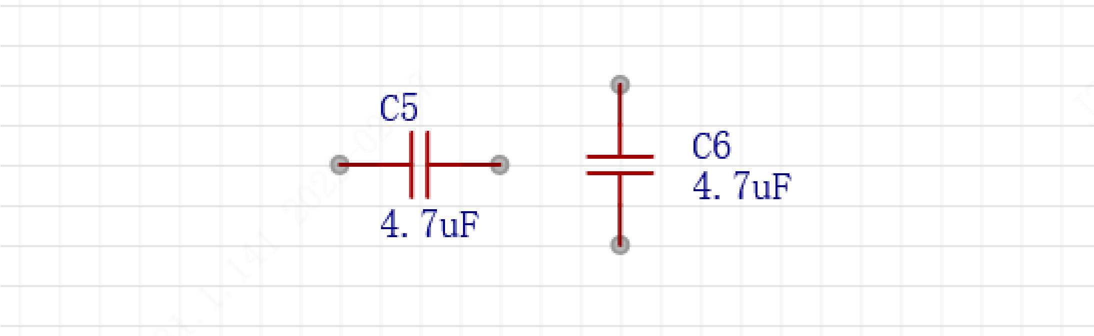
- Name: This is equivalent to the remark of the component, and usually does not need to be filled in.
- ID: ID used internally by the editor.
- Unique ID: The ID associated with the PCB. The component corresponding to the PCB is determined by this ID. It will be automatically assigned when the PCB is updated, or it can be entered manually.
- Add BOM: whether it can be exported in the BOM.
- Go to PCB: Whether it is possible to go to PCB.
- Tag: Component tag. When the device is placed, the bit number is automatically assigned by default, which can be set in the setting-schematic diagram. Components of a multipart or sublibrary are not aware of automatic tagging.
- Device: The device this component belongs to.
- Symbol: The symbol template associated with this component.
- footprint: The footprint template associated with this component can be clicked to replace the new footprint.
- Device Properties: Properties filled in when creating a new device.
- Custom Properties: Custom properties added to components after opening the schematic. Add a custom attribute name, you can add it in: Settings - Attributes.

Check the attribute name or attribute value to display on the canvas.



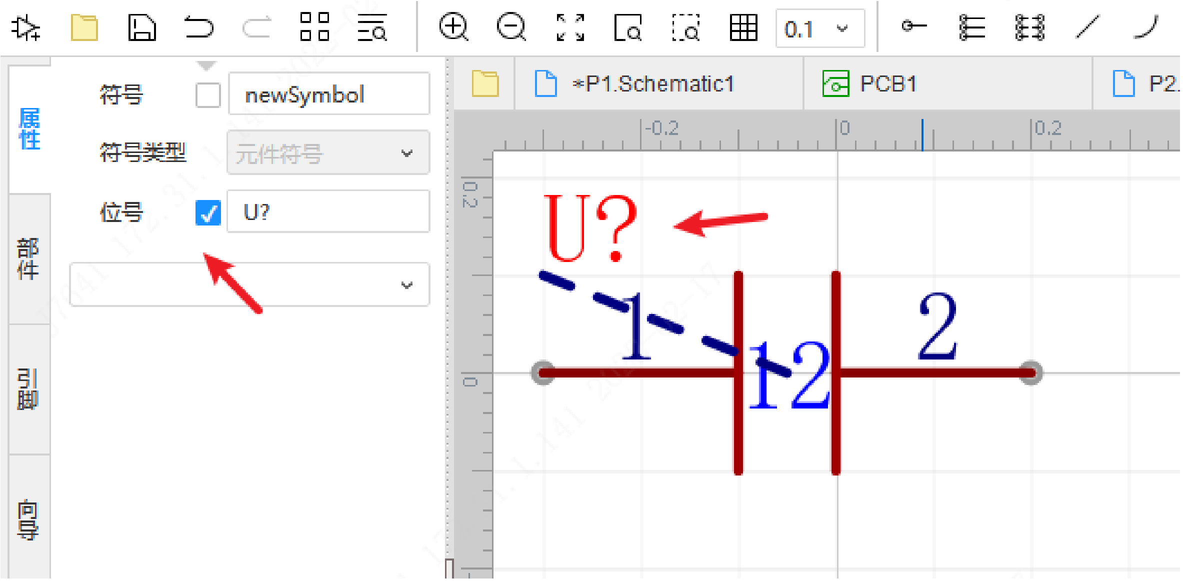
When the canvas is clicked on a property, an indicator line will appear. This indicator can be turned off at: Settings - Schematic Settings.

When the component is rotated, the position of the attribute is automatically adjusted by default, and it can also be turned off in the settings:



You can also set attribute positions in batches through the attribute position function: Top Menu - Layout - Attribute Position.

If you want to preset the location of attributes such as the tag number, you can set the position after displaying the tag number and other attributes on the canvas in the left property panel when creating a new symbol. When placing the device on the canvas, it will be preset according to the symbol. position is displayed, but will be readjusted after rotation.



Component right-click menu

After selecting a component, the right-click menu is supported to provide some commonly used functions such as operations on components.

Figure 4

View product details: The product details page of LCSC Mall will be opened.

View Data Sheet: The web page will be opened according to the link of the attribute "Data Sheet".

Update Device: Opens the Device Update dialog.

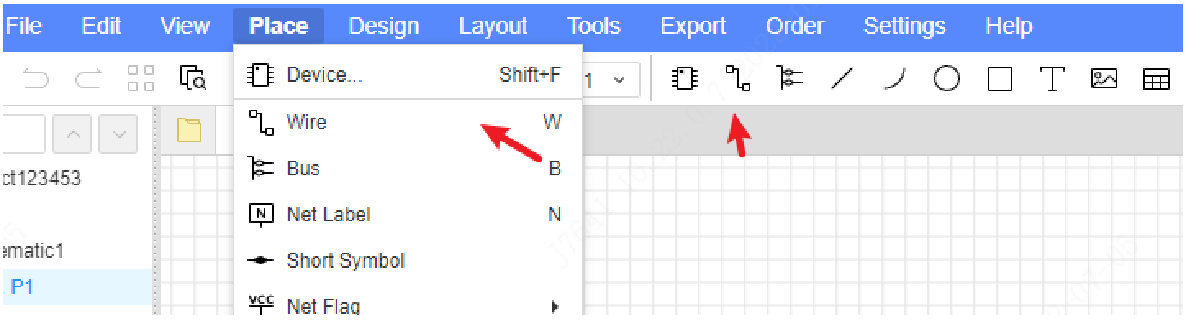
wire/net

wire drawing

There are three modes to enter wire drawing mode.

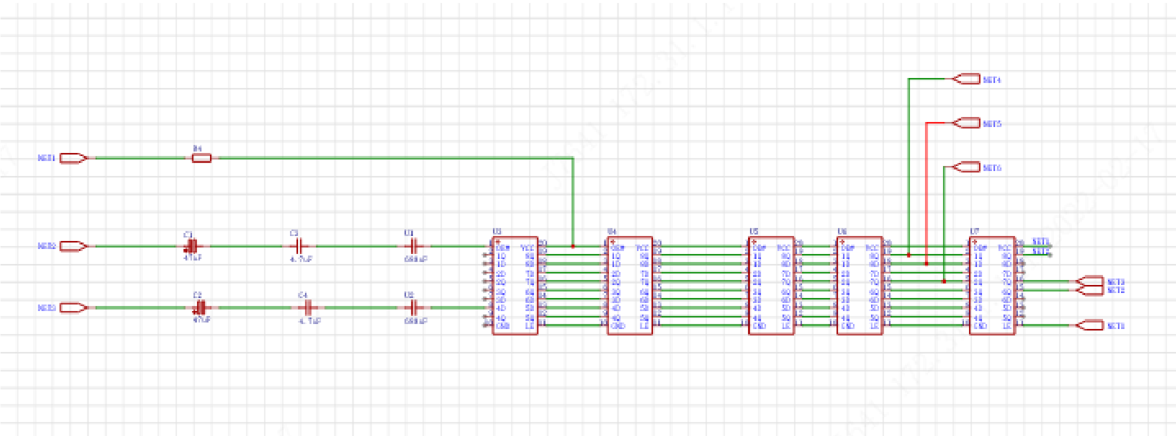
In a schematic, you can draw lines in any direction. Modify line style, color, fill color, etc.

1. Top shortcut bar - Wire
2. Top Menu - Place - Wire
3. Shortcut key W

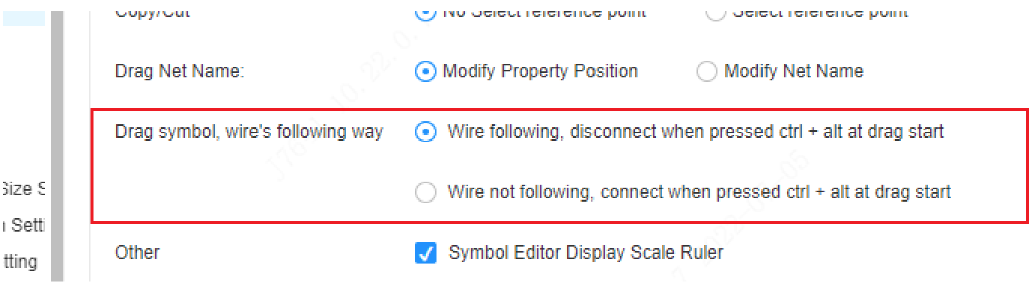


Drawing wire the undetermined segments are translucent to distinguish determined segments.

The figure below is a schematic diagram that has been wired up.

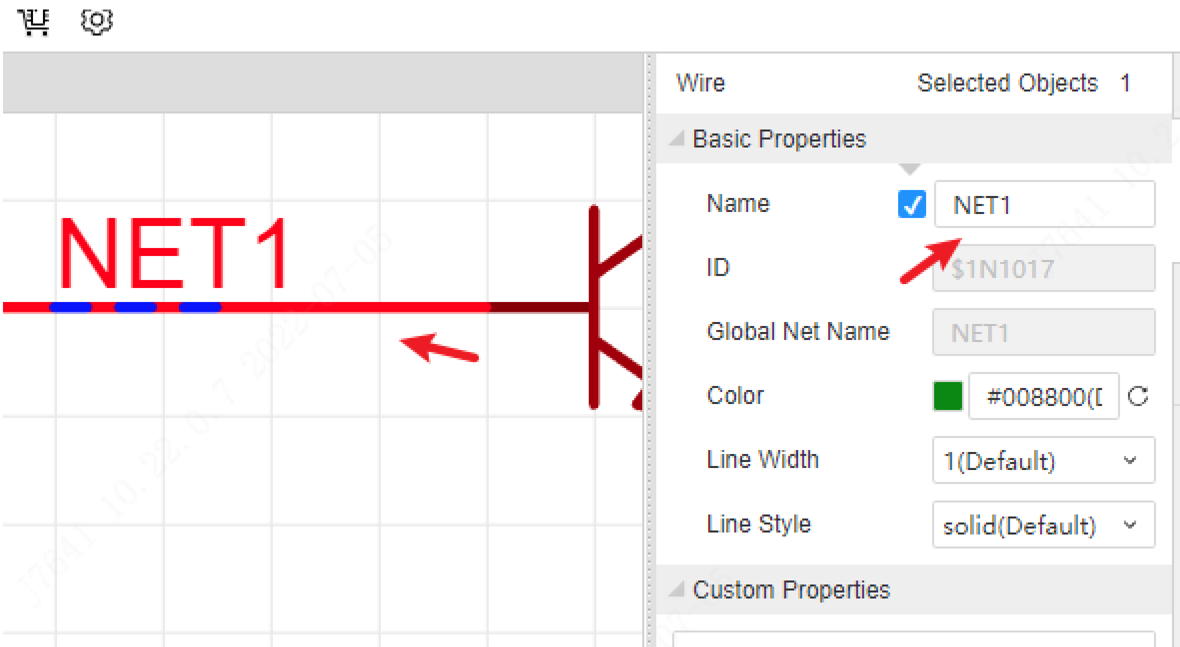


When moving the component, whether the wire follows the support setting item setting:



Wire properties

EasyEDA Pro supports the function of placing net labels. The net label function of the professional version is the same as setting the property name for the wire directly, which is different from the net label of the standard version.



Property description:

- Name: The name of the wire, which is generated according to this property when generating the network name. When the default name is not filled in, the editor will automatically generate a system default network name according to the wire ID.
- Global Net Name: Because the schematic supports hierarchical diagram design. The wire name and global net name of the underlying schematic may not be the same. The global net name is the net name used when going to the PCB.

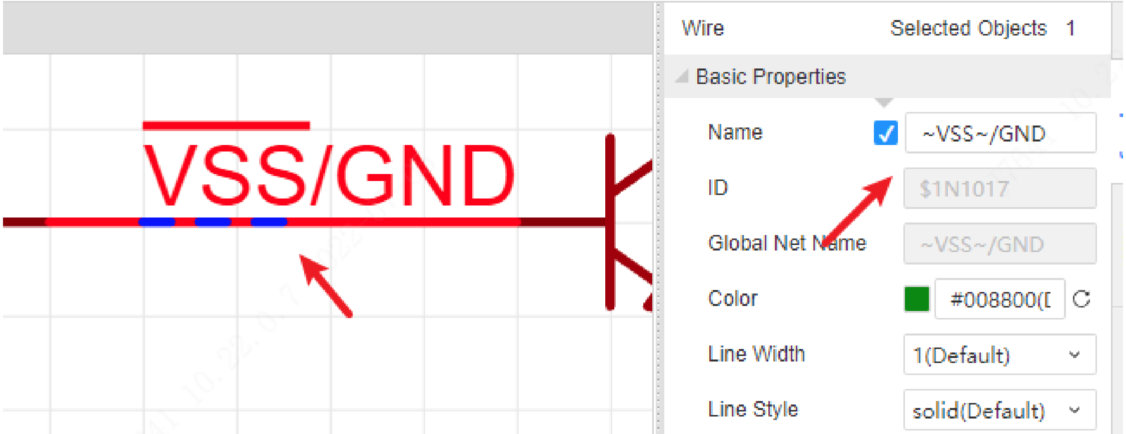
Wire modification name support:

1. Directly double-click the wire name to modify
2. Click on the wire and modify the name in the properties panel on the right

Notice:

- EasyEDA Pro no longer supports multiple network names on one wire at the same time. If you need to connect different networks together, please use a short-circuit character to short-circuit the two networks.
- To add a dash to the wire name, enter a tilde at the top of the wire's net name, such as `~VSS`. If you need to have a network name with an upper dash and no upper dash at the same time, enter a tilde again, for example: `~VSS~/GND`, then there is an upper dash above VSS, but not after the slash. When the net name is transferred to the PCB, it will also be

`~VSS~/GND`.

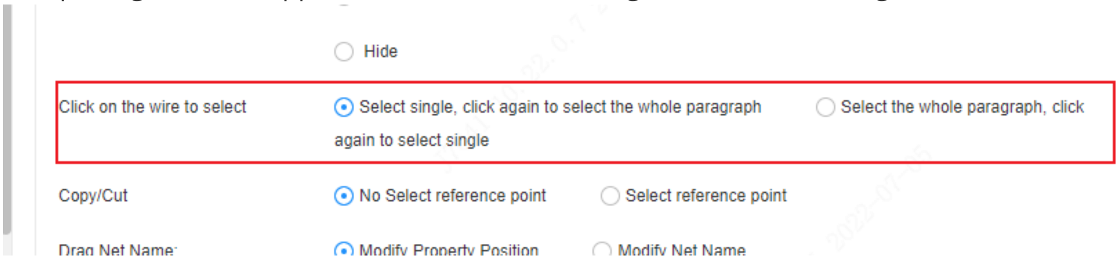


- When a wire is connected to a net ID, the wire name will follow the name of the net ID first.
- When wires are split or merged, a pop-up window will appear to select the net assignment after the operation.

Wire pick selected

EasyEDA Pro supports two wire picking methods:

1. By default, click to pick a complete wire segment, and click again to pick a single segment of wire; or click to select a single segment, and click again to pick a complete wire.
2. This picking method supports modification in Settings - Schematic Settings.

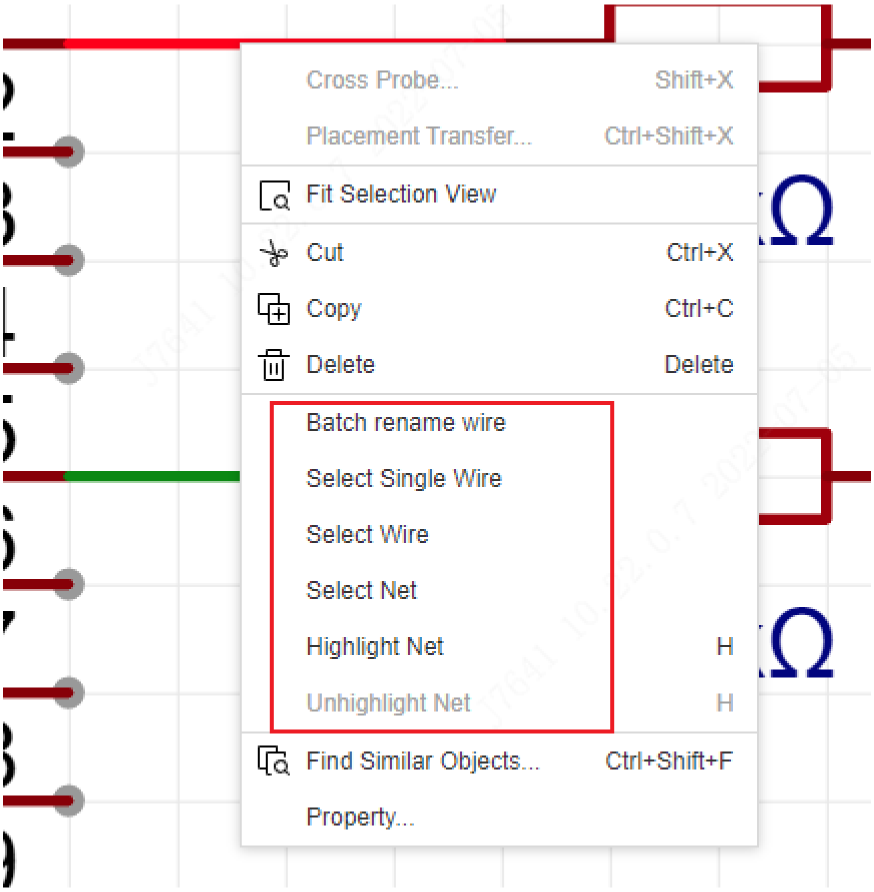


3. When you click to select a single line segment, you can press the TAB key to select the complete wire.

Wire right-click menu

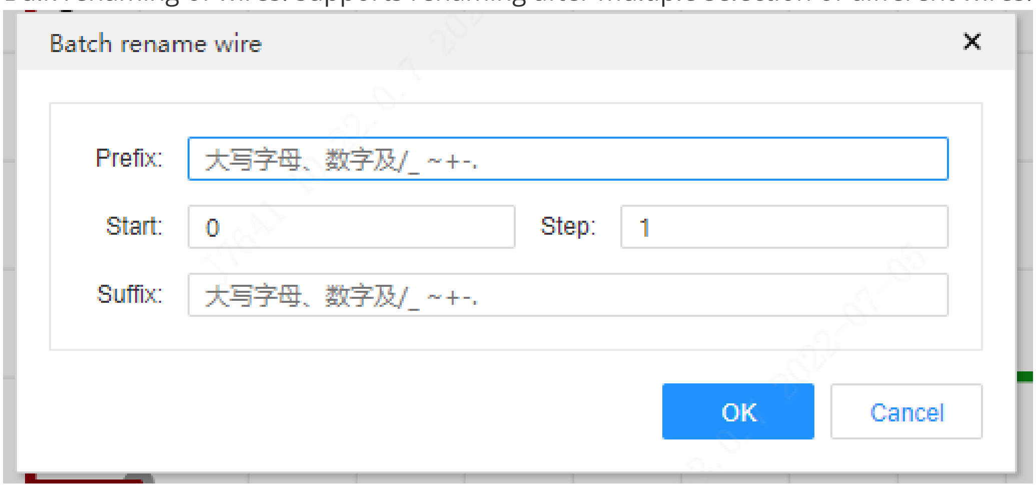
In the schematic design, the right-click menu of the wire is provided, which supports multiple functions to facilitate design and inspection.

After selecting the wire, you need to right-click on the wire to display the right-click menu of the wire.



Menu Description:

- Bulk renaming of wires: supports renaming after multiple selection of different wires.



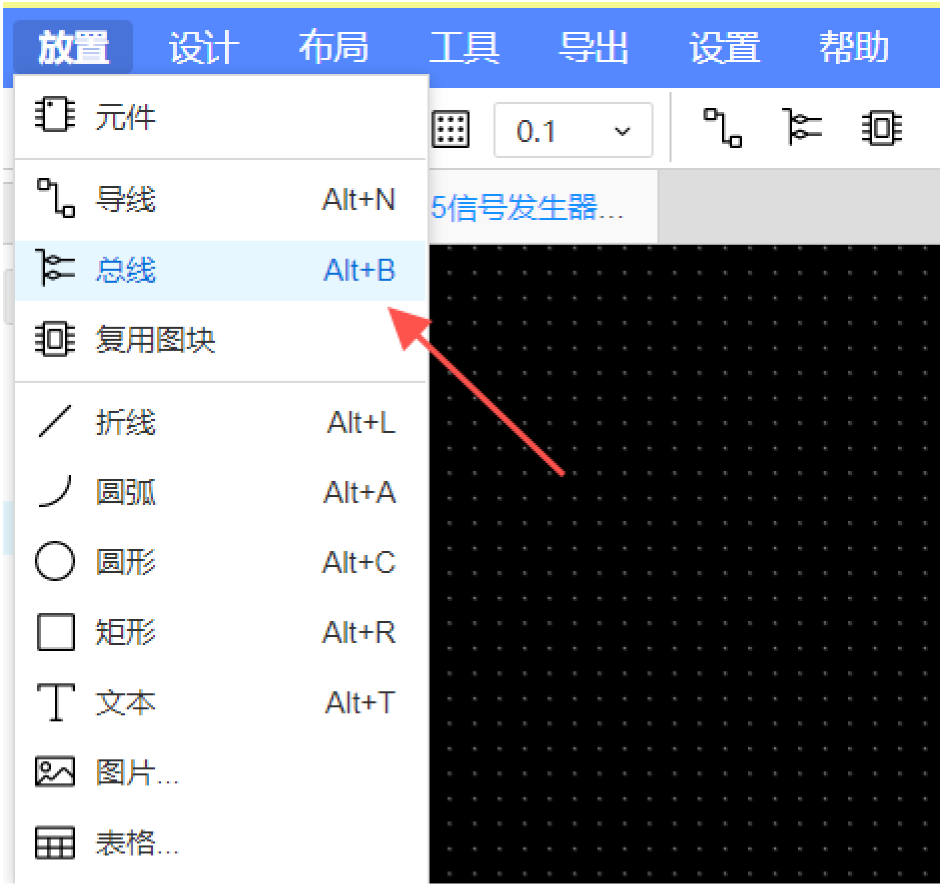
- Select single-segment wire: When the selected wire is an entire wire, this menu supports the selection of a single-segment wire.
- Select Wire: When a single wire segment is selected, this menu supports selection of a complete wire.
- Select Network: Select the current wire of the current drawing page and all wires of the network to which it belongs.
- Highlight Net: All wires with the same net of the current wire are continuously highlighted. The network tab on the left also supports right-clicking the network name to highlight the network.
- De-Highlight Nets: Deactivates all highlighted wires.

bus

When you are drawing a schematic that requires many nets, it will be very difficult and time consuming to draw lines for each net, and you can use the "bus" function.

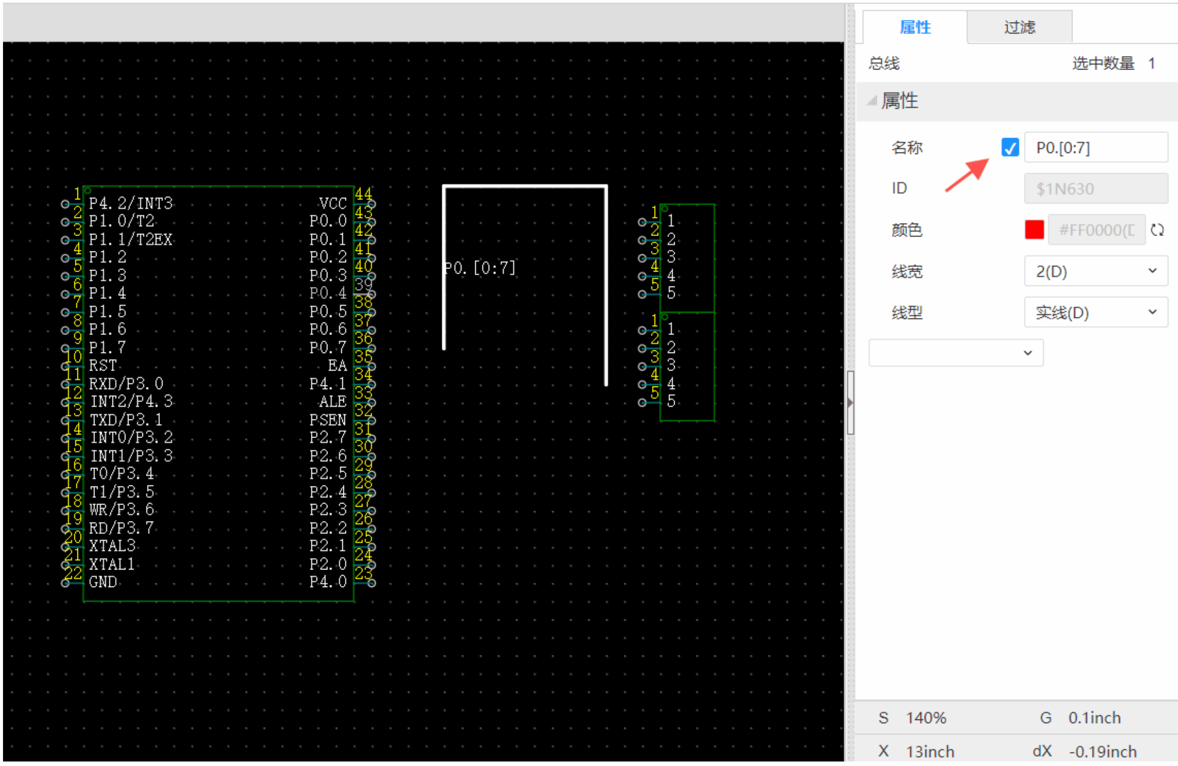
In EasyEDA Pro, the bus function already has the direction of the signal beam, which can be applied to the PCB.

1. Top Menu - Place - Bus;
2. Top shortcut bar - bus icon;
3. Shortcut key - ALT+B;



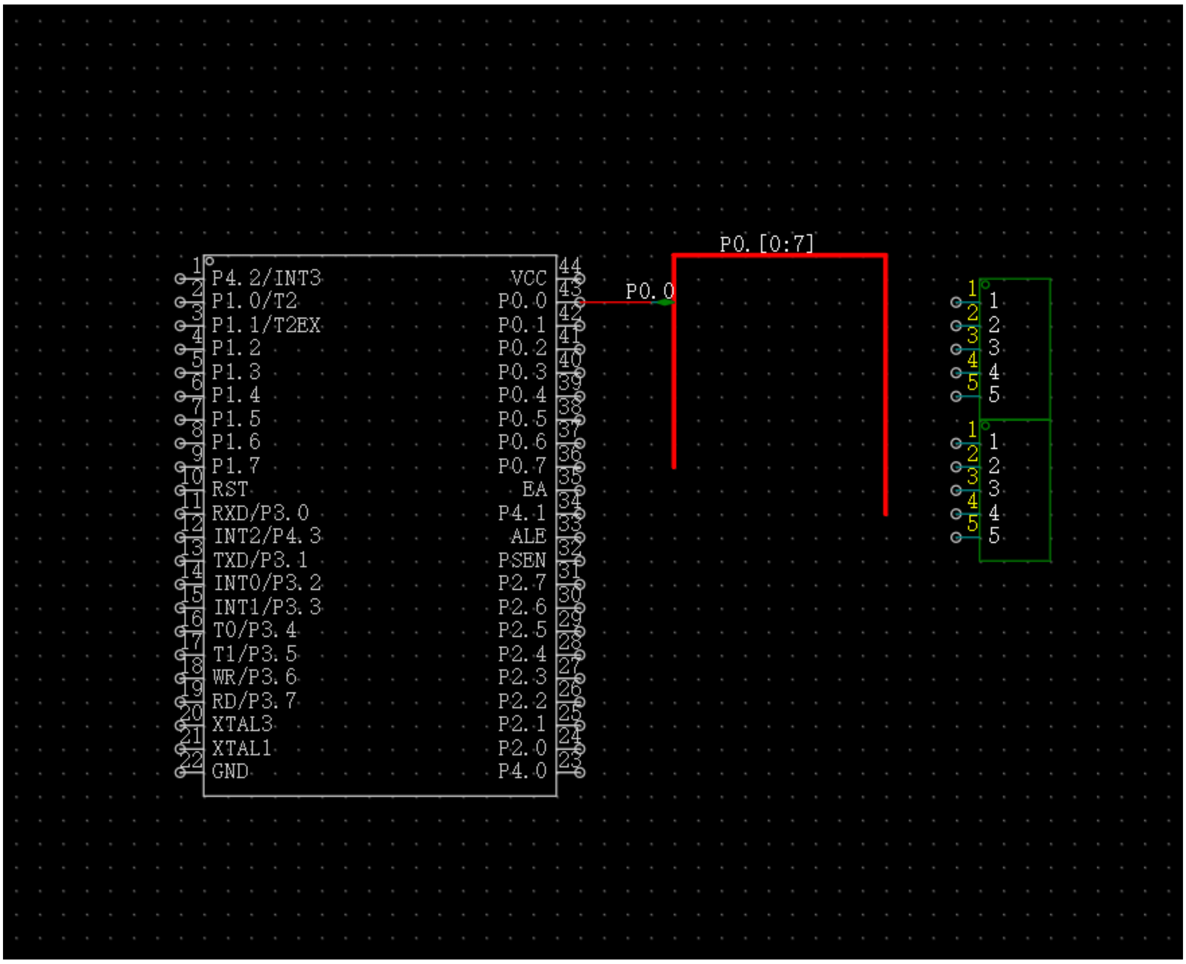
Note: The bus naming of the professional version must conform to the specification `NET[0:5]` in order to be used.

For example, if we need to connect the P0.0 - P0.7 ports of the 51 single-chip microcomputer, we need to use the bus to connect, then we need to name the bus as P0.[0:7], in order to name it successfully.

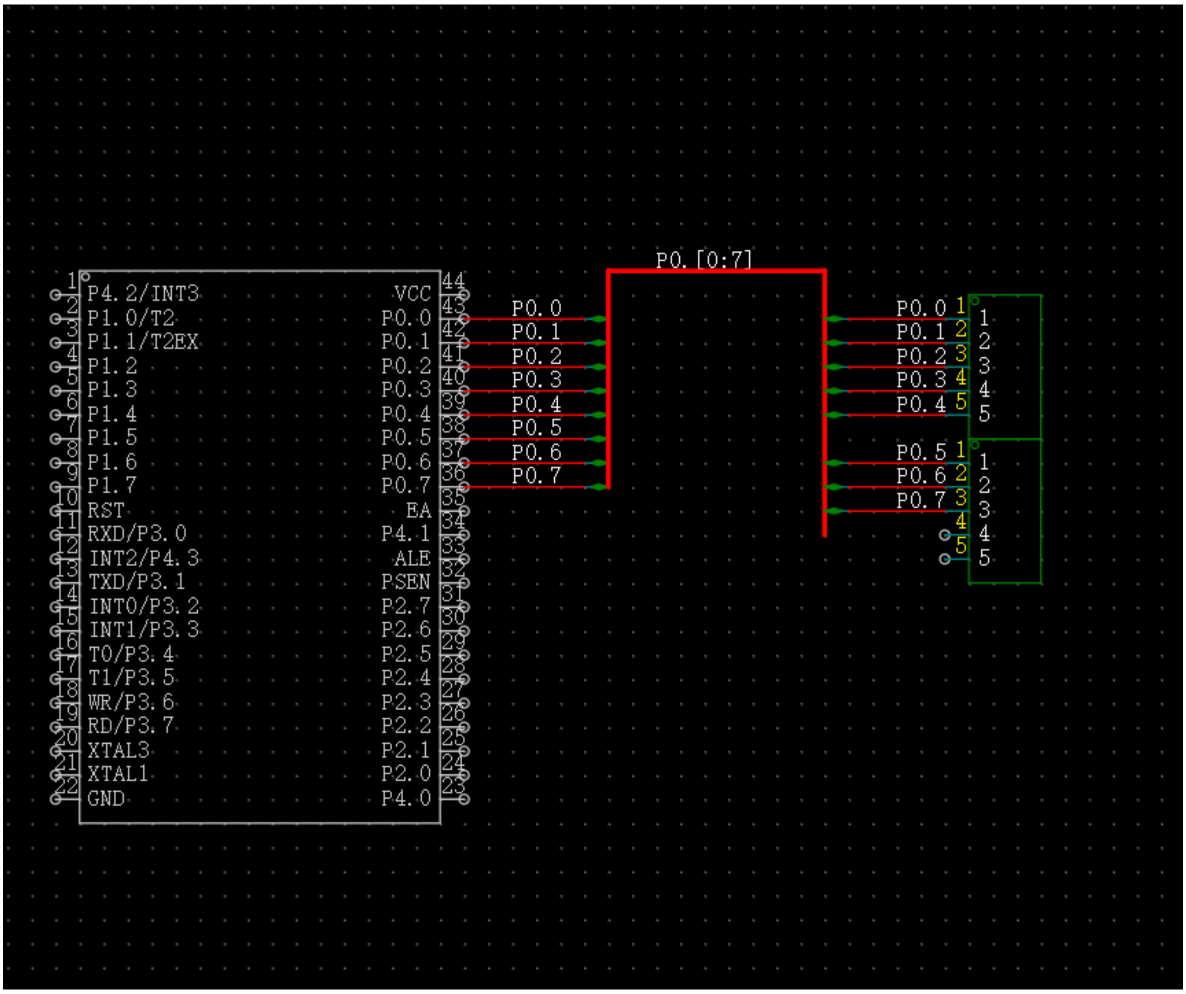


Then select the wire or shortcut key w wire to connect the bus in the top menu. When a short-circuit symbol appears at the bus end, it means the bus connection is successful.

The order in which the bus is pulled out is sorted and placed according to the name you named, and if the number exceeds the number, it will be arranged from the first one.



The bus after the connection is complete.



Net Label

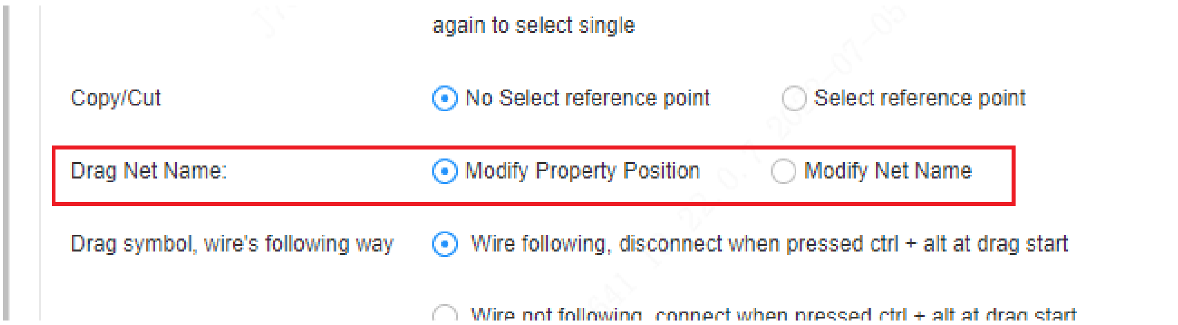
Due to the difference in design, EasyEDA Pro does not support the independent network label primitives of the standard version. The network label of the professional version is virtual, and a name attribute is set to the wire after Place. Therefore, the interaction after Place is quite different from the standard version.

In terms of interaction, since v1.7, EasyEDA Pro supports two modes:

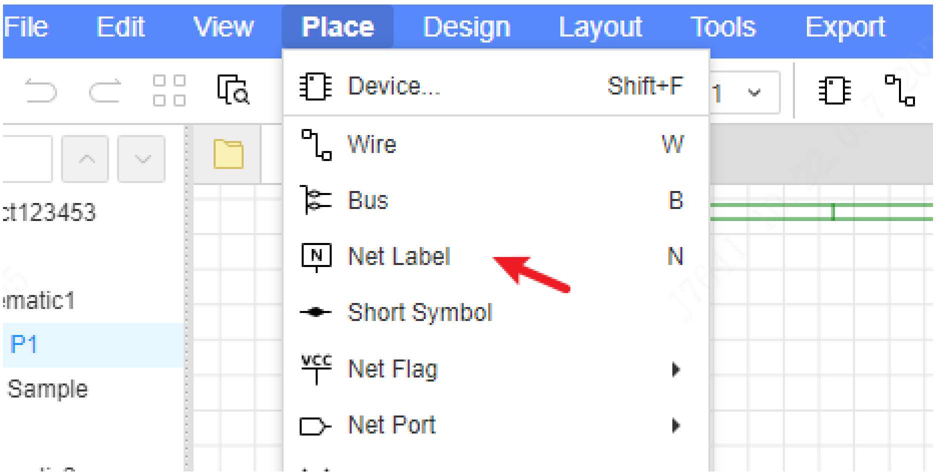
1. Similar to the standard version or Altium's network label, after the network label leaves the wire after moving, the wire network name is automatically cleared
2. Similar to the network labels of PADS and Orcad in the previous professional version, after the network label is moved away from the wire, the network name of the wire remains unchanged, and only the position of the network name is modified.

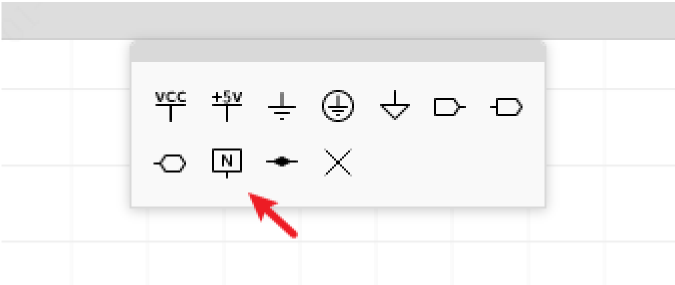
You can set it according to your usage habits.

Setting entry: Top Menu - Settings - Schematic - Common - Drag Net Name

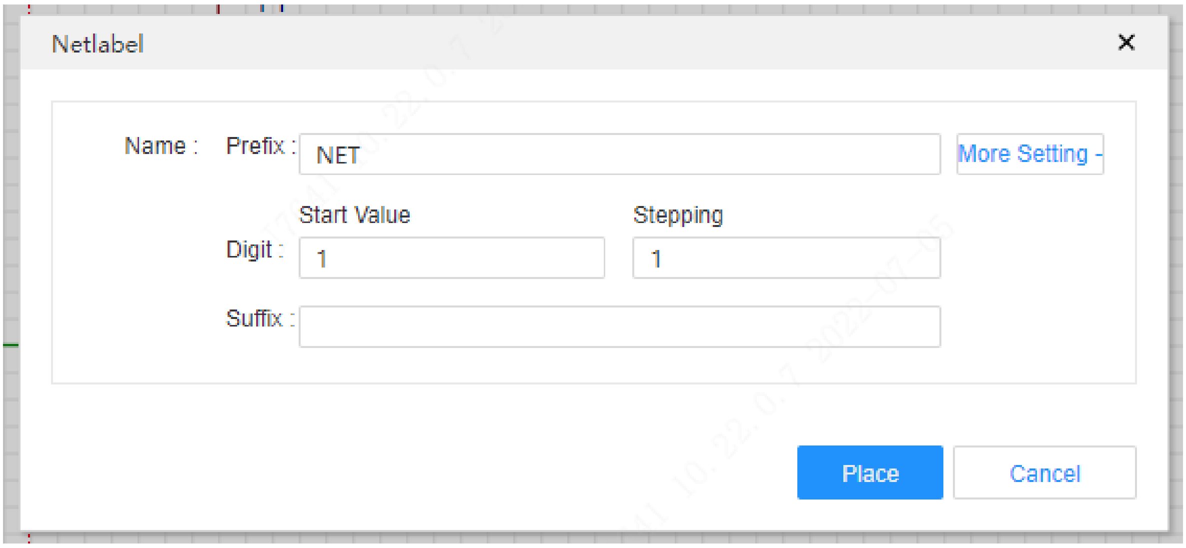


Entry: Top Menu - Place - Net Label



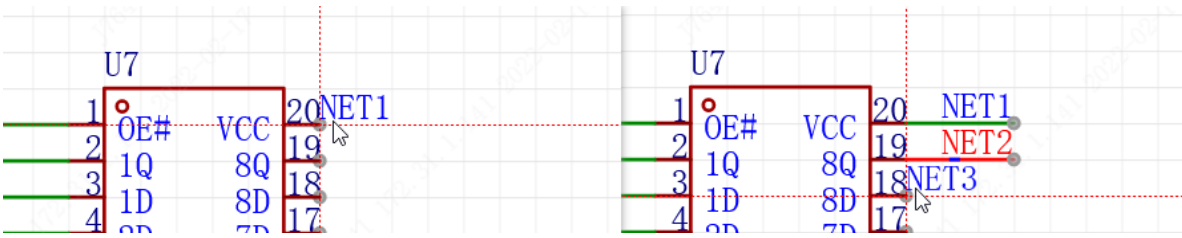


When placing, you can press the TAB key to set the name of the network label and the increasing order rule.



After being placed, it becomes the name attribute of the wire, which is the same as setting the name directly in the wire's attribute panel.

Supports direct Place on the pin of the symbol, a wire will be automatically generated, and the wire will be given a name.

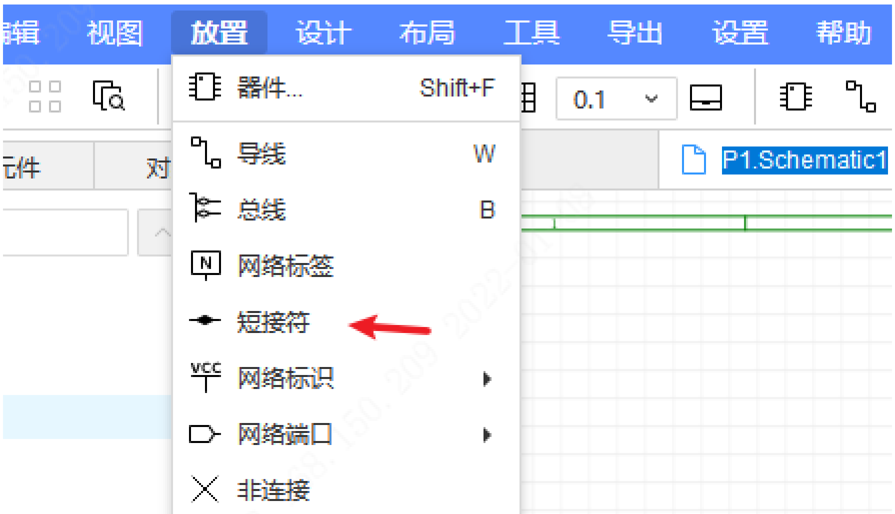


Short Symbol

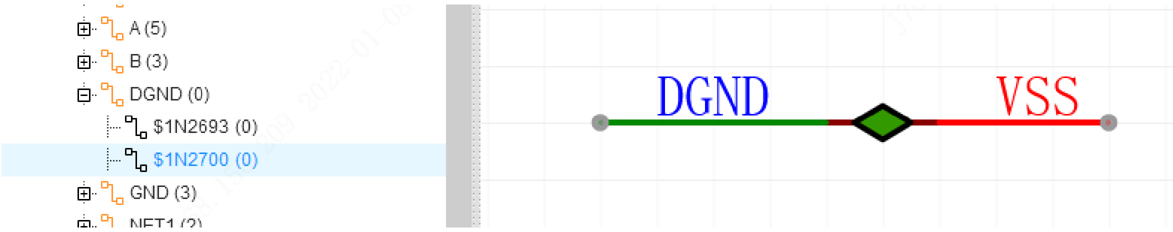
EasyEDA Pro does not support setting or placing two different net names on the same wire at the same time.

Two different networks can be connected together by using a shorting character. When generating a netlist and transferring a PCB, one of the network names (naturally sorted by alphabet) will be taken as the final network name.

Entry: Top Menu - Place - Short Symbol.

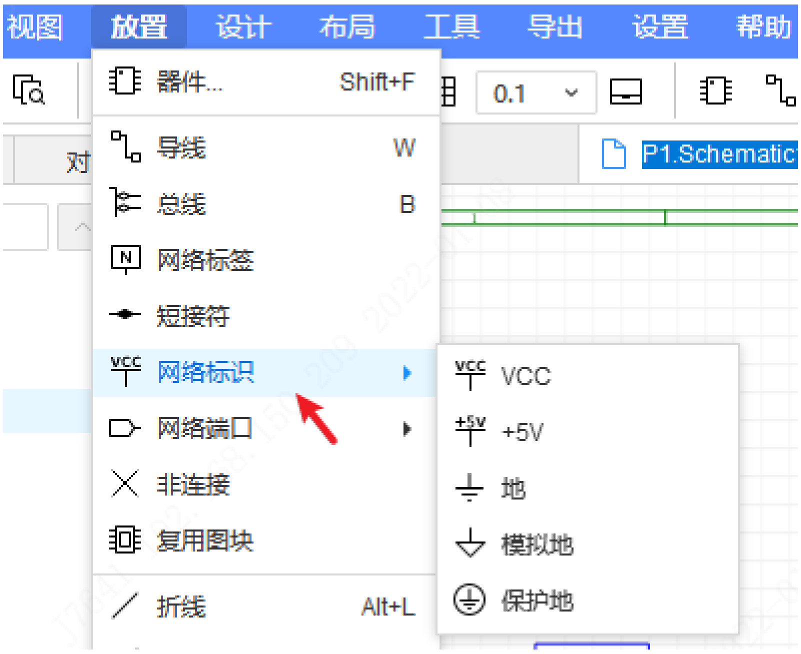


As shown, VSS and DGND will be connected together, which can be viewed in the global net in the tree on the left. The final net name will be DGND.

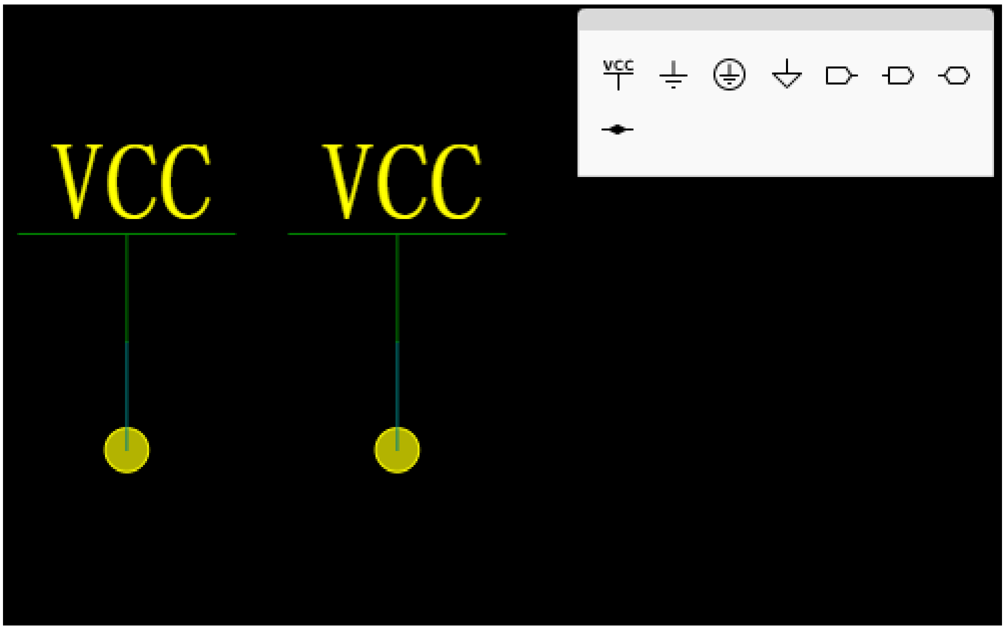


Net Flag

EasyEDA Pro has built-in five commonly used network identifiers. If the symbol style does not meet the needs, you can create a new symbol - network identifier by yourself.

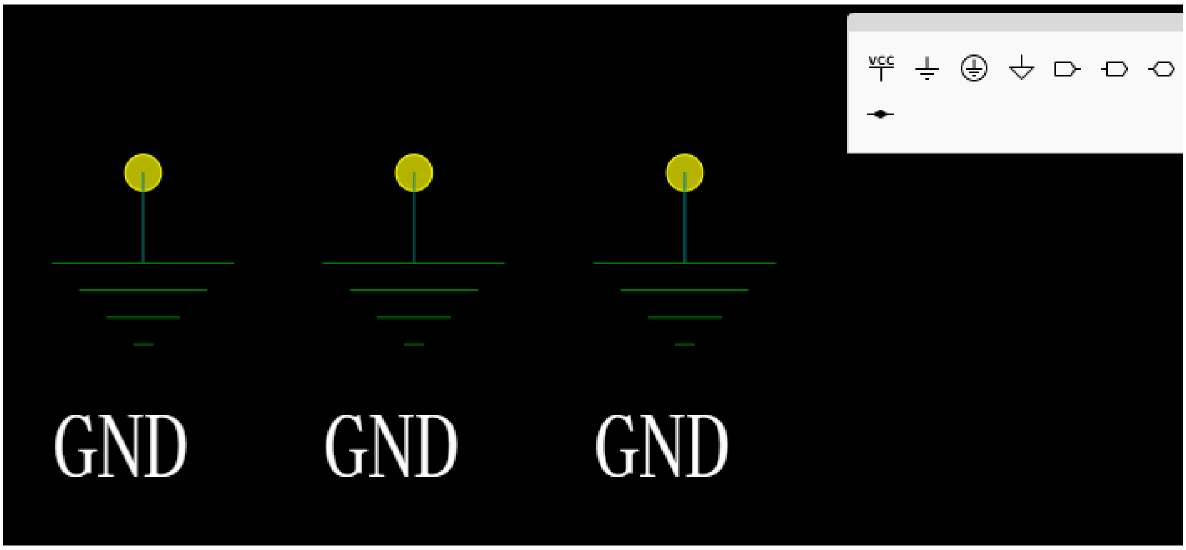


Add a net flag for VCC to the schematic.



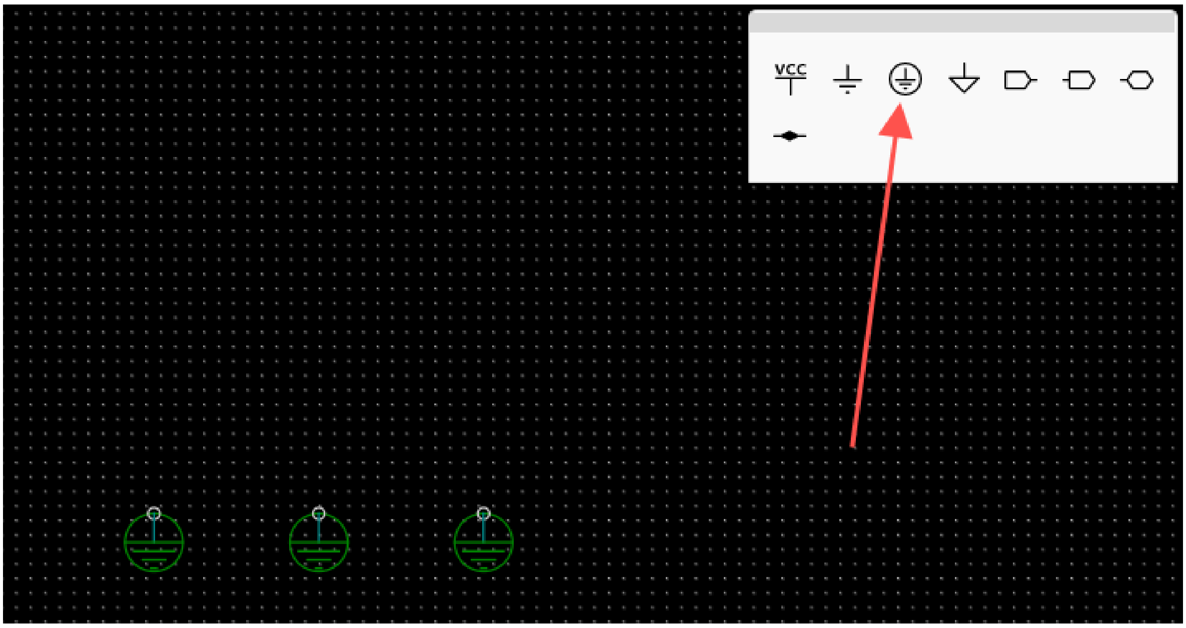
GND net flag

Place GND net flag



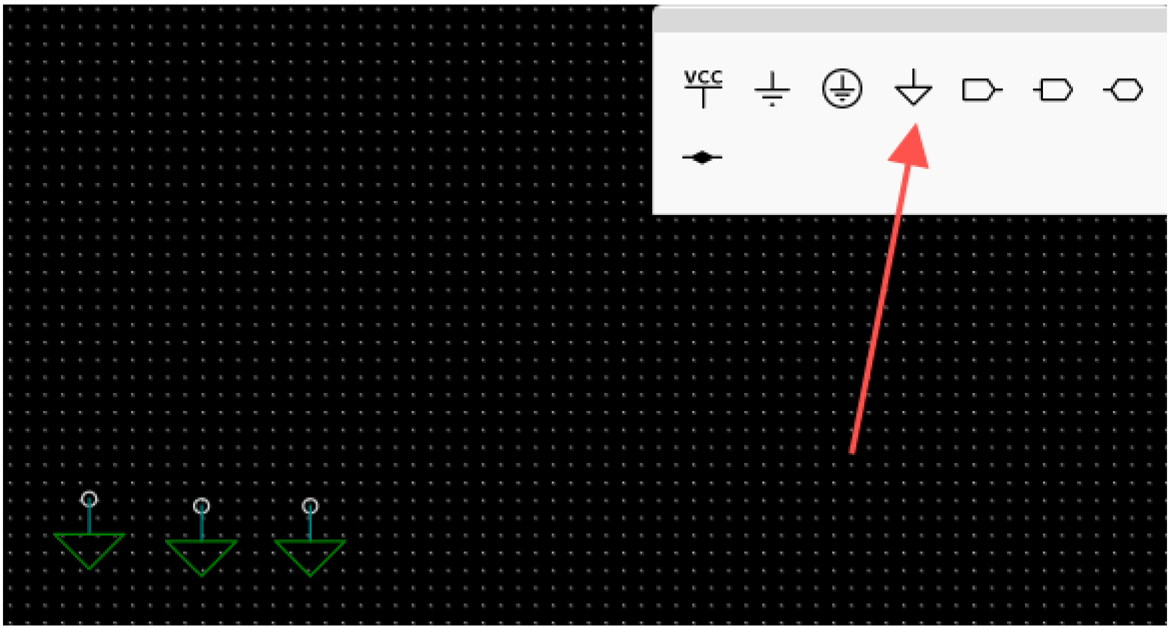
Protected ground

Putting a protective ground and GND net flag is similar,



analog ground

Used to distinguish signal ground and digital ground.



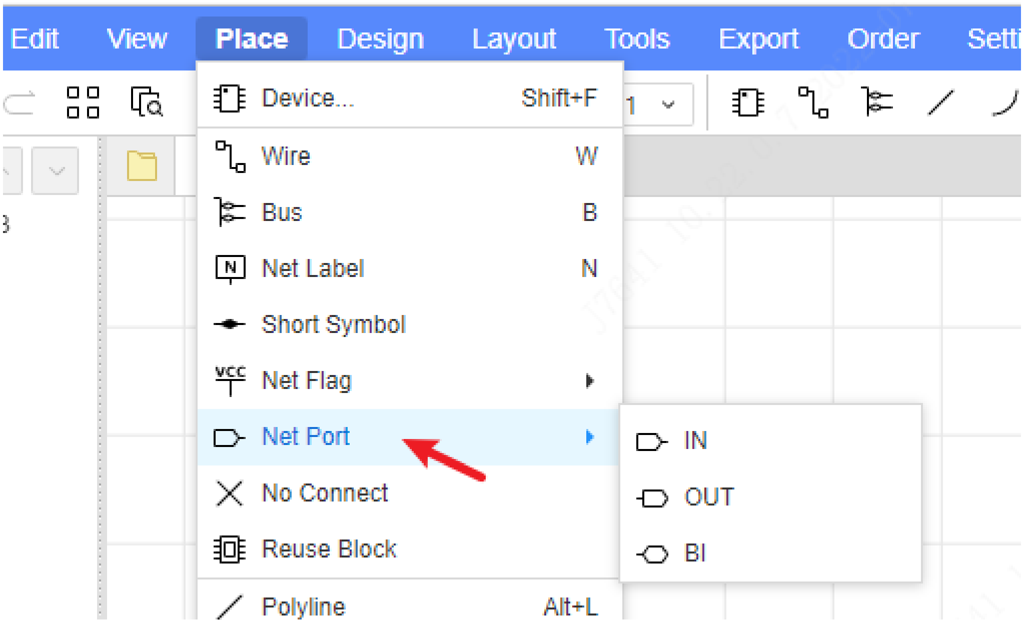
net port

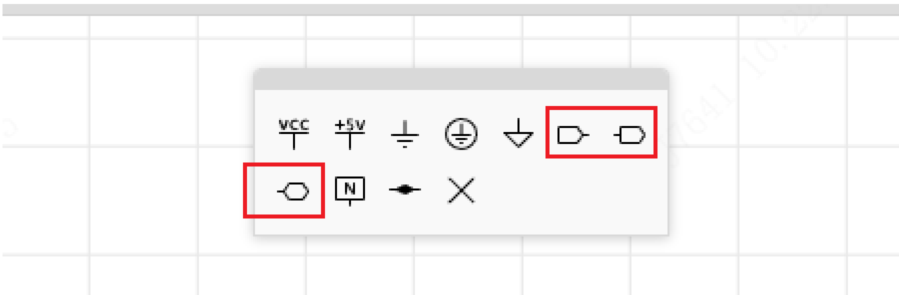
net ports serve two purposes:

1. Interconnection of the network in the picture page
2. Reuse the connection of block/hierarchy diagram

It is recommended to reduce the use of the first use, net ports are often used for hierarchical graph connections.

Entrance: Top Menu - Place - Net Port.

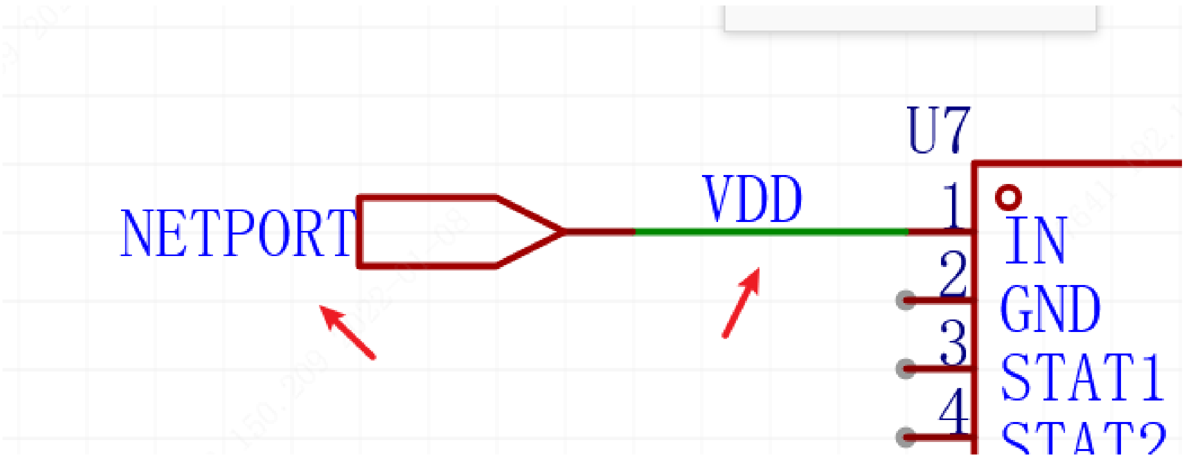




Network Connection: The net port is used as a network label.

Hierarchical graph connection: When drawing a multiplexed block, the pins of the multiplexed block symbol will be connected to the ports on the map page as a connection relationship. When a port is generated from the multiplexed tile symbol, or the multiplexed tile symbol pin is updated based on the port, it is updated synchronously. For details, see the "Multiplexing Blocks" chapter.

Because the net port is mainly used for the connection relationship in the hierarchy diagram, the name of the port and the name of the connected wire may not be consistent. As shown in the figure:

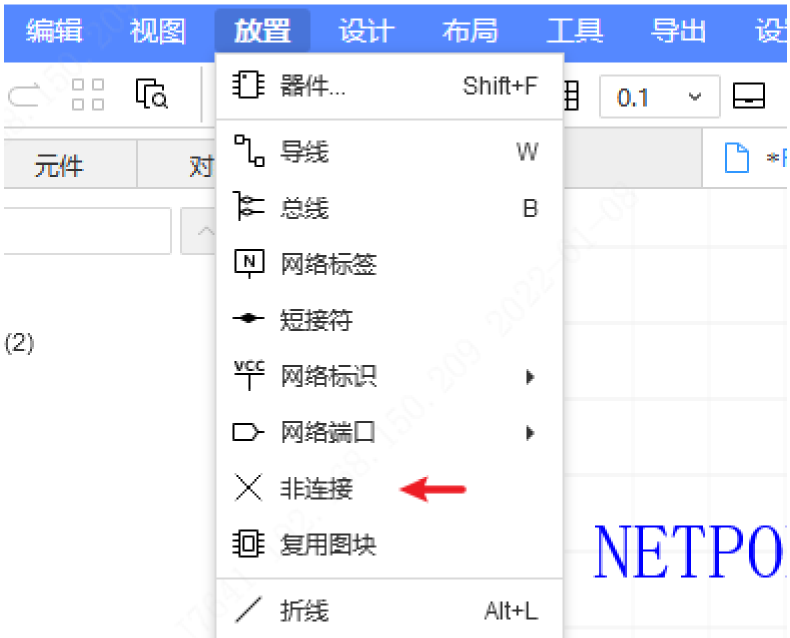


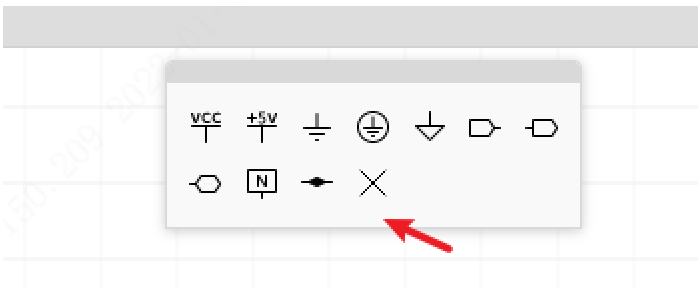
When changing the name of a net port, or when placing a port on a wire for the first time, if the wire does not have a name set, the wire is automatically given the same name.

No Connect

EasyEDA Pro's unconnected flags are similar to network labels, and do not support separate unconnected primitives like the standard version.

Entry: Top Menu - Place - No Connect





No Connect symbols can only be placed on pin endpoints. After Place, the attribute "No Connect" is automatically set to yes for the pin of the symbol. When the No Connect flag is deleted, the attribute "No Connect" will be changed to no.

Reuse Block/Hierarchy

When drawing complex schematics, you need to use multiplexed blocks. The design of multiplexed blocks is the same as that of hierarchical schematic diagrams. For details, see Create Reuse Blocks chapter.

The reuse block is divided into two main schematic diagrams and sub-schematic diagrams. The main schematic diagram is used to draw block symbols and type flow charts, while the sub-schematic diagram is used to draw a module schematic diagram. The overall concept is that the main circuit diagram is equivalent to the entire circuit diagram. The block diagram in the machine circuit diagram, a block diagram is equivalent to a module, no device can be placed in the main circuit diagram, only pins can be placed to connect to the subcircuit.

Generally, the sub-circuit diagrams are some specific circuit schematic diagrams. The connection between the sub-circuit diagram and the main circuit diagram is realized through the ports in the block diagram.

The design idea of reuse block is usually a top-down design method. The idea is: first design the symbol of the multiplexed block, and then follow the symbol to generate a sub-circuit diagram. The bottom-up design is to first design the circuit and plan the port in the sub-schematic diagram, and then generate a main diagram to reuse the block symbol.

Polyline

Polylines are different from wires. Polylines have no electrical properties, while wires have electrical properties. Although the two are the same, the actual usage is different. Polylines cannot be placed in place of wires.

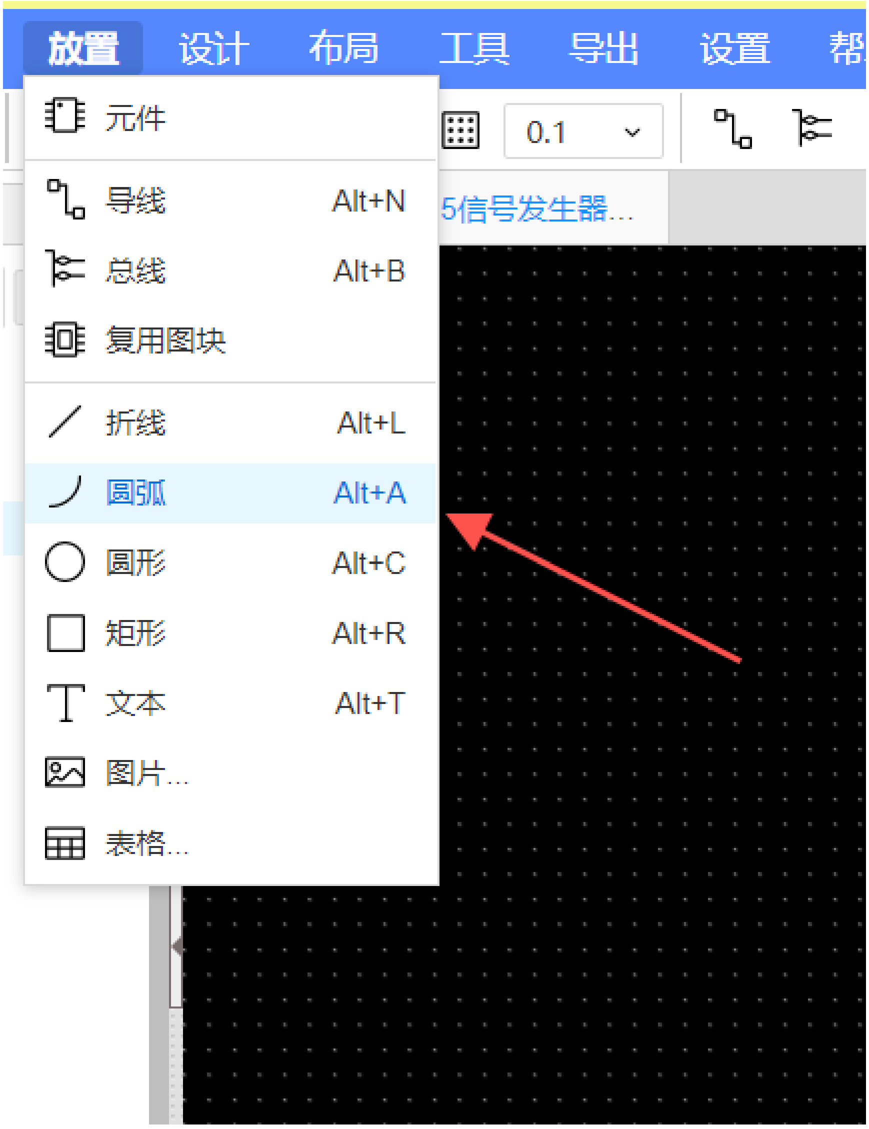
In a schematic, you can draw lines in any direction. Modify line style, color, fill color, etc.

1. Top shortcut bar - Polyline.
2. Top shortcut bar - Place - Polyline.
3. Shortcut key AIT+L.

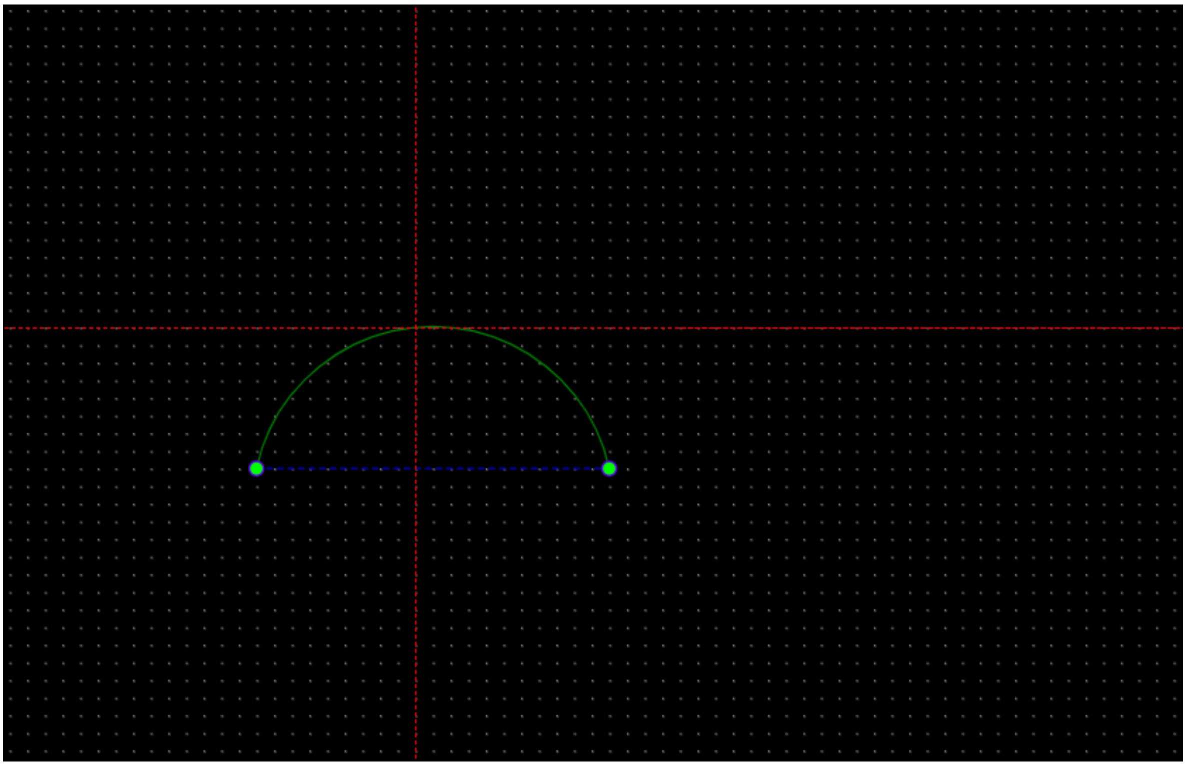
arc

Supports drawing arcs of different shapes.

Operation steps: Top menu - Place - Arc or shortcut key Alt + A



The arc is drawn for a three-point arc, and two base points need to be drawn first, and then the third point is the stretch point.



Click the left mouse button to complete the current arc drawing, and select the drawn arc;

Keep the mouse in the state of drawing an arc, left-click again, draw the starting point of the next arc, and select the next arc after the next arc is drawn.

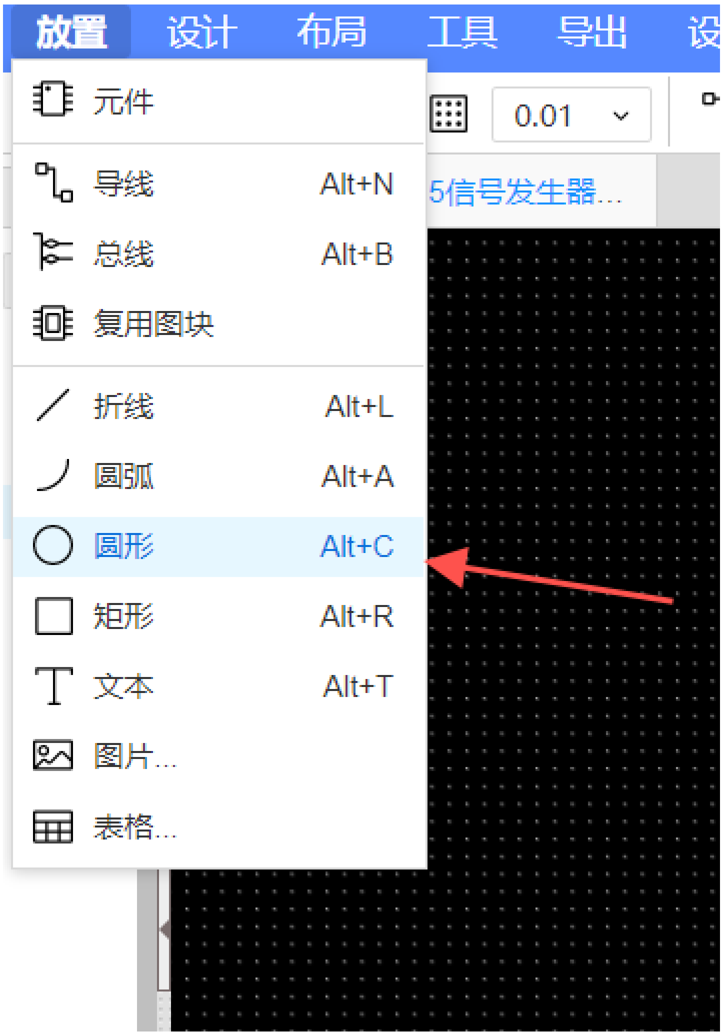
Esc exits the arc drawing and returns to the mouse selection mode; if there is an unfinished arc, clear the points of the unfinished arc and restore the mouse state to the selection mode.

Select the arc to display 3 points, left-click to drag the points to modify the arc.

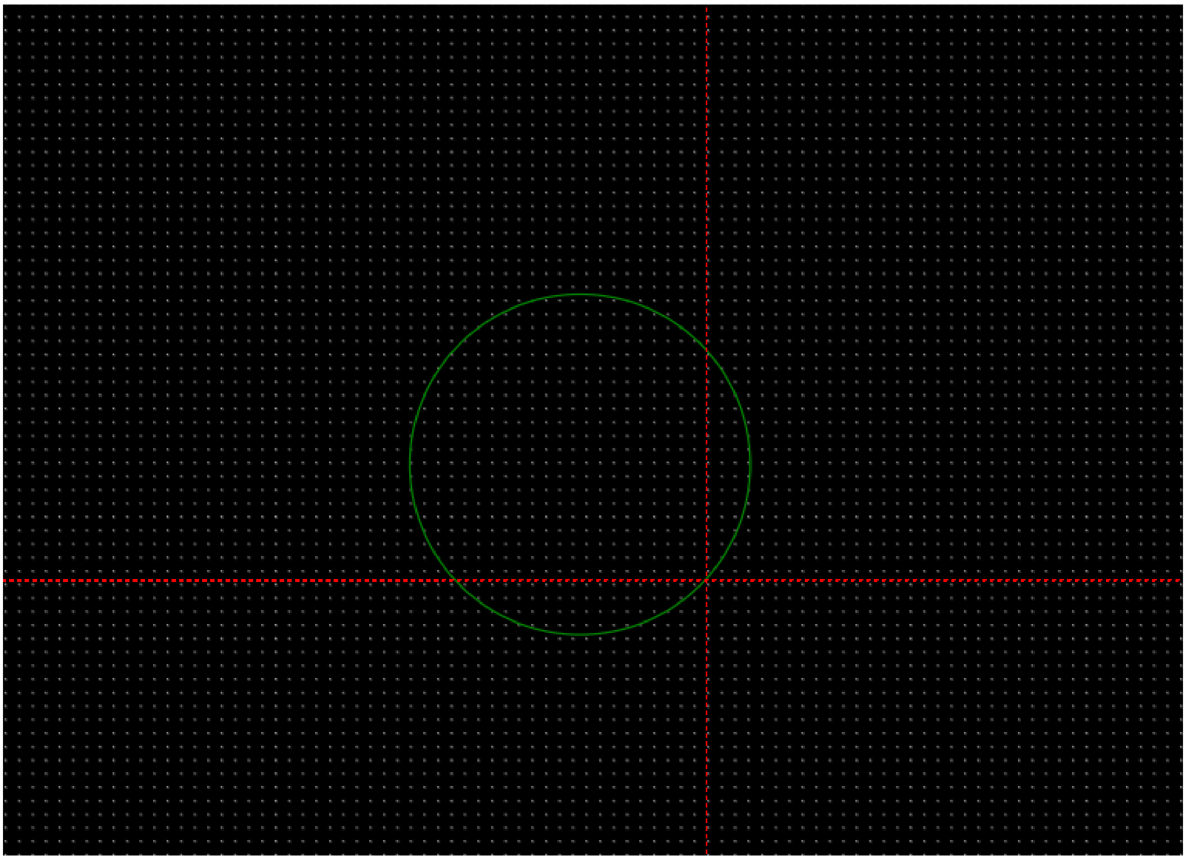
Circle

You can draw a circle in the schematic diagram, and the drawing method of the circle is the center circle.

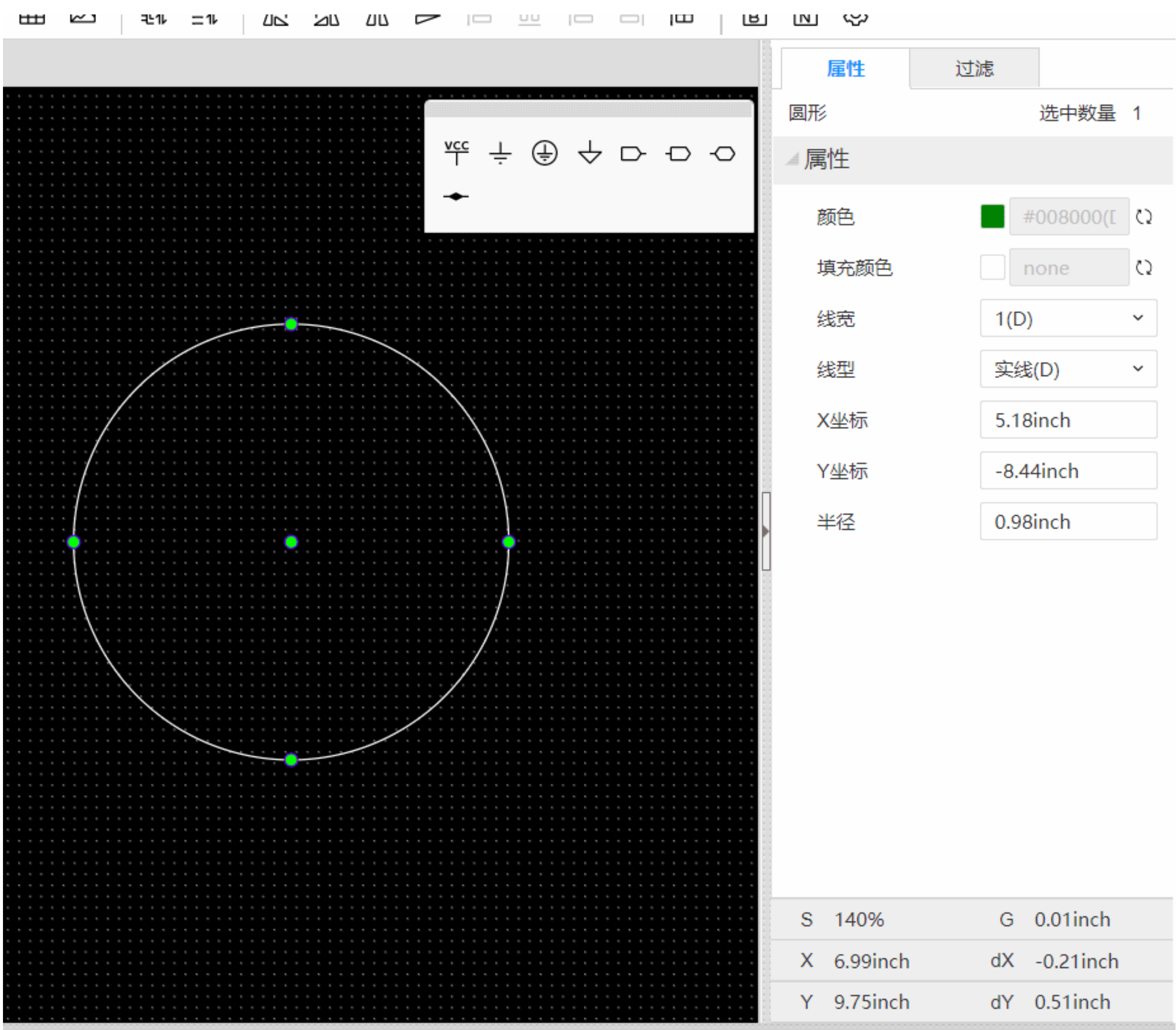
Top Menu - Place - Circle or shortcut alt + C



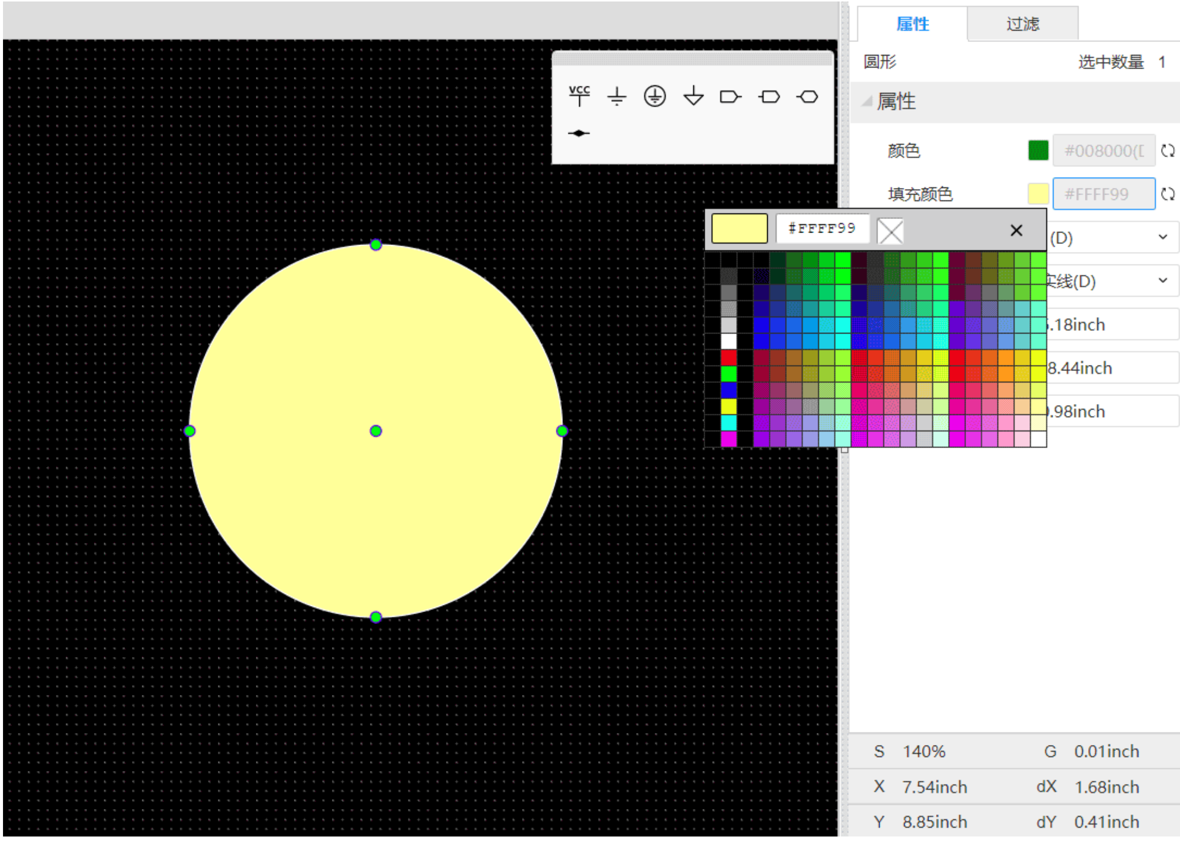
First determine the center point of the circle, then stretch it out, and click the left button of the mouse to complete the drawing of the circle.



Selecting a circle can also modify the color, line width and other settings in the properties panel on the right.



You can also increase the fill color.

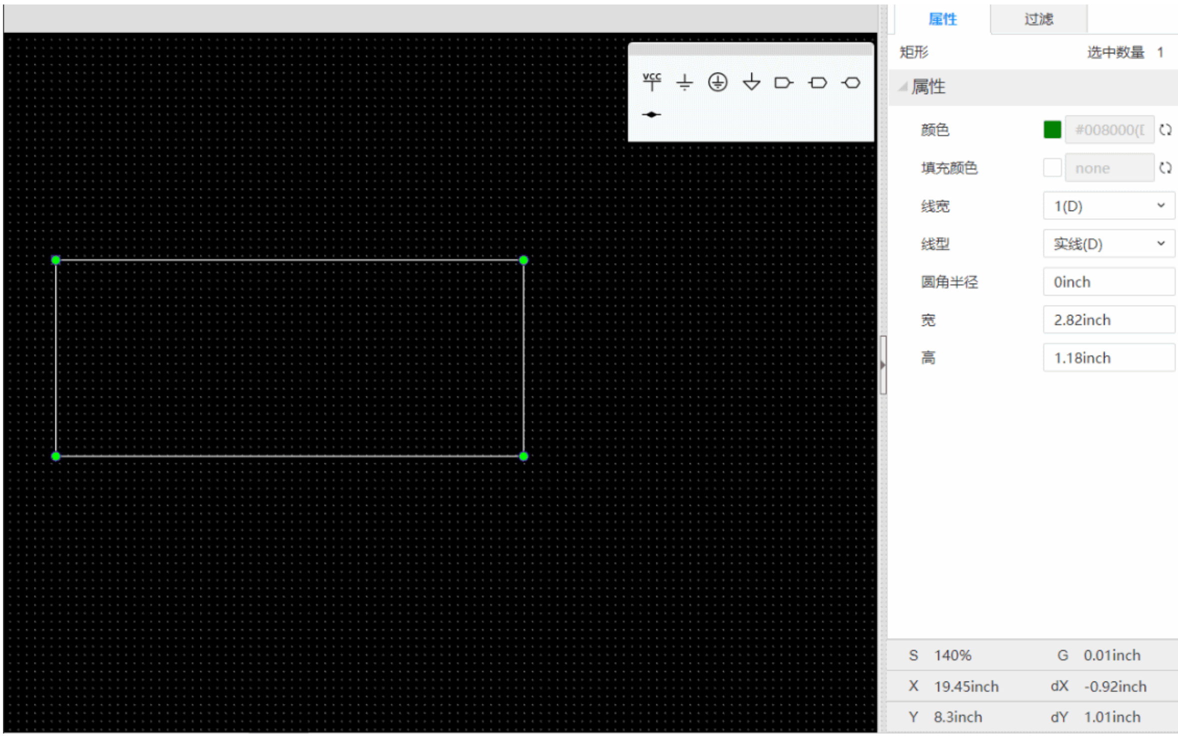


rectangle

Via: Top Menu - Place - Rectangle or shortcut alt + R

Left-click anywhere in the drawing area, set the endpoint position of the rectangle, move the mouse to display the preview effect, click again to end the drawing, and select the rectangle.

Move the mouse, click the left button to determine the end point of the next rectangle, Esc exits the rectangle drawing mode, the unfinished rectangle is deleted, and the mouse state is restored to the selection mode, and the right panel can be set to add properties after selecting.



text

Text can provide text descriptions in the schematic for the description settings of the device or schematic.

Top Menu - Place - Text or Shortcut Alt + T

Within the same schematic, the editor will remember the property parameters of the last text and automatically apply the last property when placing it.

Displays the text settings window (initially empty).

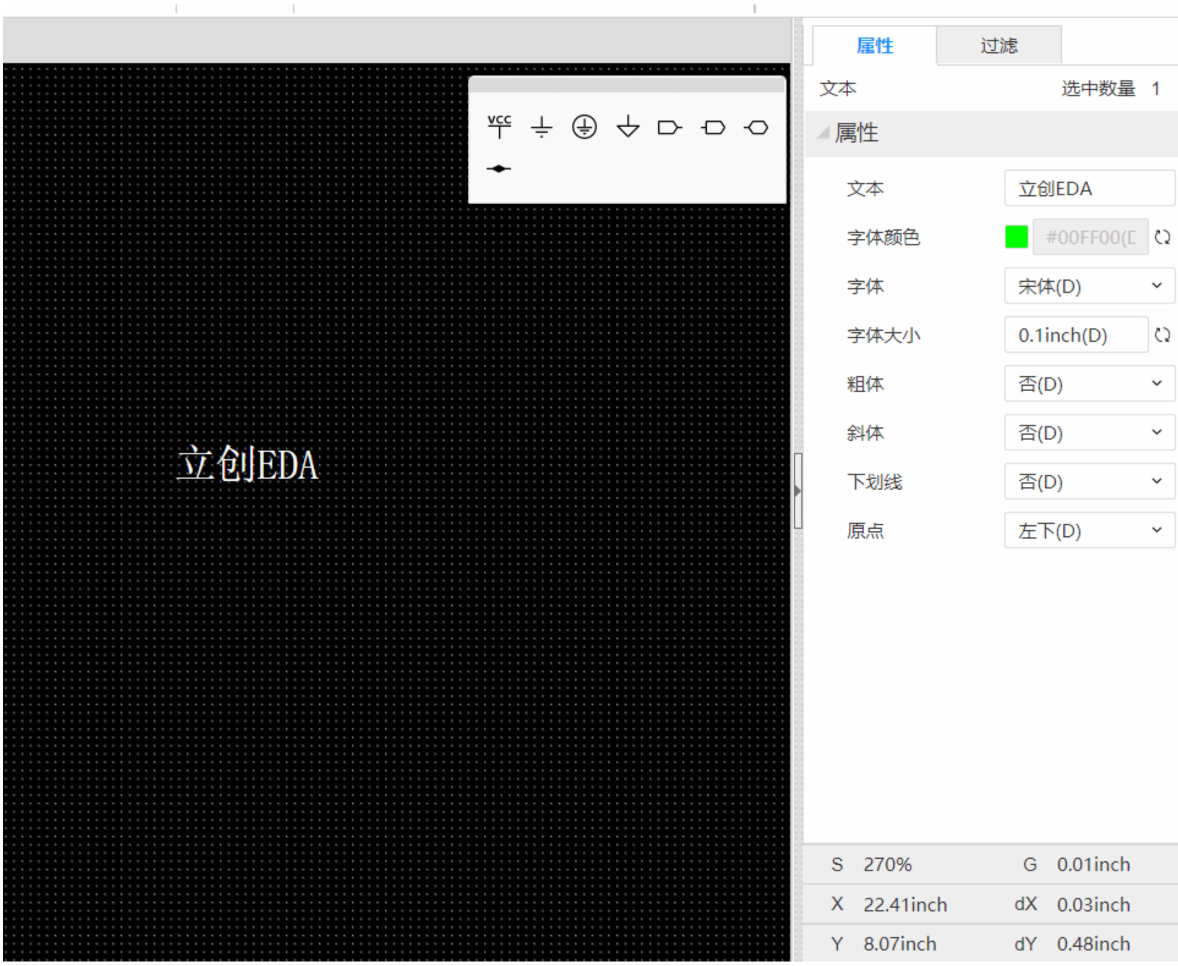
image-20210518152736518

Enter text in the text box, click OK to generate a preview, and click the left mouse button to place it.

image-20201219142704545

image-20201219142828228

Select the text to set some properties of the text in the property panel on the right.

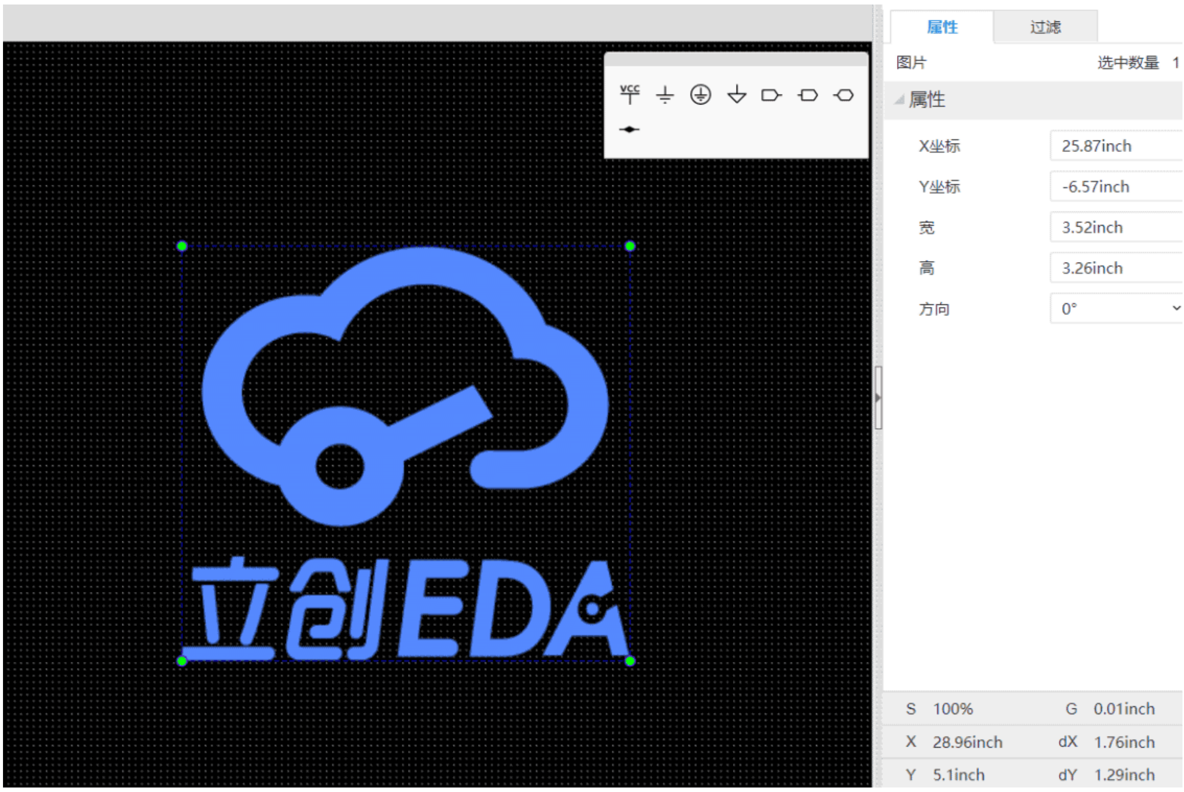


image

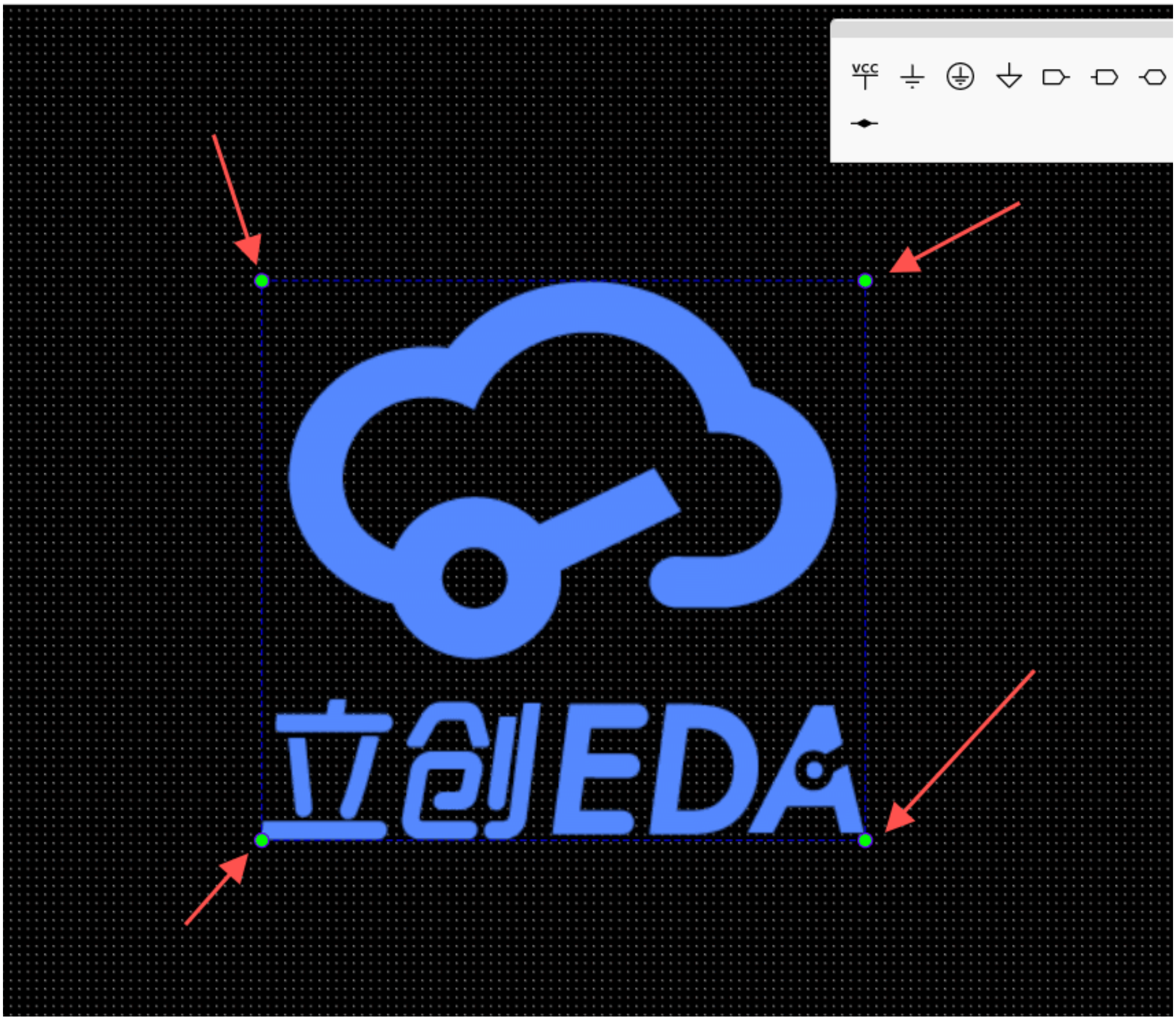
Top Menu - Place - image

After selecting the images, you can generate a preview and click the left mouse button to place it.

Select the image to set the image in the property panel on the right.



You can also click any of the four corners of the image to adjust the proportional zoom.



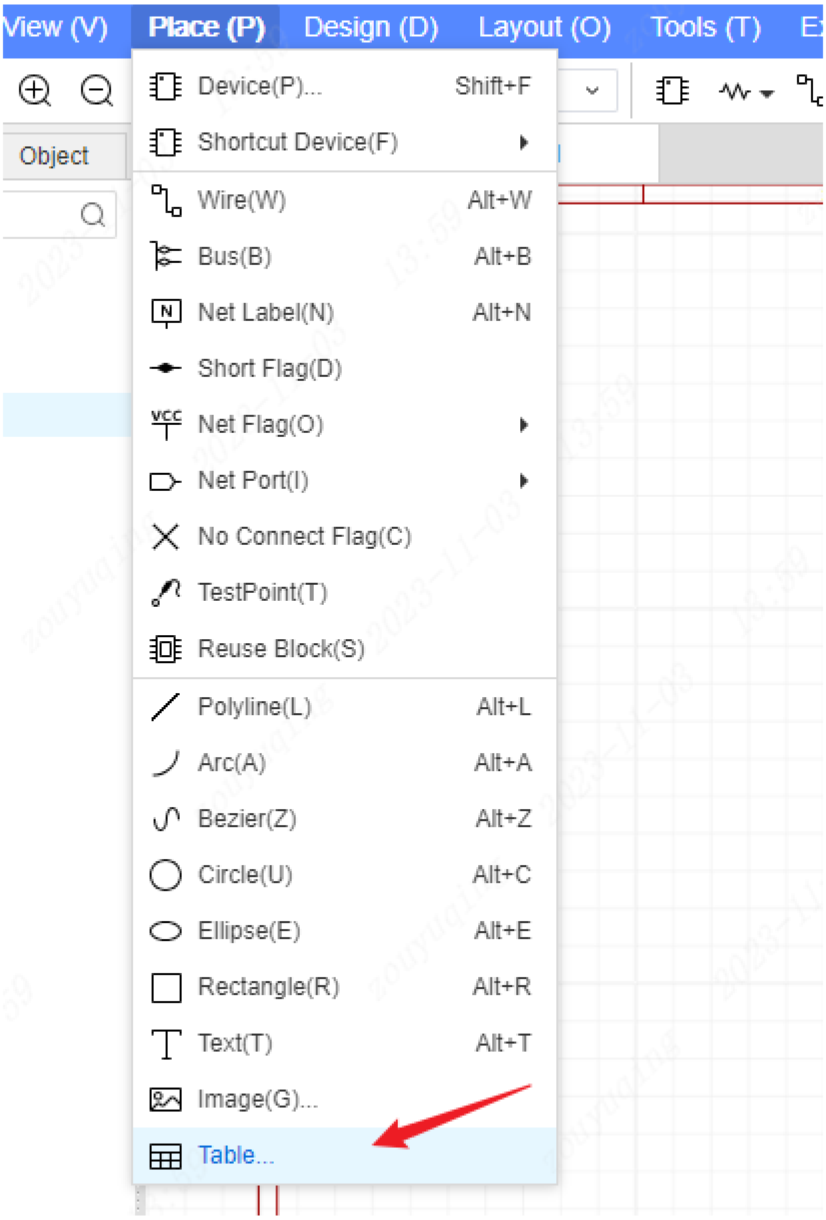
Table

Table

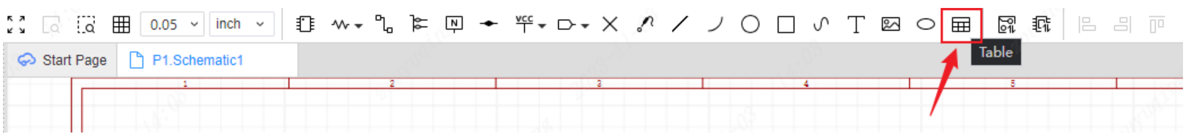
In EasyEDA Pro, the schematic diagram can be placed in a table, which is convenient for everyone to explain and view the schematic diagram.

Operation entrance:

- Top Menu - Place - Table...



- Or directly click the table icon in the top toolbar

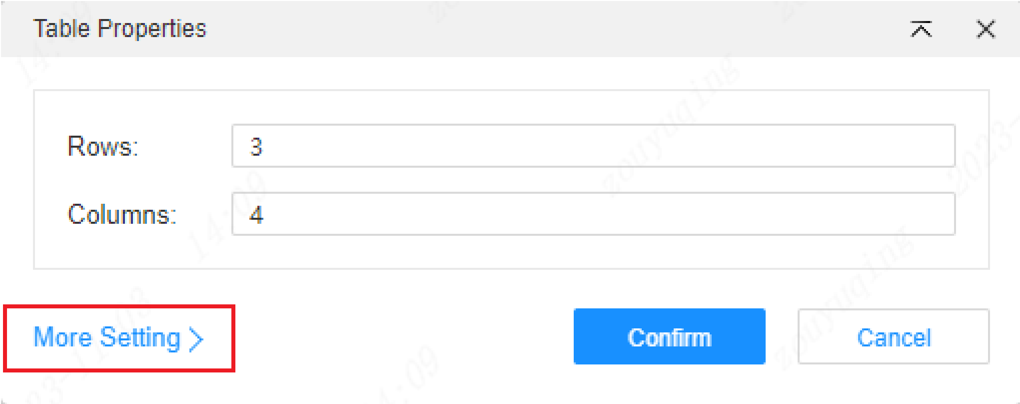


Create Table

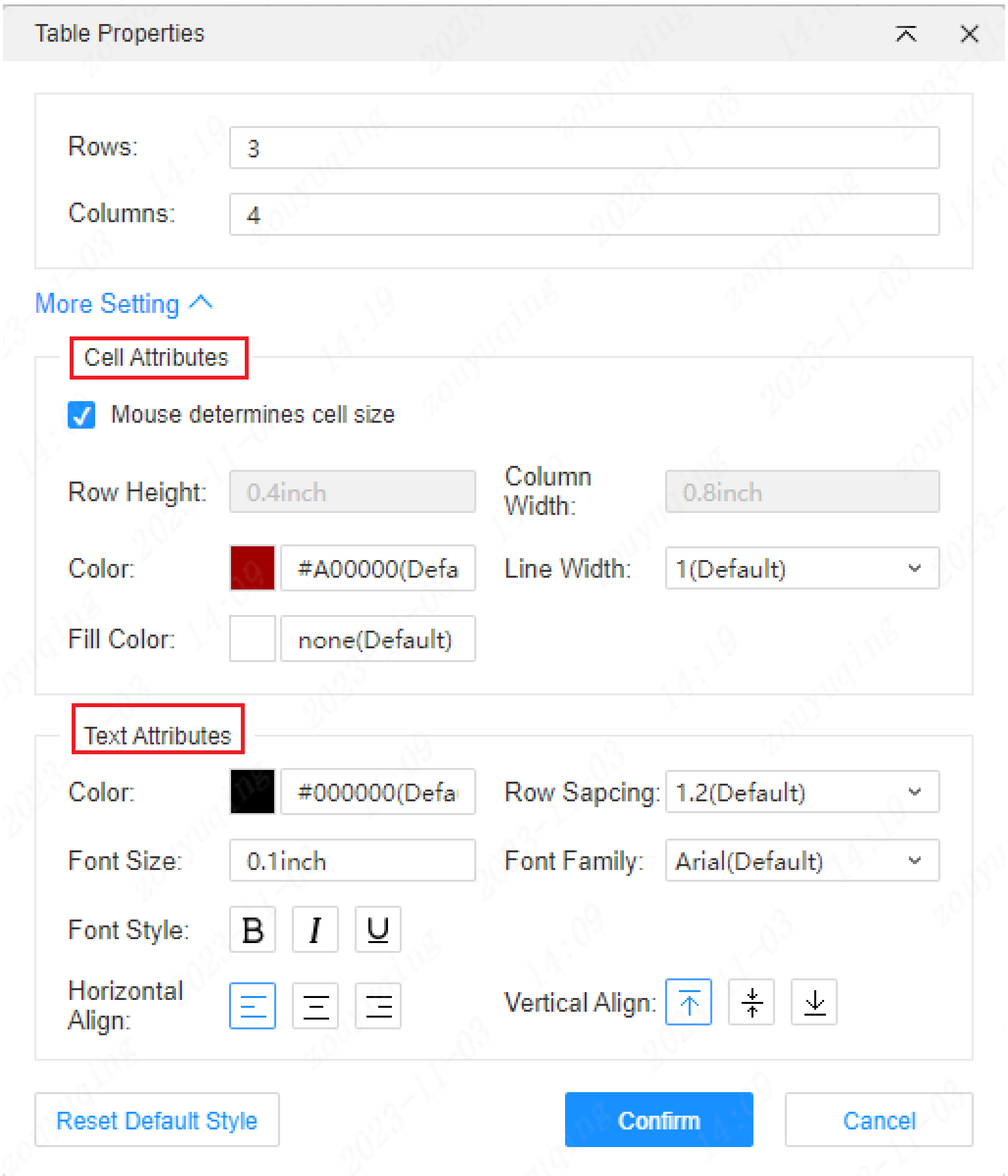
Table Properties Pop-up

Before drawing the table, the table property setting pop-up window will pop up first, that is, you need to customize some table properties before starting drawing.

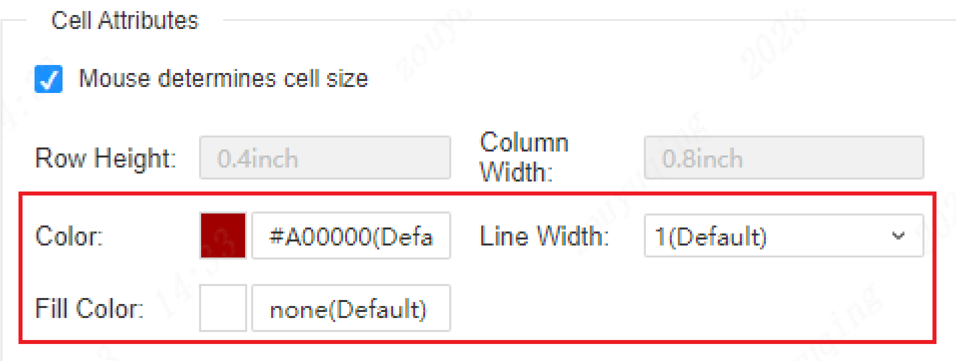
- The default number of rows and columns can be seen in the pop-up window. Click the "Confirm" button directly to determine the properties of the table to be created and start drawing.



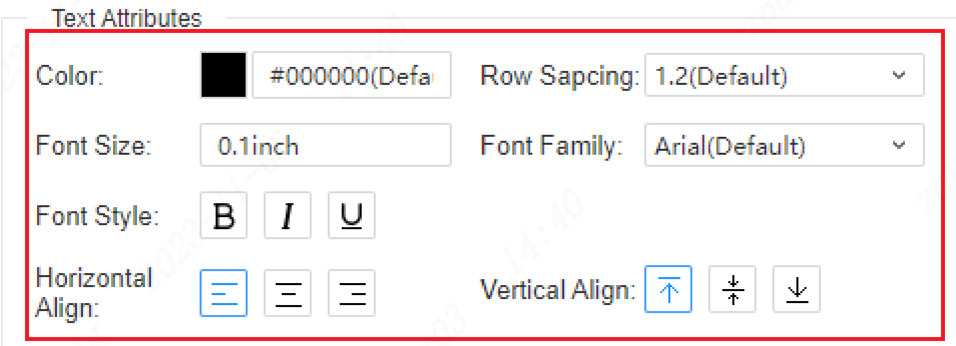
- Collapse more table property settings by default, and click the **More Settings** button in the lower left corner of the pop-up window to expand and view more.
- In the collapsed More Table Property Settings, you can customize the cell properties and text properties of the table.



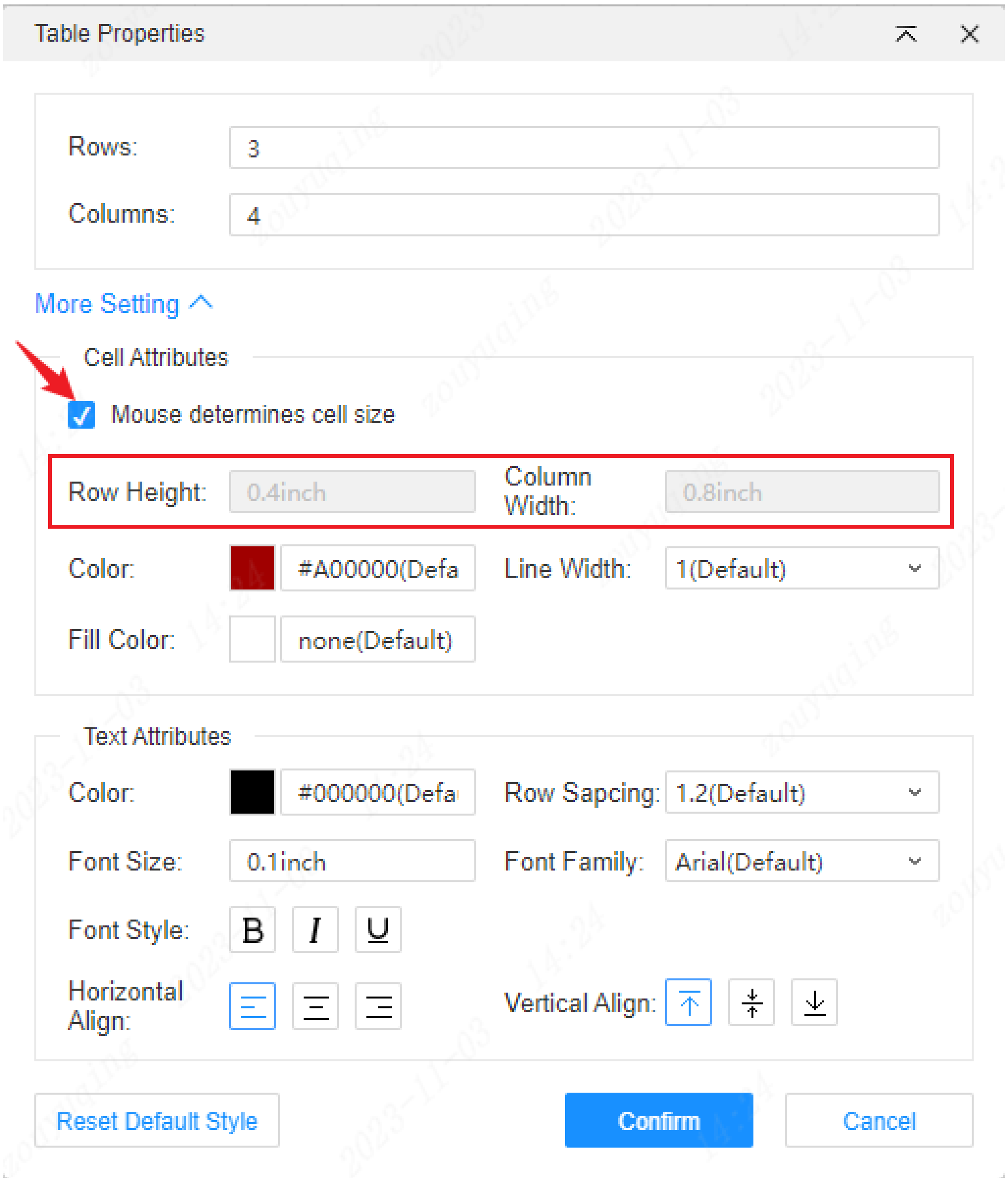
- The cell border color, fill color, and border lineweight all support customization.



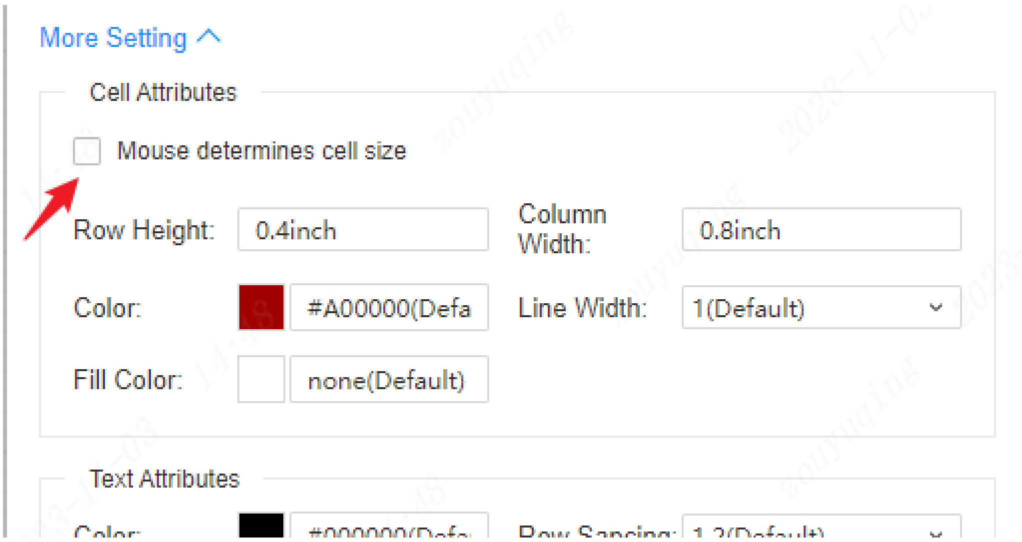
- The color, font size, font, and font style (bold/italic/underline) of the table text all support customization, and line spacing and alignment can also be set.



- Default **check** "Mouse determines cell size", cell line-height, column width is the default value of grey out.
  That is, after clicking the Confirm button, you can move the mouse on the canvas of the page to determine the width and height of the cells in the table.



- After manually **unchecking** "Mouse determines cell size", the line-height and column width text box of the cell will be grey out.
  After customizing the properties, click the Confirm button. A table will be automatically created based on the property setting information in the pop-up window, and the table can be placed directly on the canvas with the left mouse button.



- Expand More Settings pop-up window in the lower left corner also provides a restore default style button, you can quickly reset the table properties to the default configuration.

Note:
The number of rows and columns should be at least 1, and the line-height and column width should be at least 0.1inch (2.54mm). Illegal values that do not meet the rules in the numerical input box will be reset to the previous legal old value.

The values in the table property pop-up window will remain the previous settings. After expanding more settings, the next time the table property pop-up window is displayed, it will also be expanded; the same goes for folding operations.

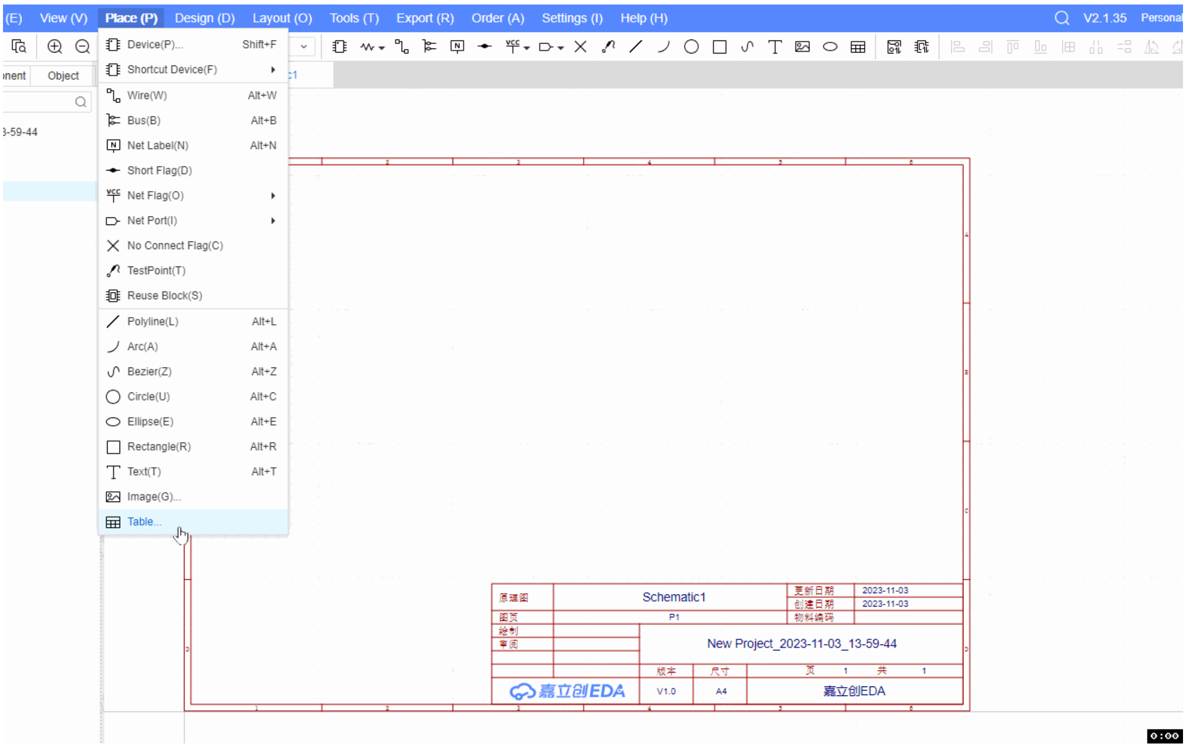
Draw Tables

- **Mouse decision (default)**

When "Mouse determines cell size" is selected, complete the various customizations in the table property setting pop-up window and click the confirm button, the table icon will be displayed in the upper right corner of the cursor.

The first left mouse button at any position on the canvas of the image page determines a vertex of the table, moves the cursor, and previews a table with a fixed number of rows and columns. The line-height column width of all cells will change in real time according to the movement of the mouse.

Adjust to the appropriate size, the second left mouse button can determine the final style of the table, complete the drawing of the table, and enter the state of the next table to be drawn (also the mouse determines the cell size).

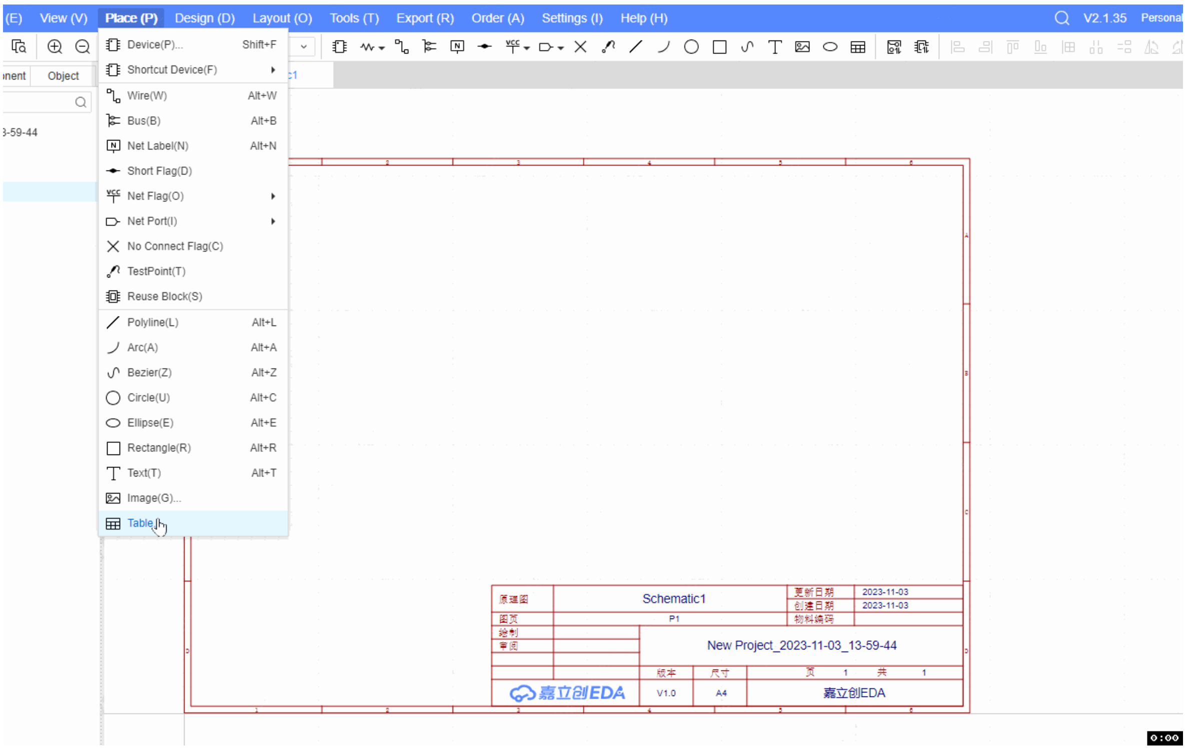


Note: Before completing the drawing, you can press Tab, and the Table Property Settings pop-up window will pop up again to continue editing the initial creation values of each table.

- **Automatically create based on settings**

When "Mouse determines cell size" is not selected, complete all customizations in the table property setting pop-up window and click the Confirm button, and a table (automatically created according to the settings in the table property pop-up window) will be displayed directly at the cursor.

Move the cursor, and the left mouse button can place the table in the appropriate position, complete the drawing of the table, and enter the state of the next table to be placed (displaying tables of the same size to follow the cursor).



Edit Tble

Box selection or clicking on the inner/outer borders of a table can select the entire table. The outer border of the table is highlighted, and a green padding circle is displayed at the 4 vertices.

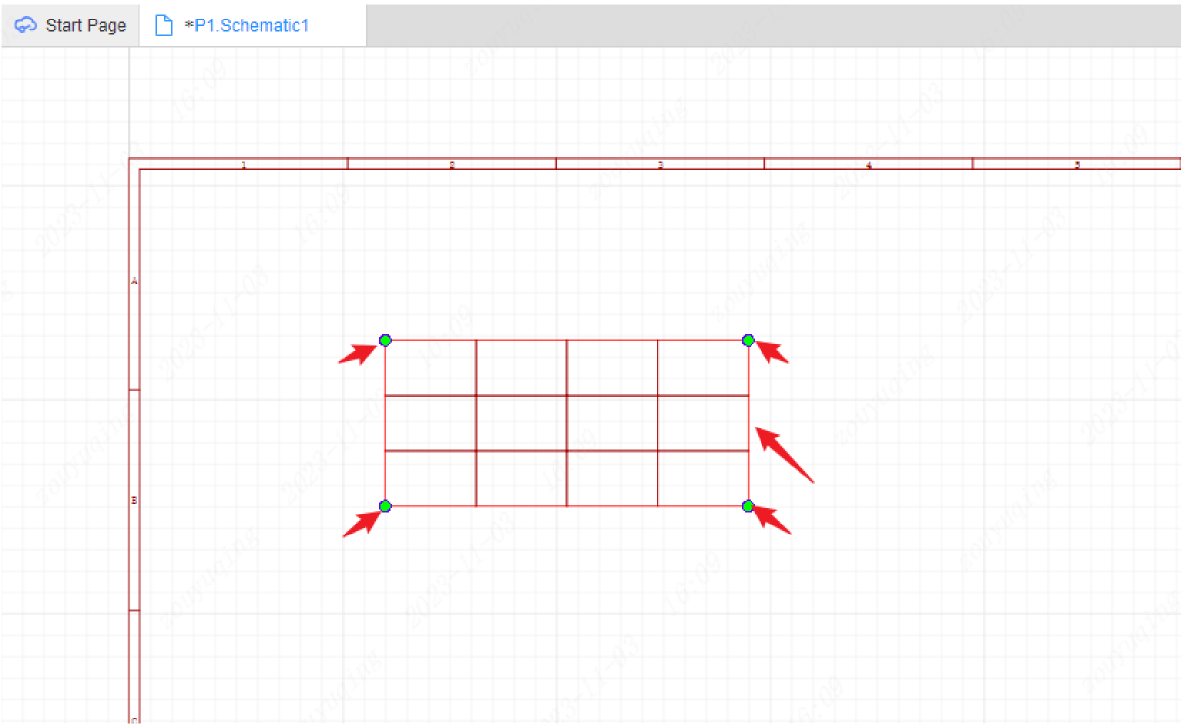
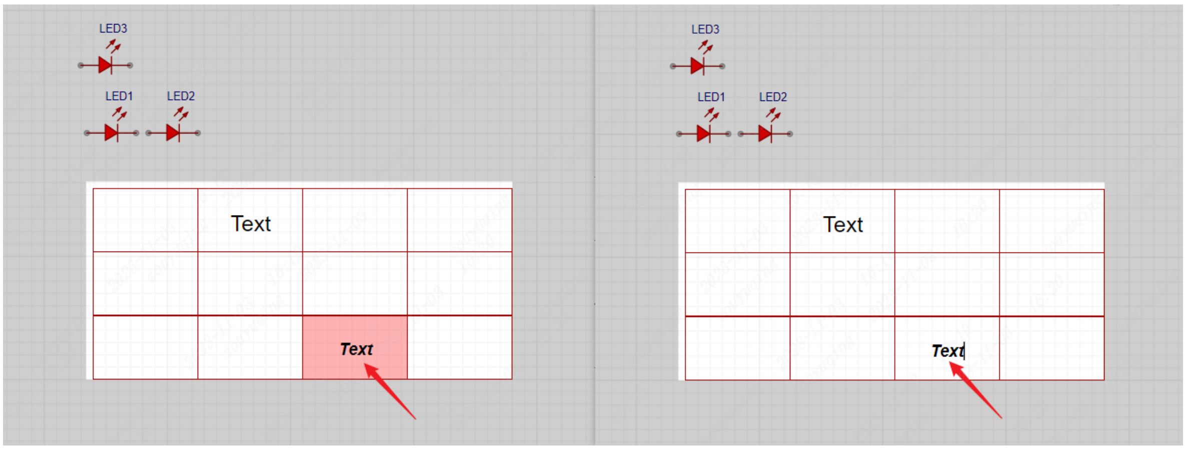


Table Mask

When you directly click to select a cell or double-click to enter the cell text editing, you will enter the mask mode of the table.

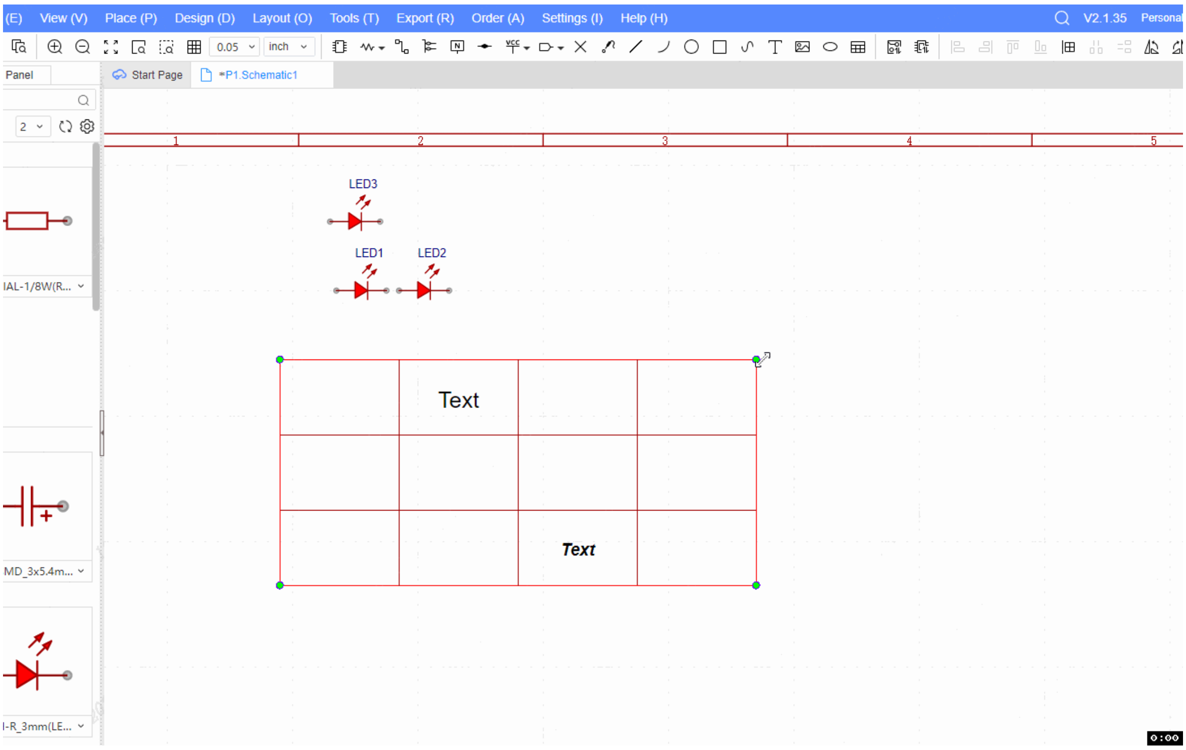


Only the normal brightness display of the table part can be edited, and all other non-table parts show the effect of being covered by transparent gray.

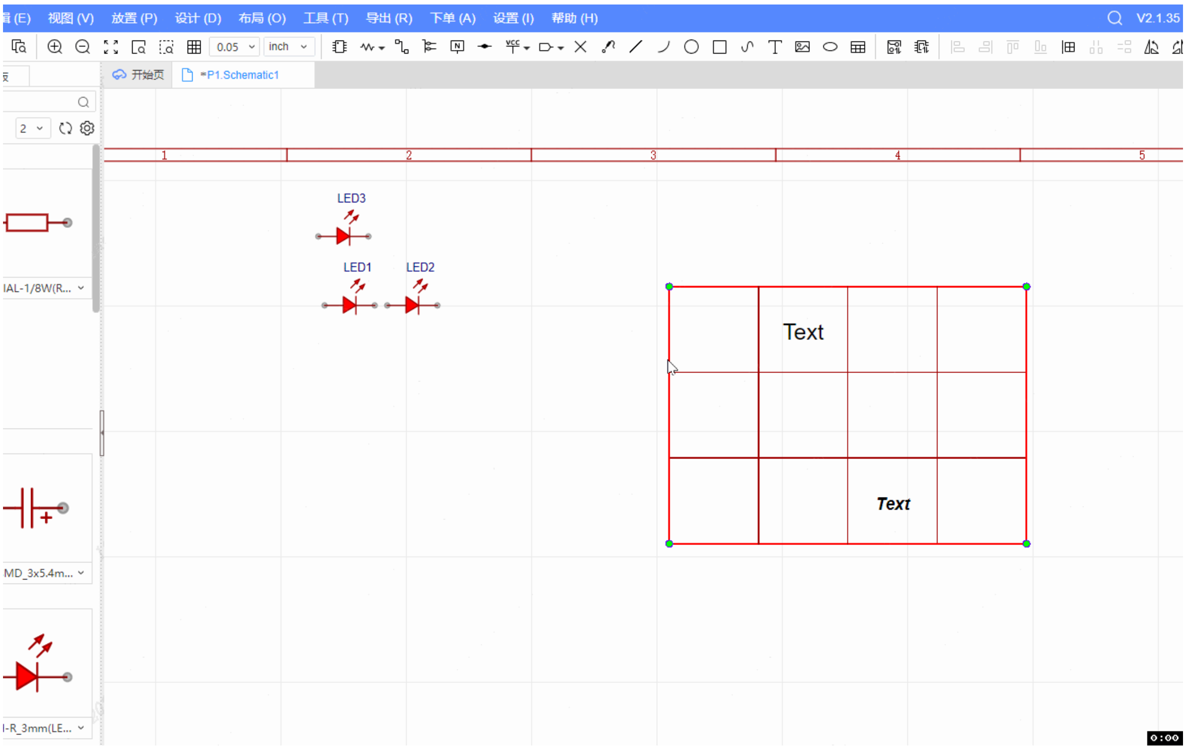
Click the non-table position to exit the mask and uncheck the table. Click the inner/outer border to exit the mask and keep the entire table selected.

- **Before Entering the Mask**

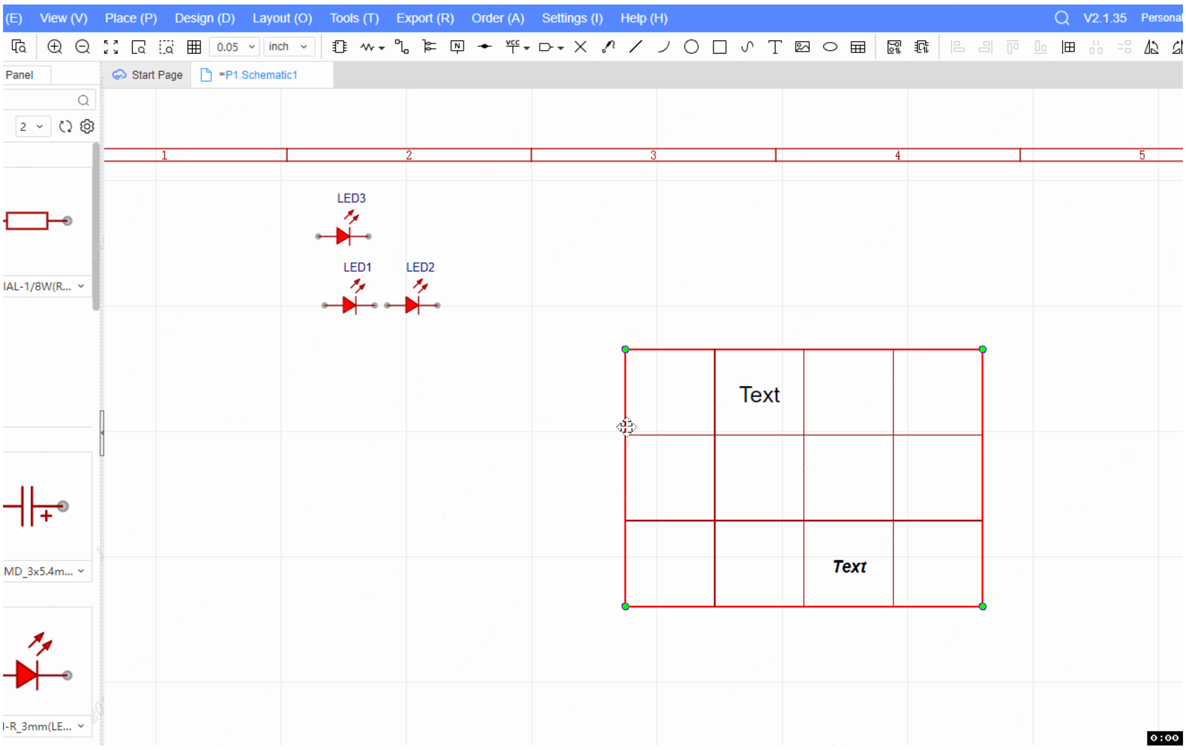
Select the entire table, move the mouse to the vertex position of Table 4, and the cursor changes to the width and height ratio stretch icon style accordingly. Holding down the left button to stretch the table will adjust the width and height values of all cells in the table in batches.



After selecting the entire table, the drag ribbon is on the outer border line of the table (excluding the 4 vertex stretch ribbon), and the cursor changes to the "drag icon" style, which can move the entire table.

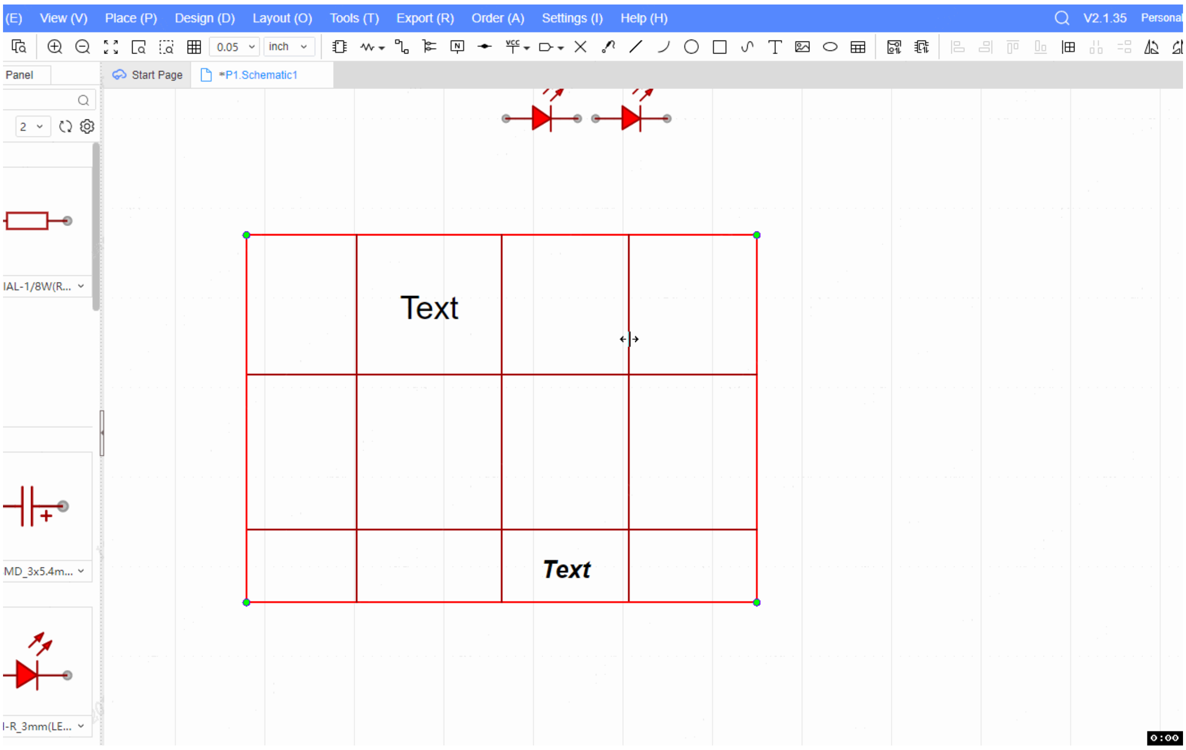


When the table is not selected, the cursor falls on the table border line/text, or you can directly drag the entire table, and the entire table will be selected after moving.

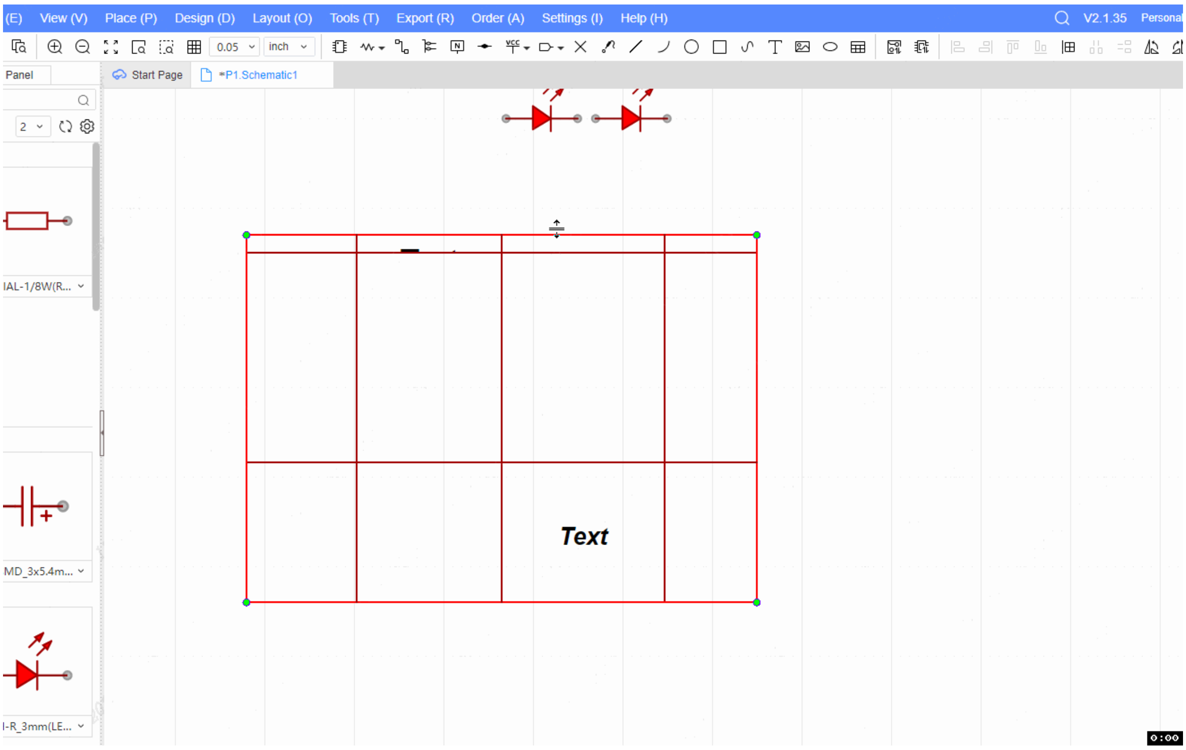


Move the mouse to the inner border line of the table, and the cursor changes to the style of "up and down/left and right stretch icons". At this time, hold down the left button to drag the corresponding edge to move and change the line-height/column width value of the cell.

Dragging the inner border line will never affect the width and height dimensions of the table (non-pushing method), only the size of the associated row and column cells.



Note: There is a range limit for moving the inner border line left and right/up and down. Always keep it from overlapping with adjacent border lines (ensuring the minimum cell size is 0.1inch). The same goes for moving the inner border line up and down.

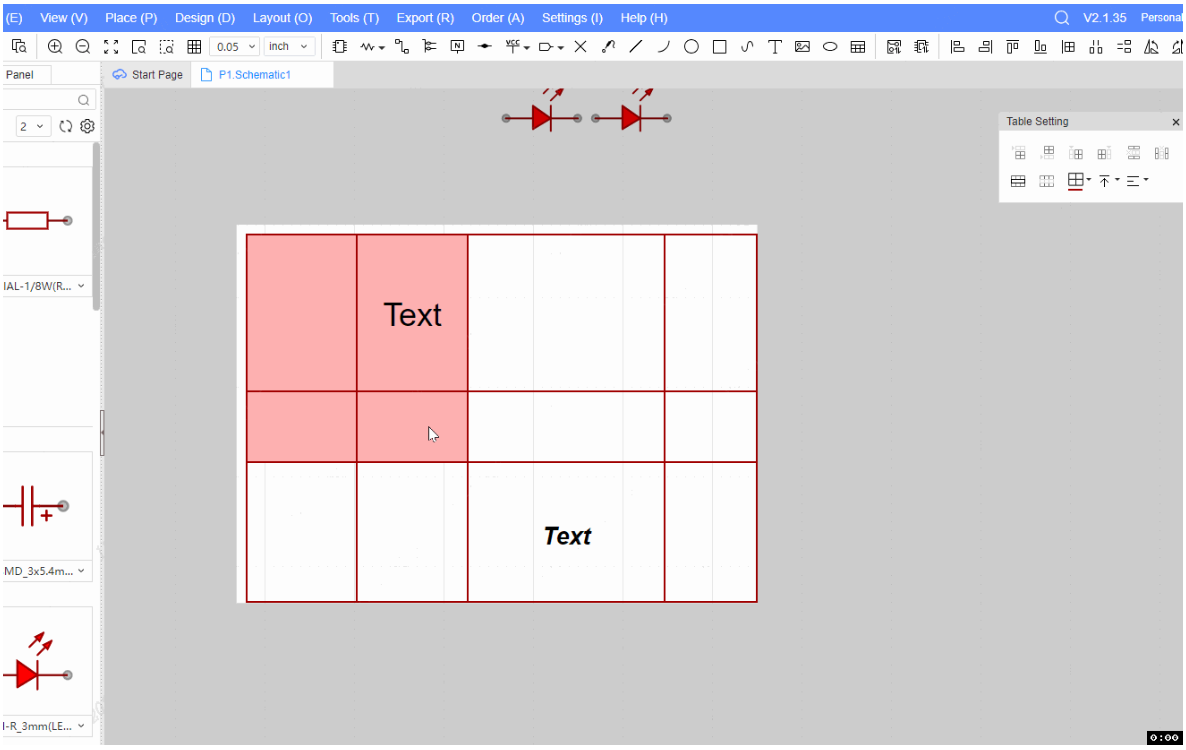


- **After Entering the Mask**

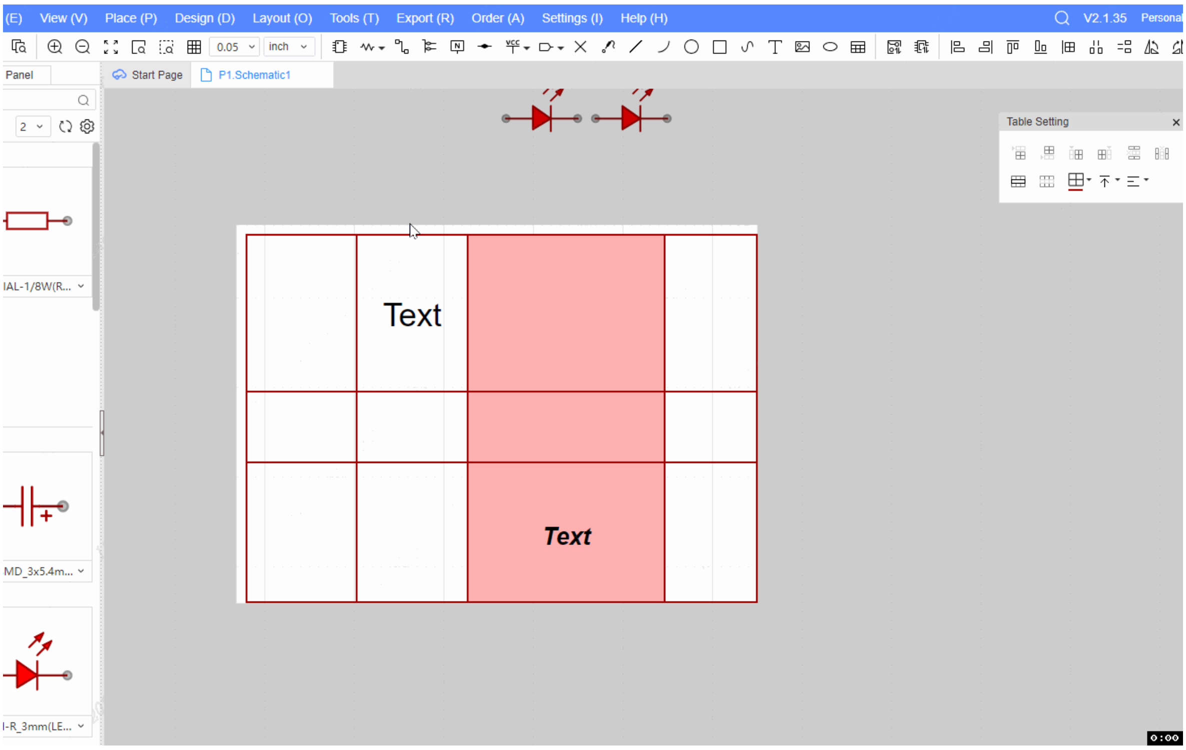
Selection of Cell

As long as you select a cell, you will definitely enter the mask mode of the table.

Support holding down the left mouse button to box select multiple cells, also support Ctrl, Shift multi-selection.

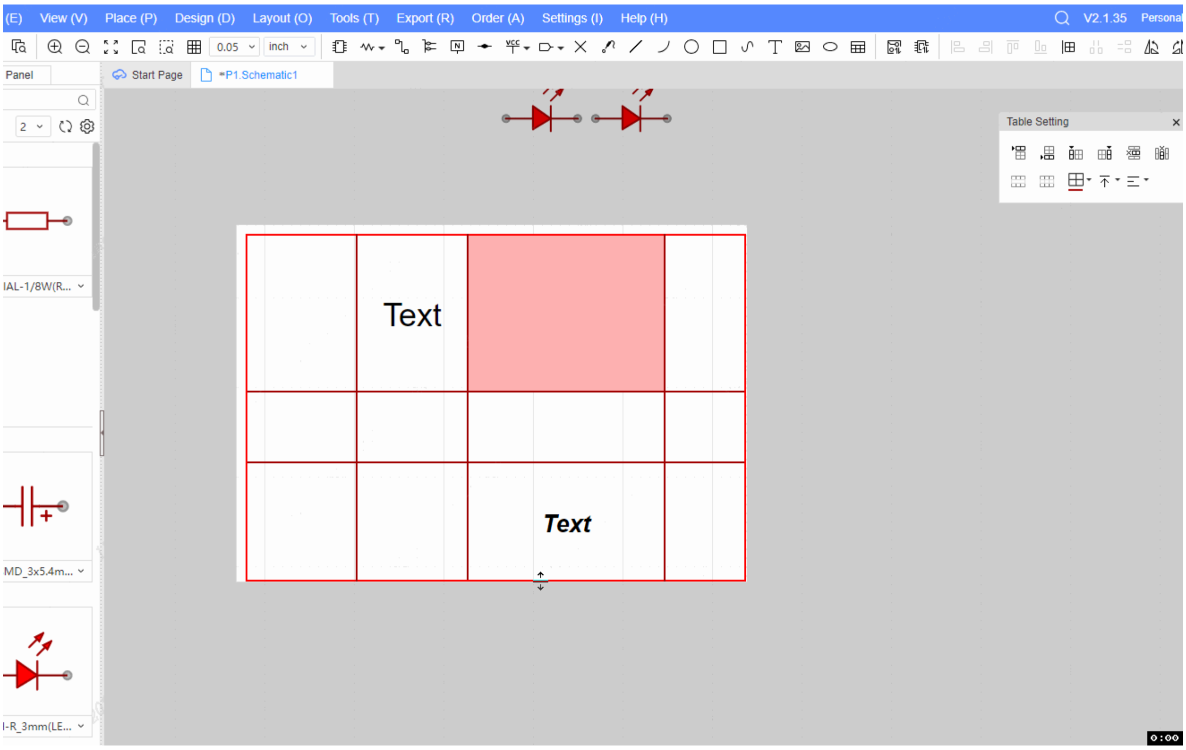


Move the mouse to the 5px wide area outside the left/upper side of the table, and the cursor changes to the function icon style of the selected row and column, which is used to indicate the entire row/column cell pointed to in the optional.



Edge Stretching

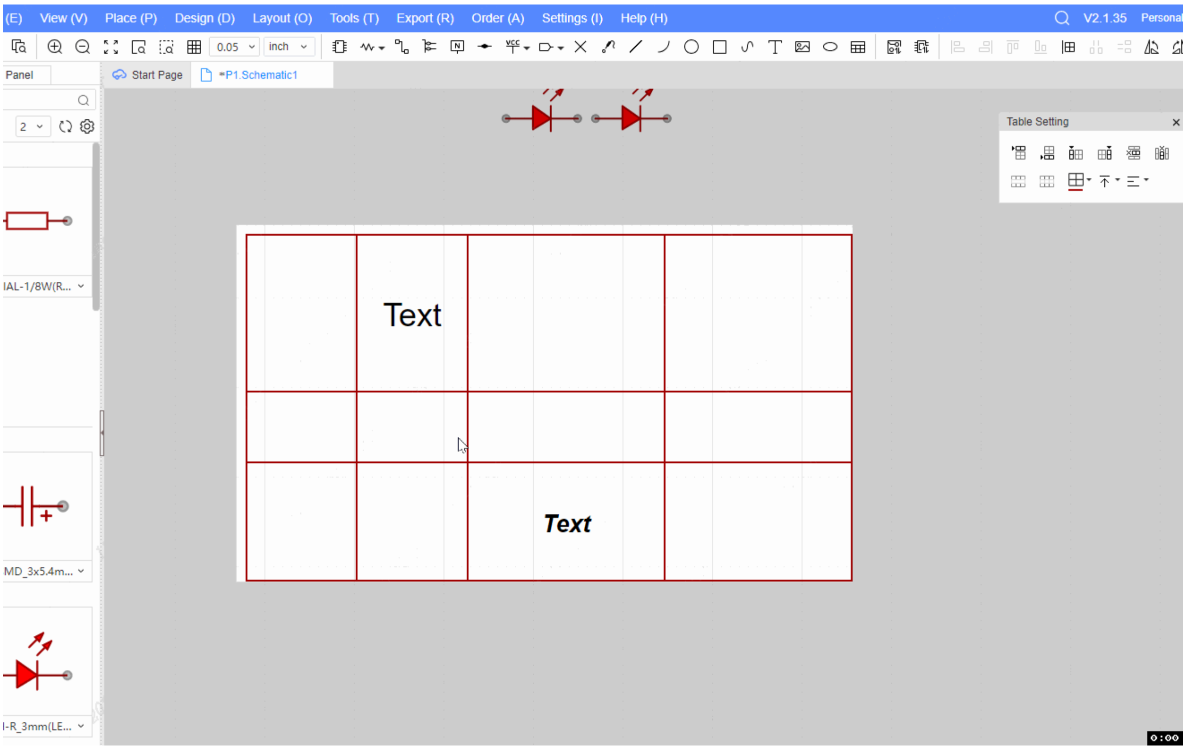
The outer border line of the table is no longer a drag ribbon, but a stretched edge ribbon. Move the cursor to the line of the inner and outer borders of the table to change the up and down/left and right stretch icon style. At this time, hold down the left button to drag the corresponding edge to move and change the line-height/column width value of the cell (after adjustment, it must be the state of selecting the entire table, not masking).



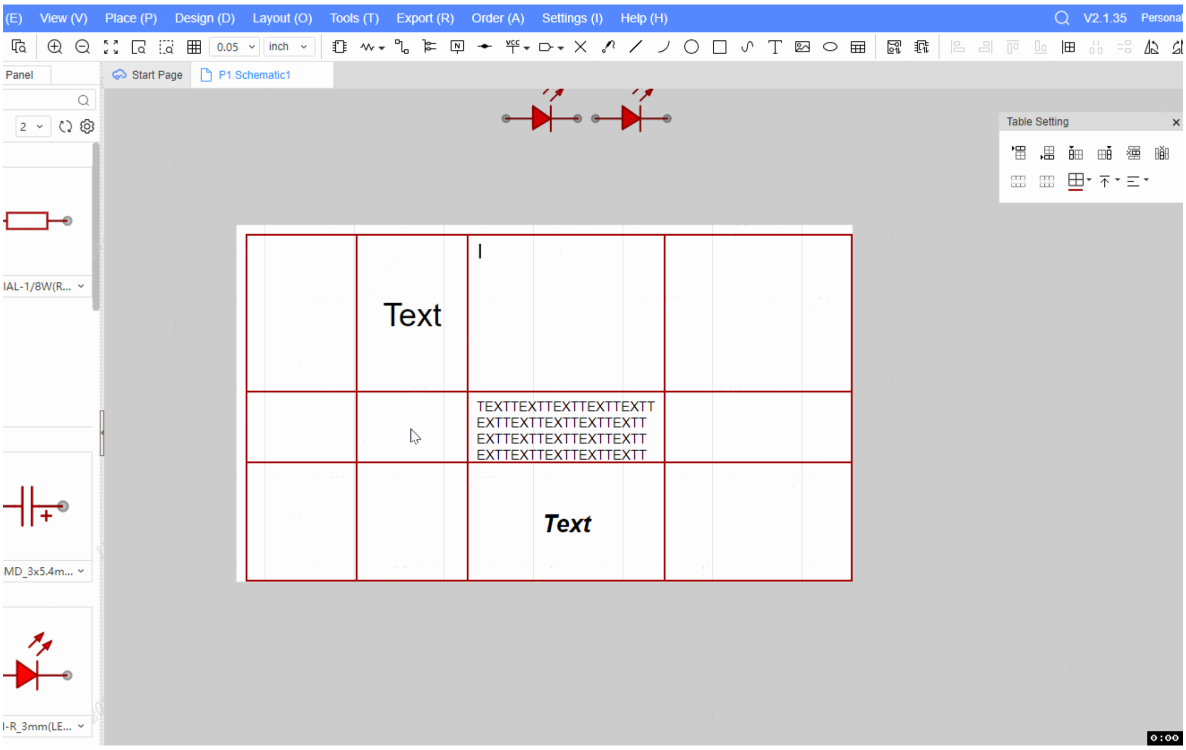
The interaction rules for stretching the inner border line of the table under the mask are the same as those before entering the mask.

Text Editing

Double-click the cell to enter the text editing state. (When editing table text, it must be in table mask mode)



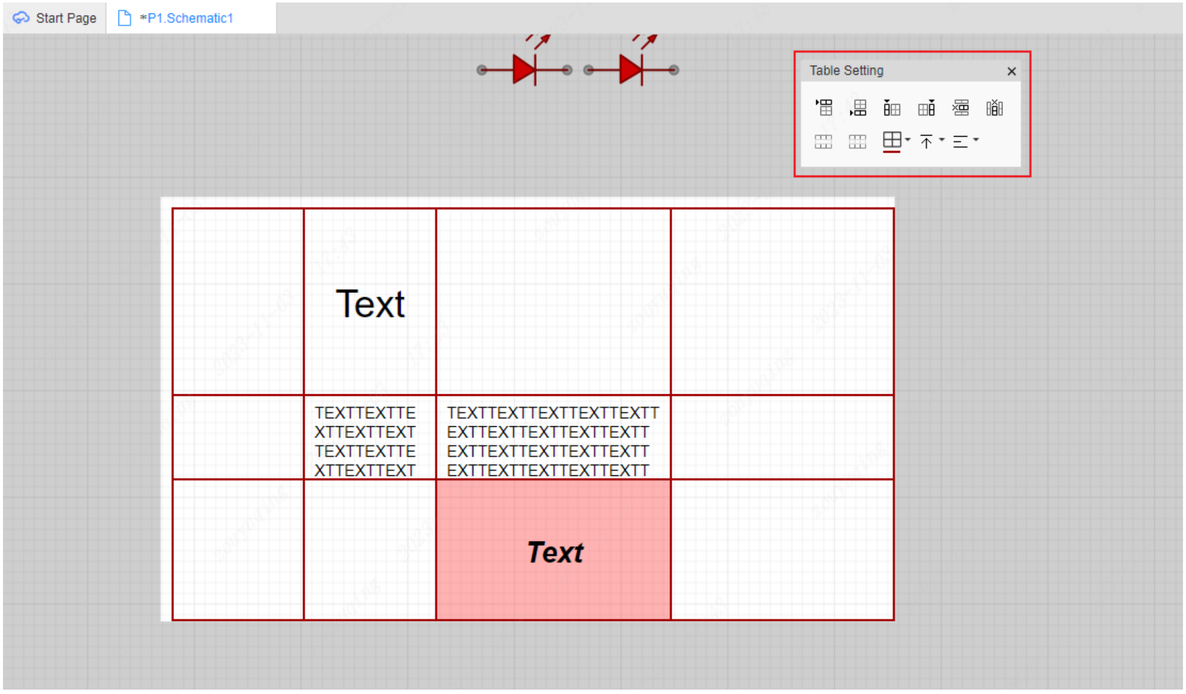
The width and height of the cell will not be self-adapted according to the text content, but will automatically wrap.
If the content is multiple and the cell is small, the display will not be complete. When the cell is expanded by manual stretching, all the text content can be seen.



Note: The size of the table will not change according to the amount of text content, unless manually stretched or modified on the right.

Form Setting Floating Window

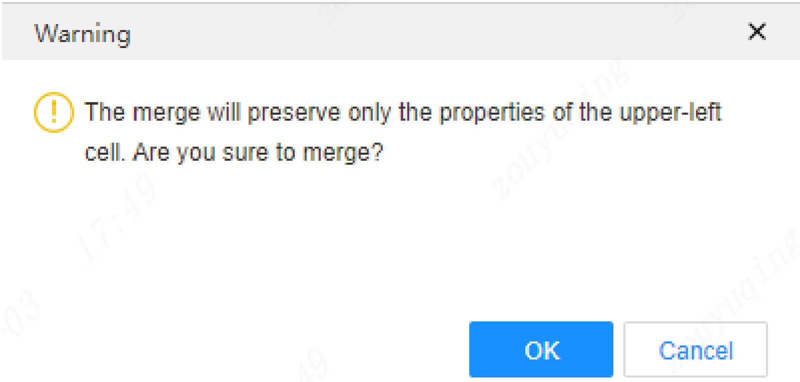
After entering the "Mask" of the table (in the state of cell selected/text editing), an independent table setting floating window is provided, which can be freely dragged to adjust the style.



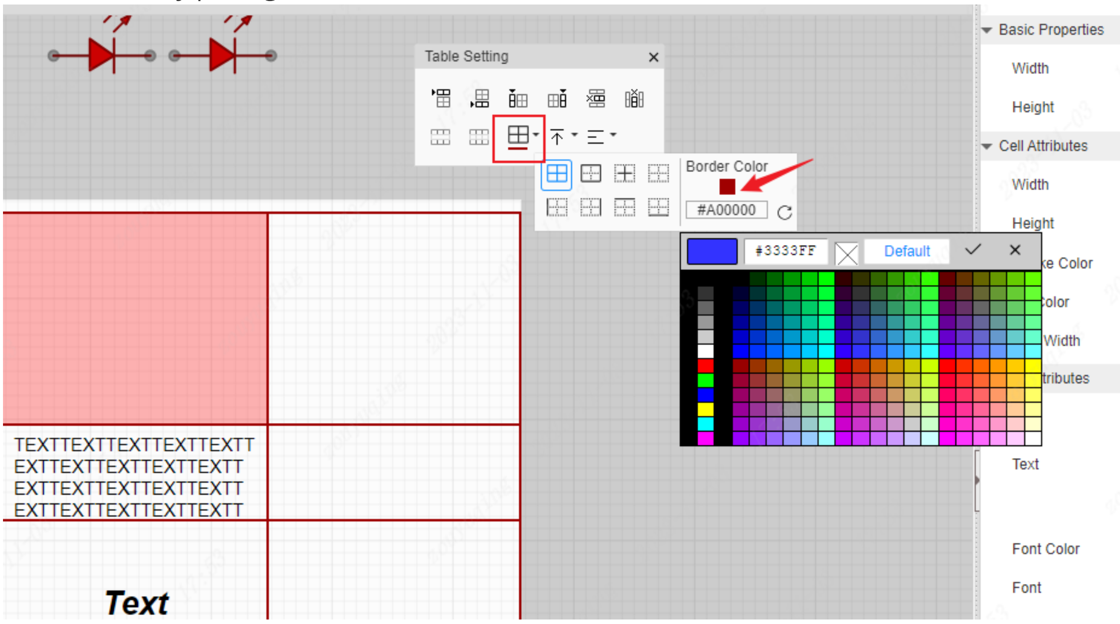
This floating window allows you to add rows and columns, delete rows and columns, merge cells, split cells, set cell border styles, and set text alignment styles for table operations.

- **Merge Cell**: - It will merge multiple selected cells into one, hide the inner border line, and take the value of "top left corner" for the text content.
  When the properties of cells are not unified, a pop-up prompt will appear: Merge will only retain the properties of the upper-left cell. Confirm to merge? (left takes precedence over

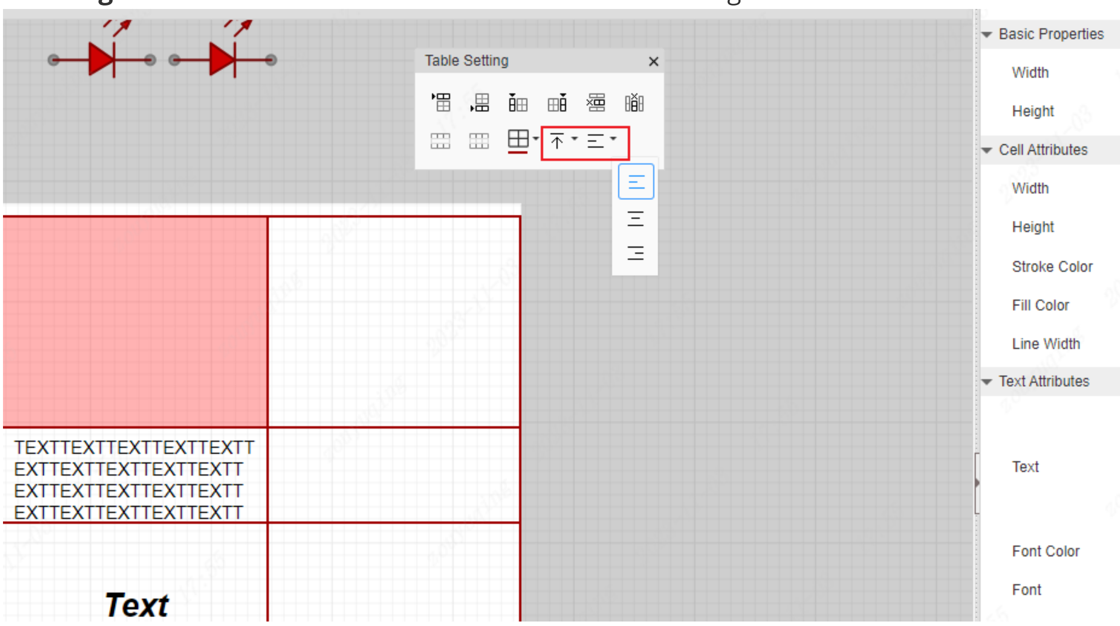
top)



- **Split Cell**: Only one cell selected is the "merge cells" operation, the icon of "split cells" is displayed normally, otherwise grey out.
- **Table Border Setting**: - directly click the border icon in the floating window, and it will be set according to the border type and border color selected last time (initial default: full border, #A00000 theme color)
  The icon contains 1 color block, which is used to display the current default border color and keep following the previous setting. You can select a specific border type and set a specific border color by pulling the box.



- **Text Alignment**: You can set horizontal and vertical text alignment.

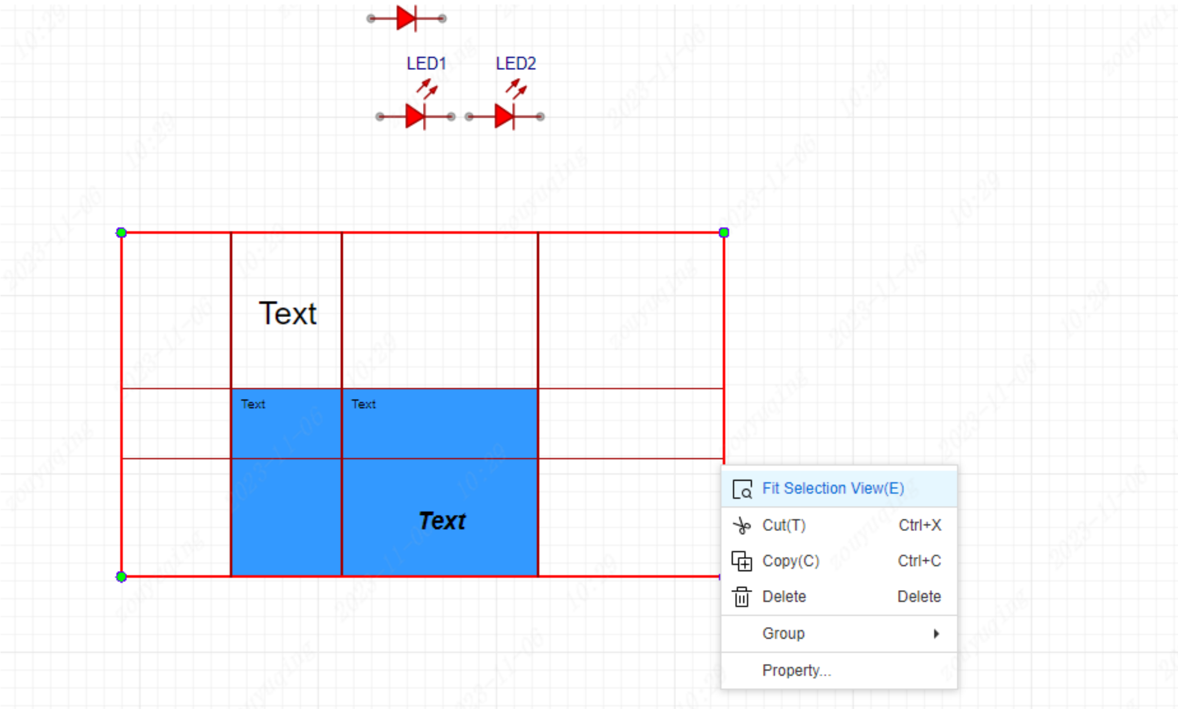


Note: The selected objects are different, and the operable items in the floating window are inconsistent. Support will display normally, and grey out is not supported.

Right-Click Menu

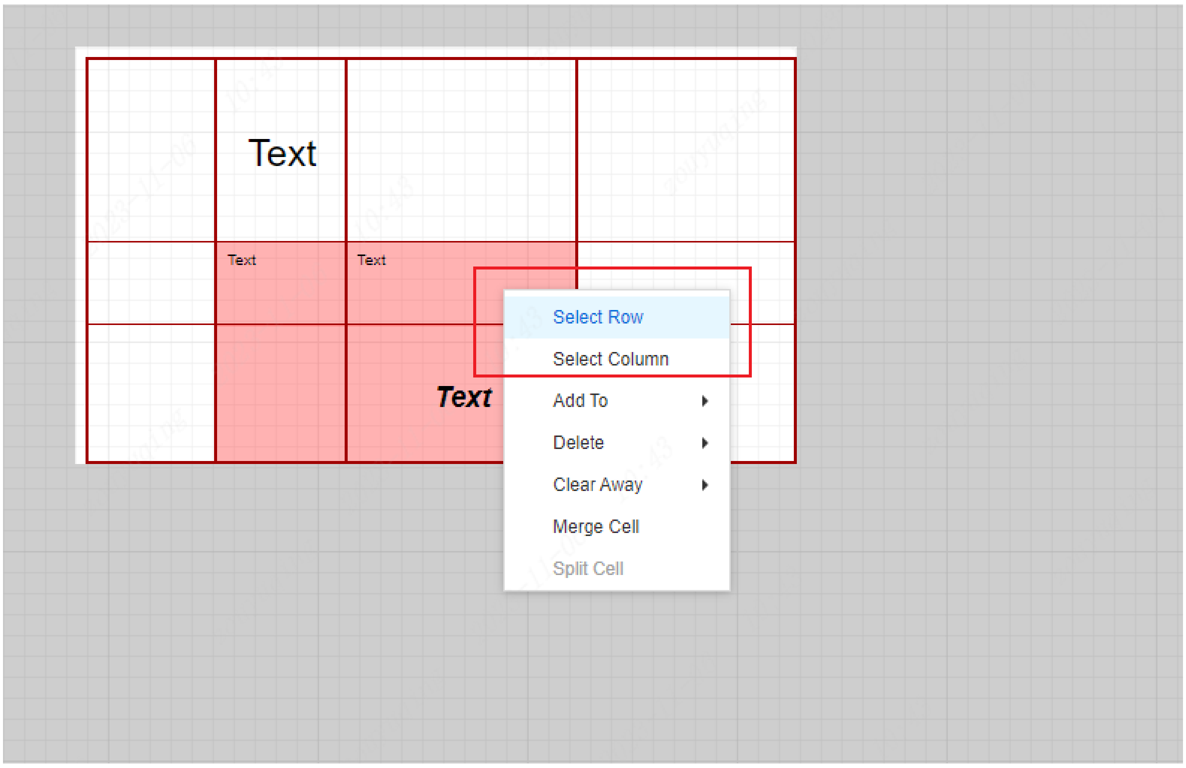
Table Right-Click Menu

After selecting the entire table, right-click the menu operation to adapt to the selection, it will zoom and move the canvas to place the table in the center of the canvas for display.
You can also cut, copy, paste, delete and other operations on the table.
Tables can also be added to the combination.

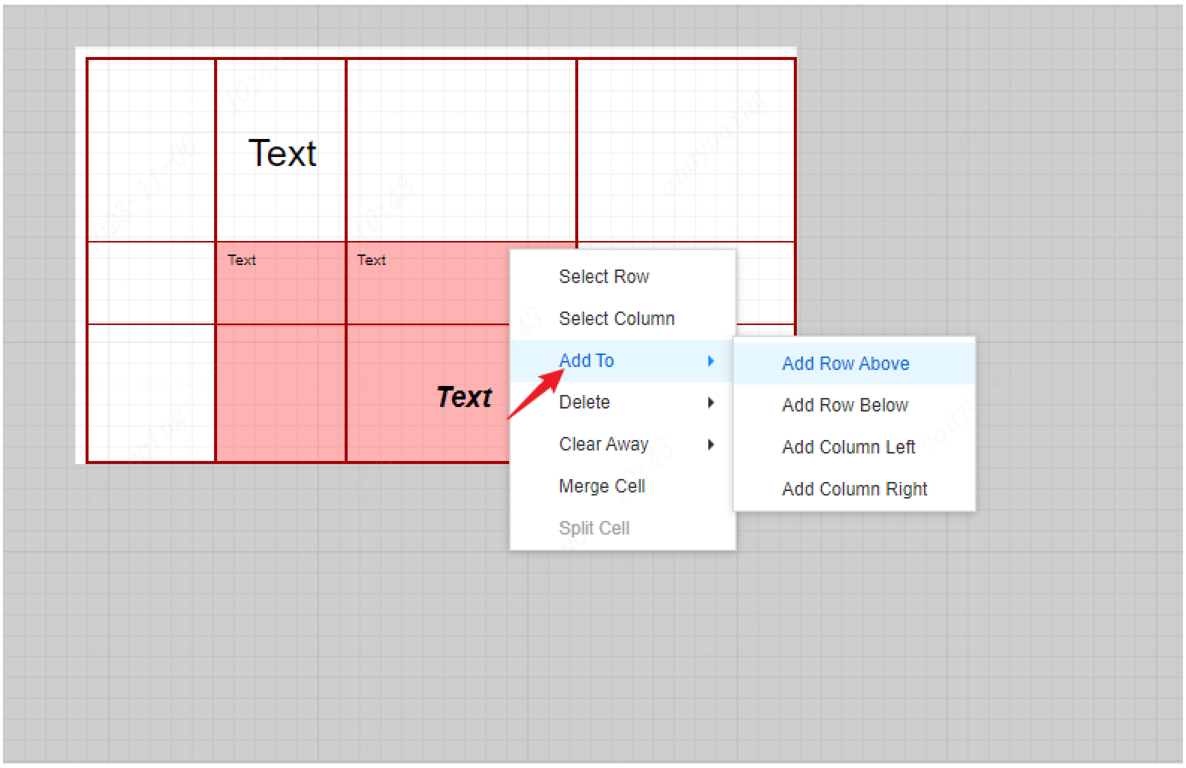


Cell Right-Click Menu

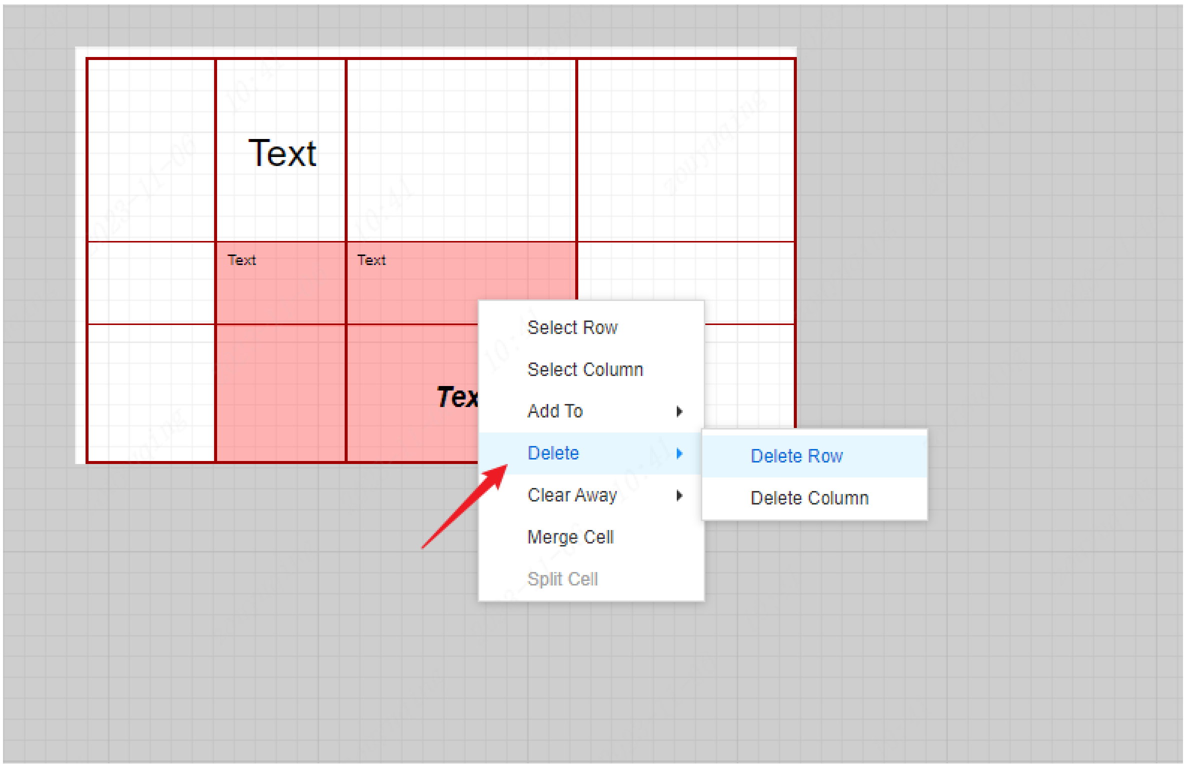
Enter the mask, select the cell, and right-click the menu to operate to select the row or column where the cell is located.



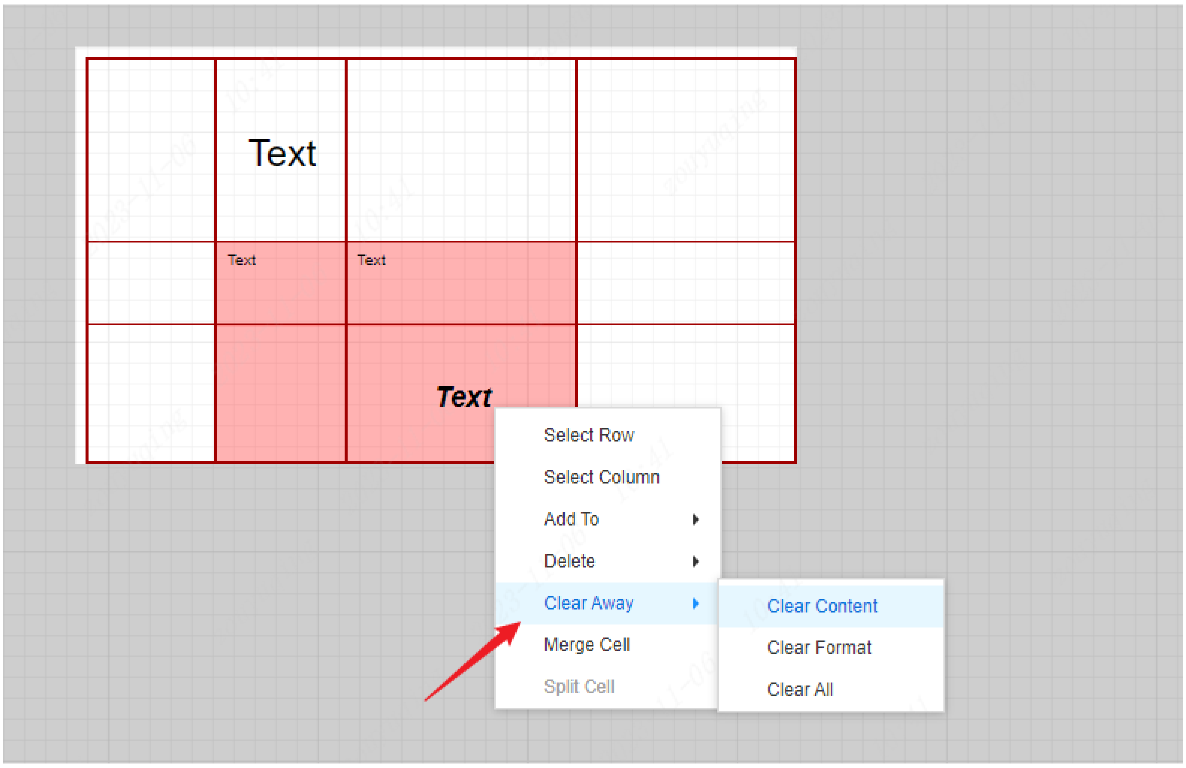
Support right-click operation to add rows up/down or left/right columns in the selected cell.



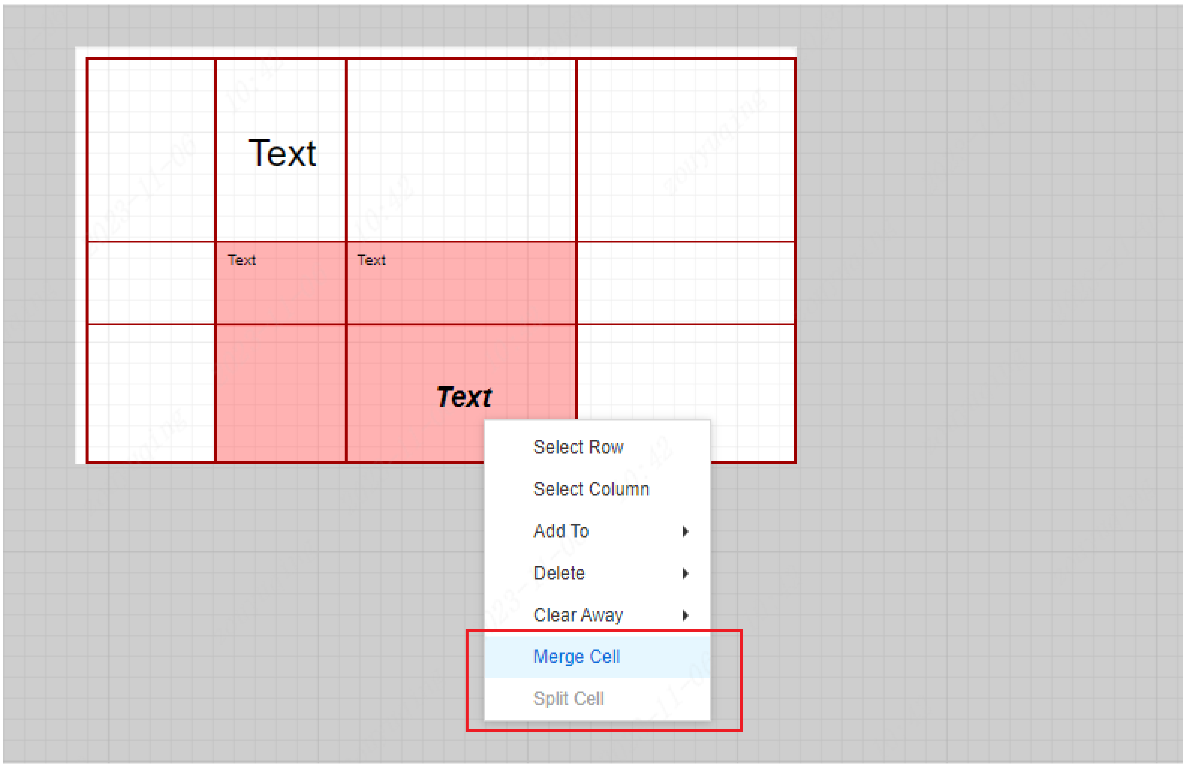
Support operation to delete all cells in the row/column where the cell is located.



You can choose to clear only the text content in the selected cells, clear only the formatting, or directly clear all content and formatting.

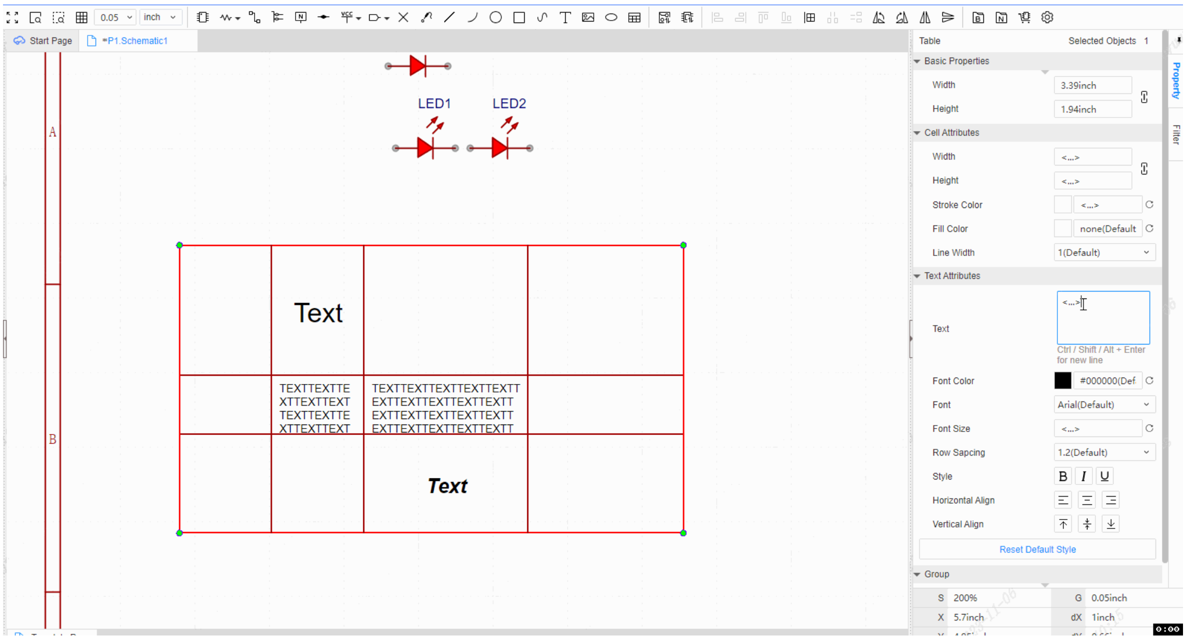


Support operations to merge cells and split cells. "Merge cells" only supports operations when the selected area is a rectangle and not merged cells; "Split cells" only supports operations when selected cells containing merged cells.

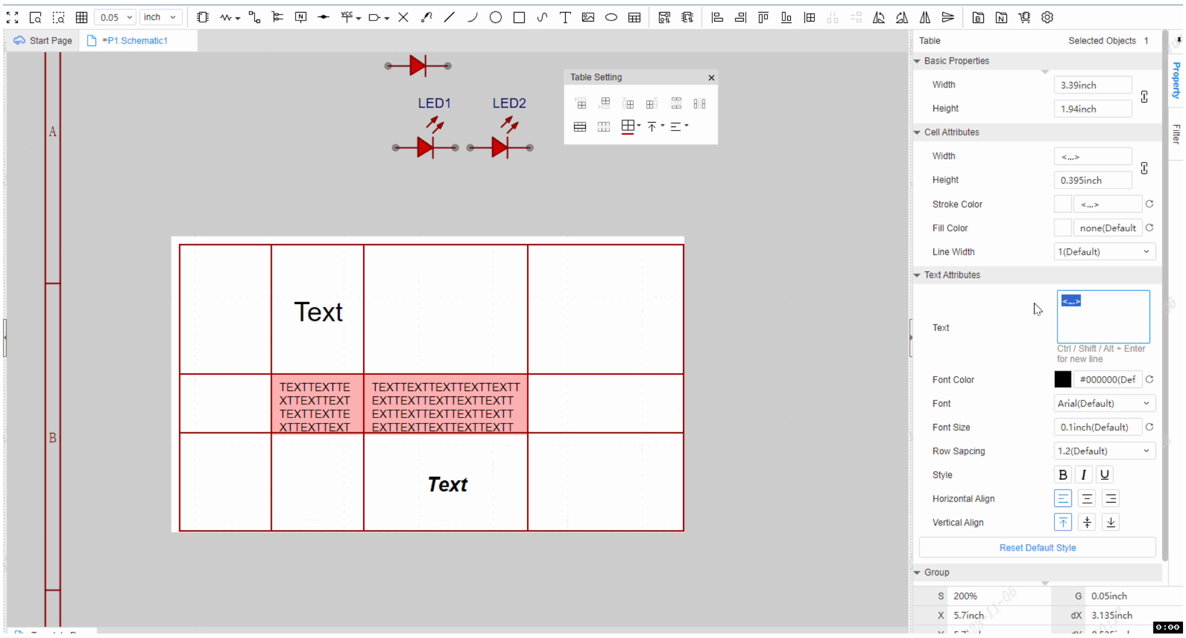


Right-Hand Property

When selecting an entire table, the adjustment of the properties on the right is for the entire table and can be adjusted to the corresponding properties of all cells and text in batches.



When selecting a specific cell or text, the adjustment of the property on the right is only applicable to the selected cell or text.



Update PCB/Convert Schematic to PCB

When you have finished drawing the schematic and want to convert it to PCB, you can click **Top Menu > Design > Schematic to PCB**

Notice:

- **To update from schematic to PCB, both schematic and PCB must be under the same board.**

When the schematic diagram is transferred to the PCB, it will be checked first according to the design rules of the schematic diagram, and then converted after meeting the conditions for transferring to the PCB.

Check footprint

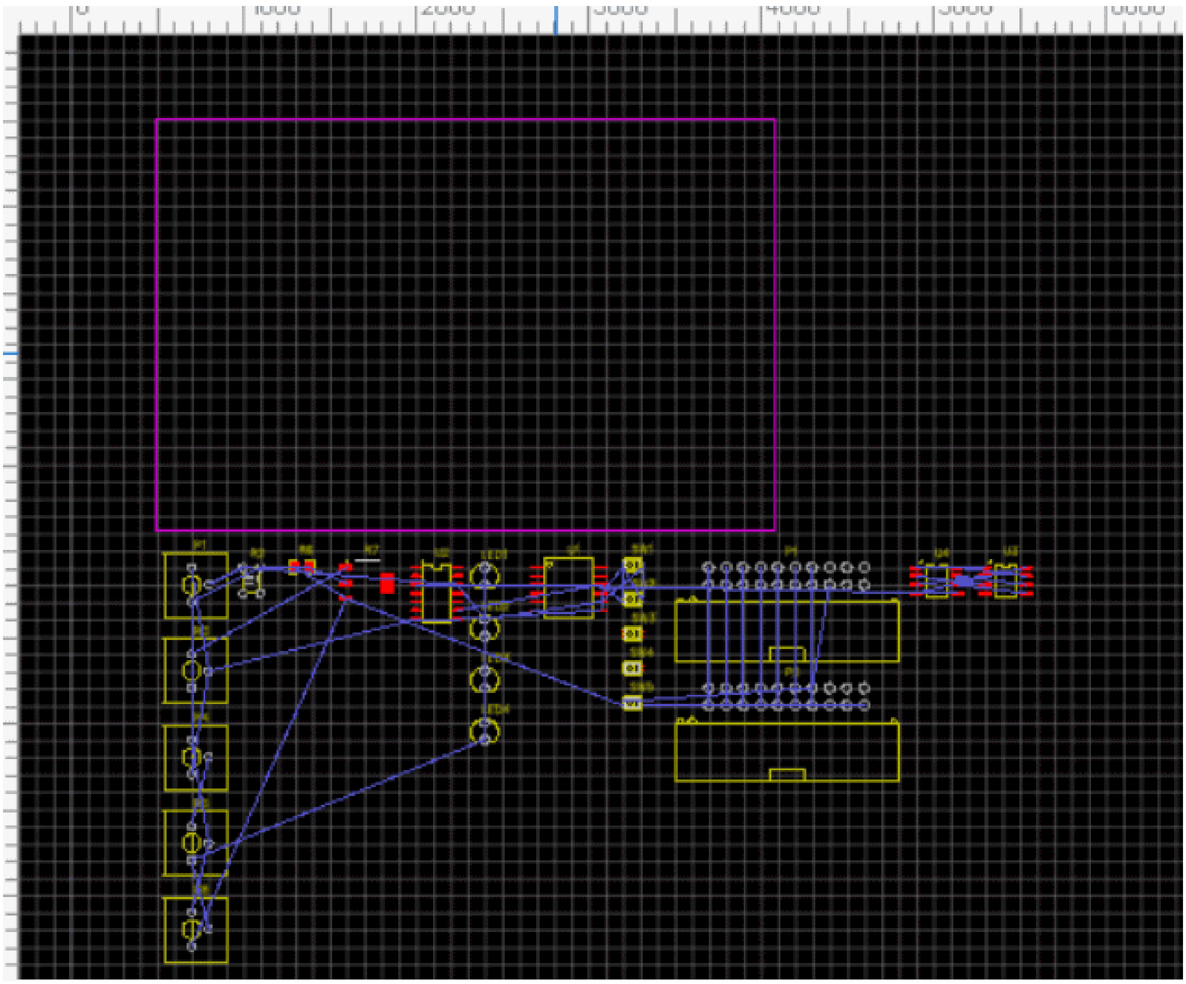
After clicking "**Schematic to PCB**", if the editor detects that the corresponding information of components and footprints is abnormal, an error dialog box will pop up, and the conversion will not continue. You need to fix errors in the footprint manager.
Because EasyEDA Pro will record the unique ID of the footprint in the symbol library and schematic diagram when specifying the footprint, so the error message corresponds to the following:

- There is a specified footprint name, but the specified footprint library title has been modified by the creator
- There is a footprint name, but no footprint has been specified through the footprint manager
- The symbol library pin number name is inconsistent with the specified footprint pad number name
- The number of symbol library pin numbers is greater than the specified number of footprint pads
- footprint is empty unspecified footprint

元件	动作	对象	导入前	导入后
C3	修改属性	DIP-CAP	容量:100nF	容量:473
C2	修改属性	DIP-CAP	容量:100nF	容量:103
J1	修改属性	HDR-TH_2P-P2-54-V-M	位号:DC电源输入	位号:J1
R3(2)				
	修改属性	可调电阻	位号:波形频率	位号:R3
	修改属性	可调电阻	阻值:50	阻值:50K
R4	修改属性	DIP-RES	阻值:10K	阻值:1K
R10(2)				
	修改属性	可调电阻	位号:三角波正弦波幅度	位号:R10
	修改属性	可调电阻	阻值:50	阻值:20K
C5	修改属性	DIP-CAP	容量:100nF	容量:104
C6	修改属性	DIP-CAP	容量:100nF	容量:473
C7	修改属性	DIP-CAP	容量:100nF	容量:473
R8	修改属性	DIP-RES	阻值:10K	阻值:1M
C8	修改属性	DIP-CAP	容量:100nF	容量:473
C9	修改属性	DIP-CAP	容量:100nF	容量:103
R1	修改属性	DIP-RES	阻值:10K	阻值:100K

Generate PCB

After the conversion is successful, as shown in the figure below, the PCB footprints are automatically arranged in a row in order. The blue flying wires indicate that the two pads need to be wired and connected, which belong to the same network.



Update PCB

When you have completed the transfer PCB action, but modified the schematic again, you do not need to transfer a new PCB, just click "Update PCB" to update the changes to the existing saved PCB file.

Changes can also be imported directly in the PCB: Top Menu - Design - Import Modification Information.

- Because the schematic net name is generated after calculation, if some nets are changed after you modify the schematic, after updating to the PCB, the routed layout of the original net will remain unchanged.
- Update the pad network and update the wires related to the pads with the pad network update. There may be changes in the wire network. You need to manually change the old network name of the wire to the new network name!
- Most schematics can be updated to the PCB after updating the net. However, if the schematic diagram has added or deleted devices, the wire network of the PCB needs to be manually modified.

Design Rules

Design rule settings

Sets the level information for prompting errors for electrical rule checks.

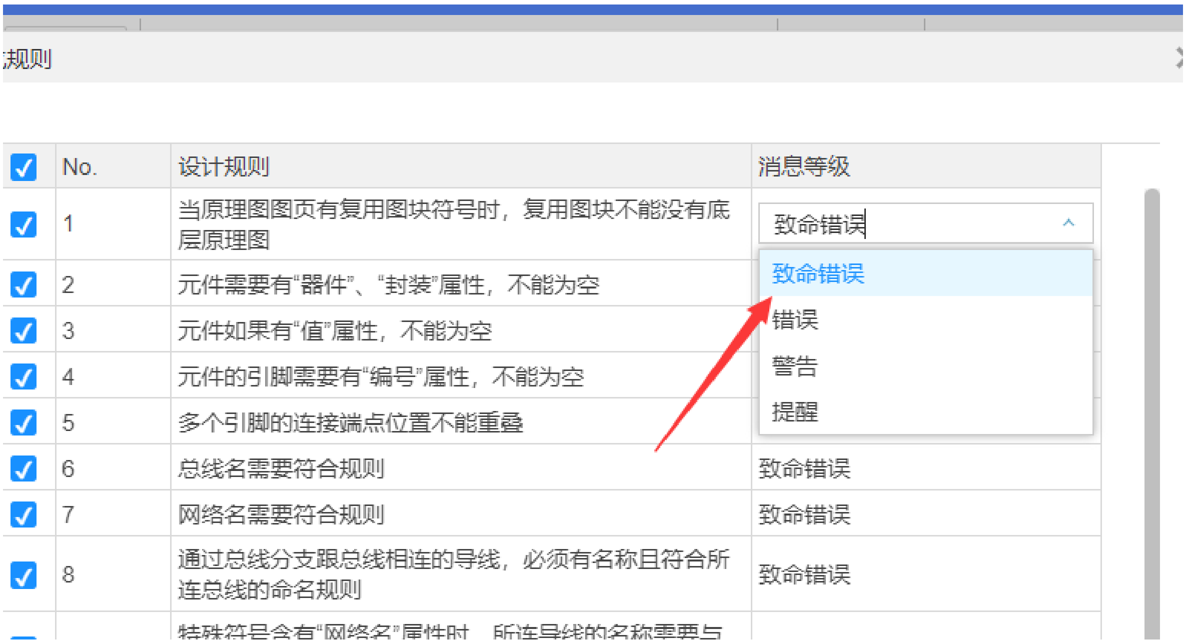
Here you can see the error message level of the rule, and you can modify the error level.

电气规则

☑	No.	设计规则	消息等级
☑	1	当原理图图页有复用图块符号时，复用图块不能没有底层原理图	致命错误
☑	2	元件需要有“器件”、“封装”属性，不能为空	致命错误
☑	3	元件如果有“值”属性，不能为空	致命错误
☑	4	元件的引脚需要有“编号”属性，不能为空	致命错误
☑	5	多个引脚的连接端点位置不能重叠	致命错误
☑	6	总线名需要符合规则	致命错误
☑	7	网络名需要符合规则	致命错误
☑	8	通过总线分支跟总线相连的导线，必须有名称且符合所连总线的命名规则	致命错误
☑	9	特殊符号含有“网络名”属性时，所连导线的名称需要与“网络名”的值一致	致命错误
☑	10	元件位号需要符合规则	致命错误
☑	11	如果元件含有多部件，每个部件的“器件，封装，位号”这几个属性必须一致	致命错误
☑	12	如果元件含有多部件，每个部件除了“器件，封装，位号”这几个属性外，其他属性必须一致	警告
☑	13	如果元件含有多部件，每个部件相同引脚编号的引脚需要连接到同一个网络	致命错误
☑	14	如果元件含有多部件，每个部件都需要出现	错误

导出 导入 恢复默认 确认 取消

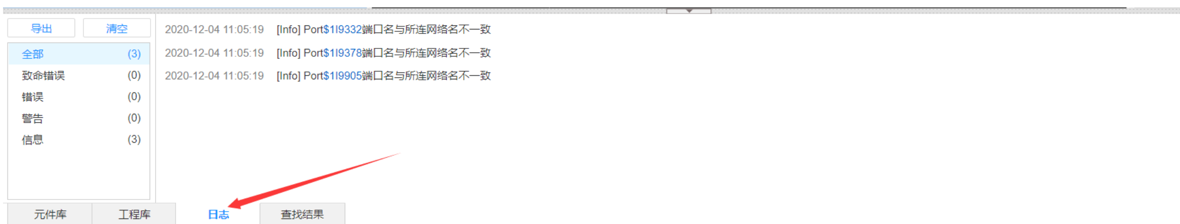
Level changes are also possible.



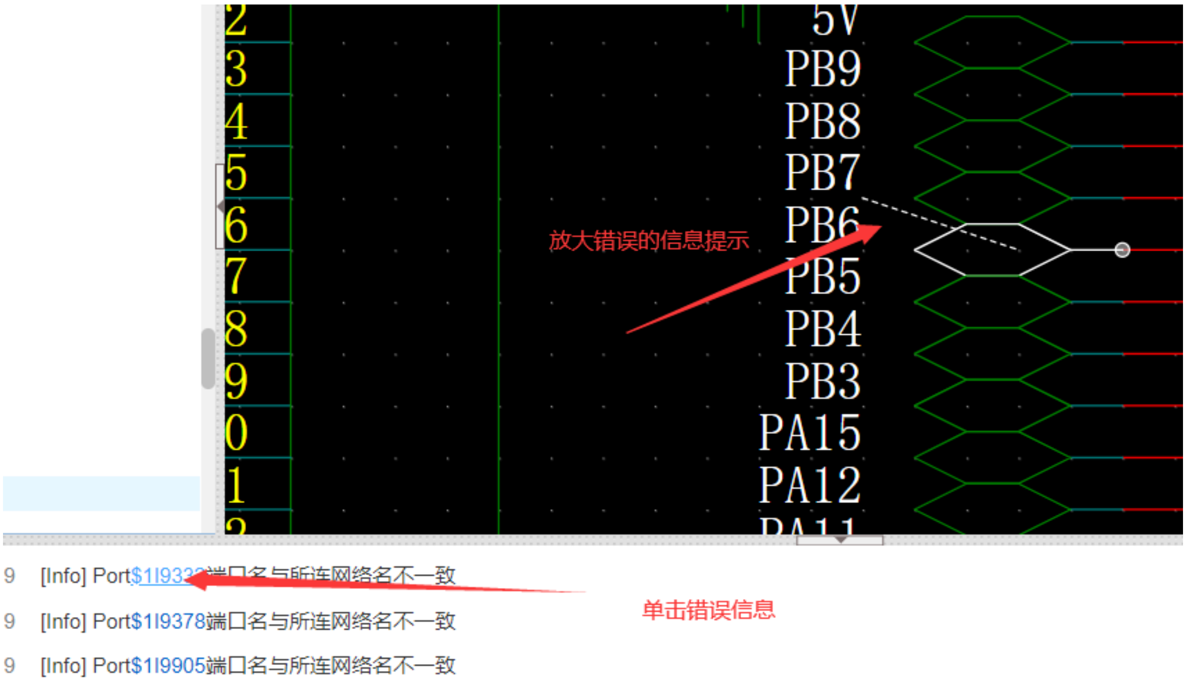
Design Rule Check (DRC)

Check that footprints, symbols, texts, etc. conform to the rules or have no conflicts before importing into the PCB.

The results of the inspection are displayed in the log at the bottom.



Click the prompt information to highlight, and double-click the prompt to highlight and locate the error point.

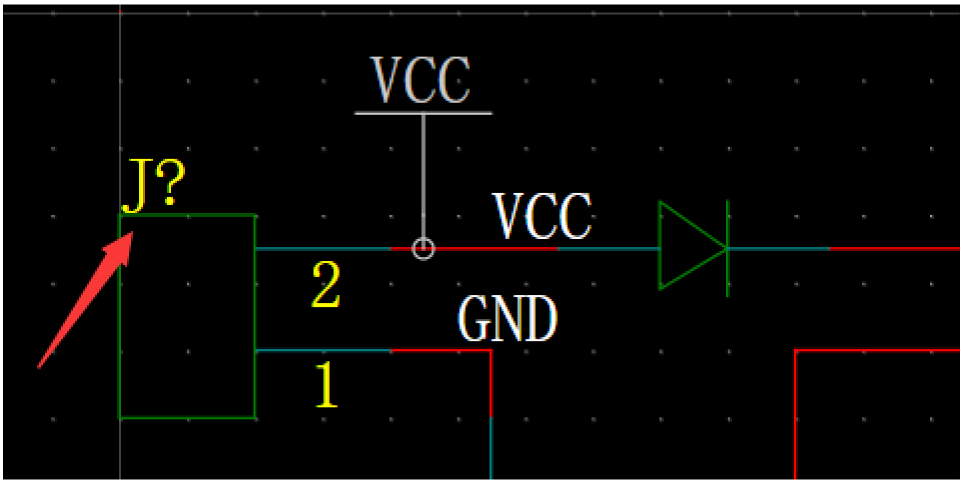


Annotate Designator

assign the designator

EasyEDA Pro supports quick assignment of tags to schematic diagrams, and does not need to be modified one by one, which greatly saves development time.

Unassigned the designator

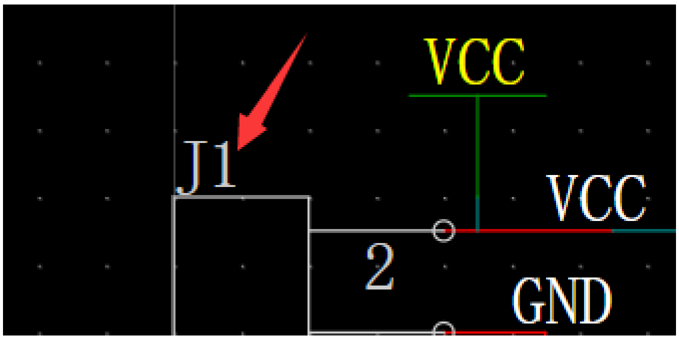


Steps:

- Top Menu - Design - Annotate Designator

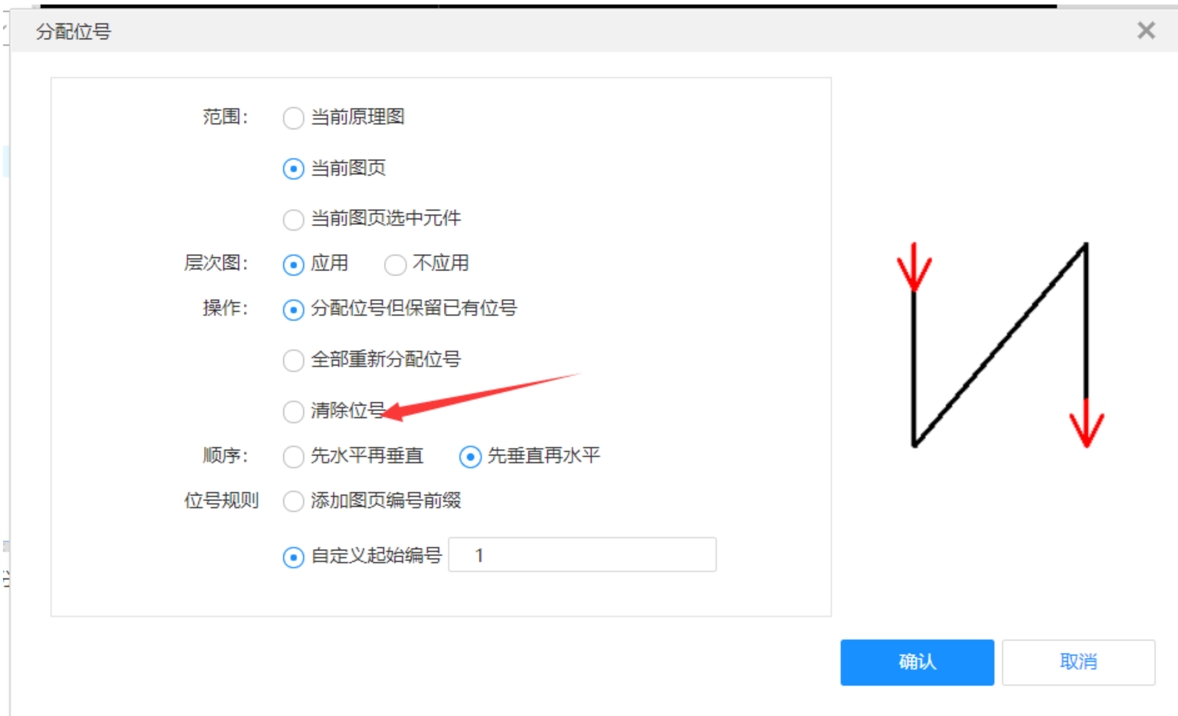
- **Range**: Range selection for assigning tag numbers.
- **Hierarchy Diagram**: The assignment tag includes the selection of the schematic device tag in the multiplexing block (hierarchy diagram).
- **Operation**: You can choose the way to assign the tag number.
- **Order**: Select the order in which tags are assigned.
- **Annotate Rules**: Unmodifiable, add the prefix of the current page number before the default device.
- **Custom start number**: You can choose which number to start assigning the tag number.

Assigned the designator



Clear the designator

There are allocation of the designator, of course, there are also clear the designator. Open the designator allocator and click Clear Tag to clear the designator of the schematic components.



Cross Probe

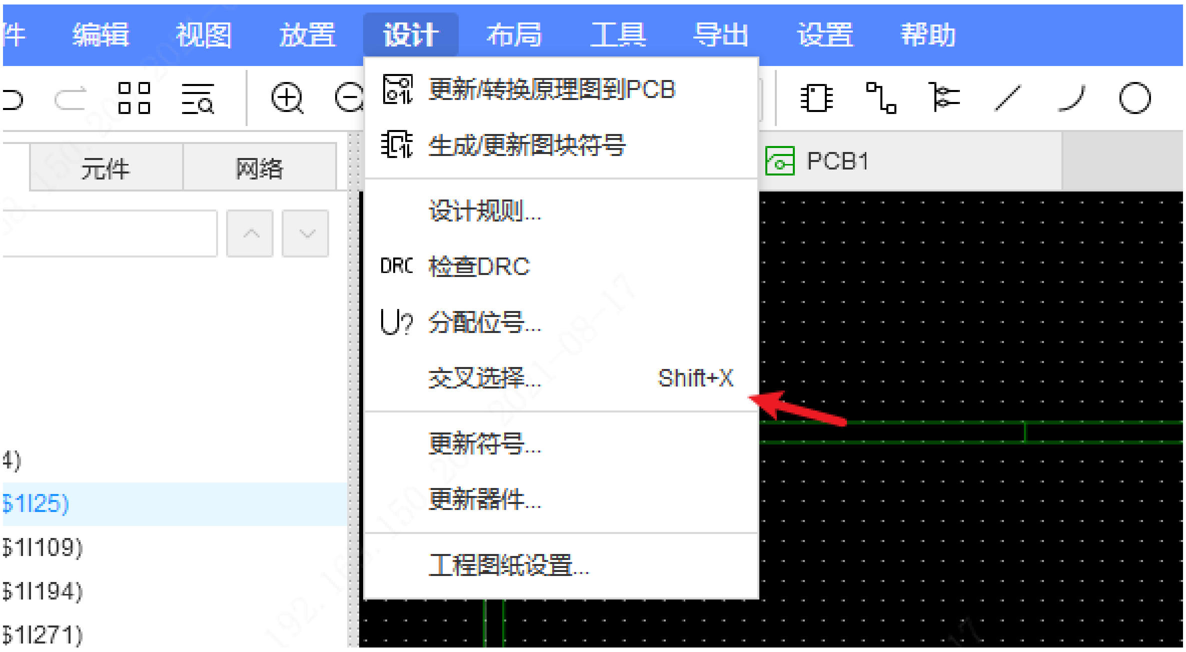
Cross Probe

Schematics and PCBs on the same board can use this cross selection to mutually locate components, pins, and wire nets.

After selecting a component in the schematic, it will highlight and select the component corresponding to the same unique ID on the PCB according to the unique ID of the component.

Instructions:

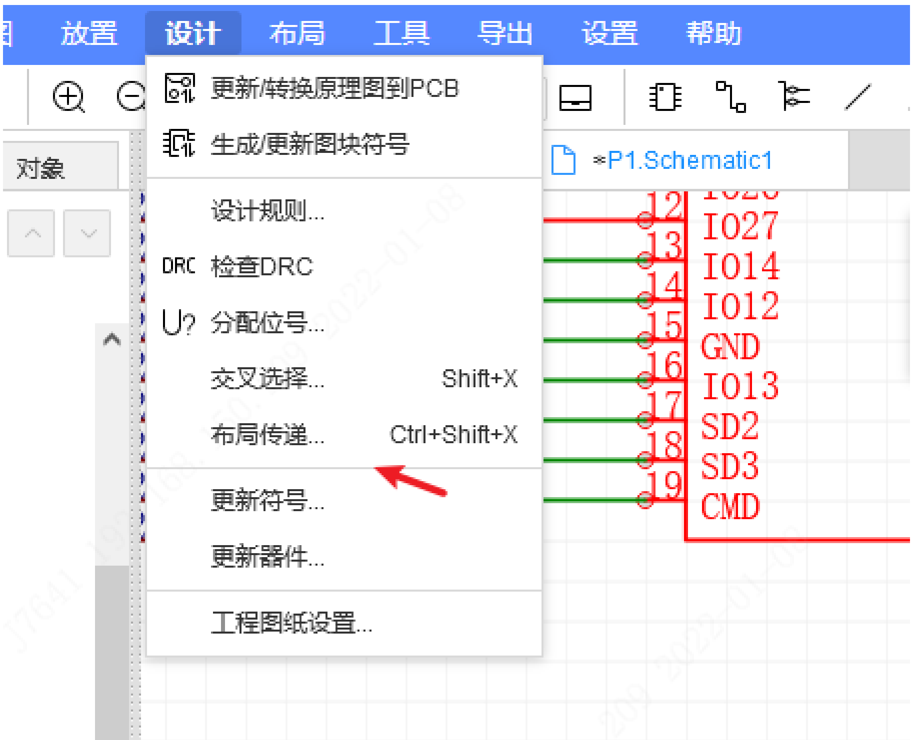
- Select Component or Wire - Top Menu - Design - Cross Probe
- Select component or wire - Shortcut `Shift`+`X`



Place Transfer

The schematic diagram and PCB under the same board can use this layout transfer to layout the components selected on the current drawing page and transfer them to the PCB, which is convenient to quickly gather the required components and quickly layout and route.

Entry: Top Menu - Design - Place Transfer.
Or Component Context Menu - Place Transfer.
Shortcut CTRL + SHIFT + X.



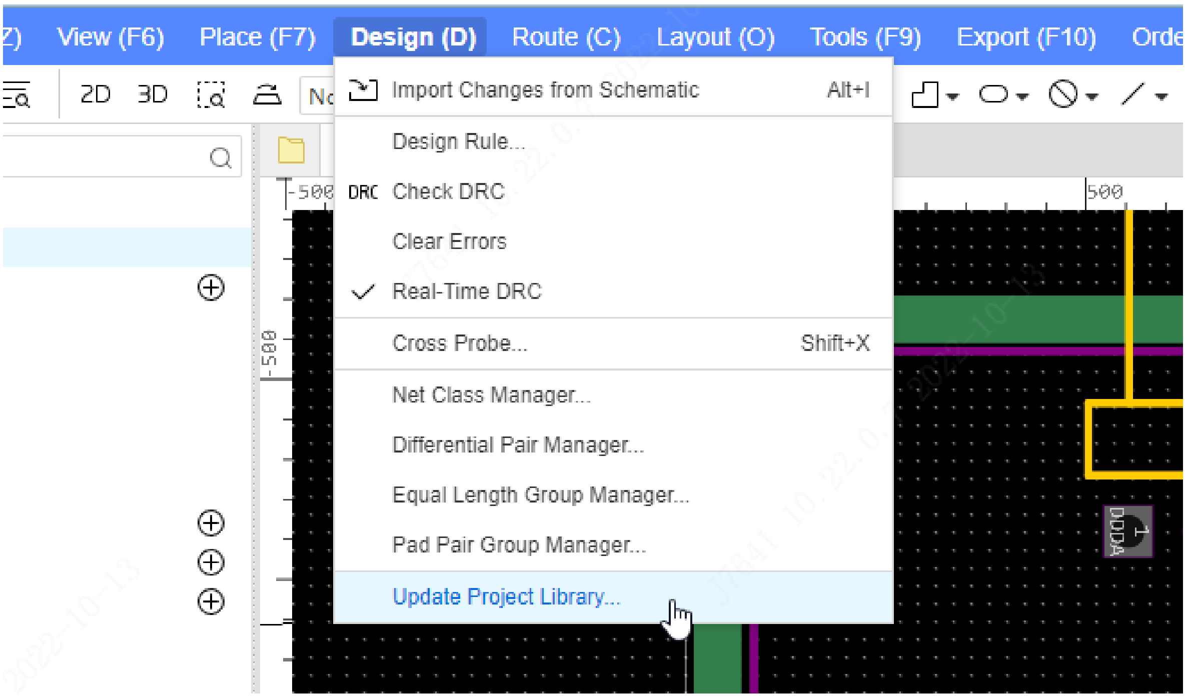
After selecting multiple components in the schematic, click "Place Transfer", it will automatically switch to the PCB, and according to the unique ID of the component in the schematic, highlight and select the PCB component corresponding to the same unique ID and hover over the mouse, waiting to be placed.

Update Project library

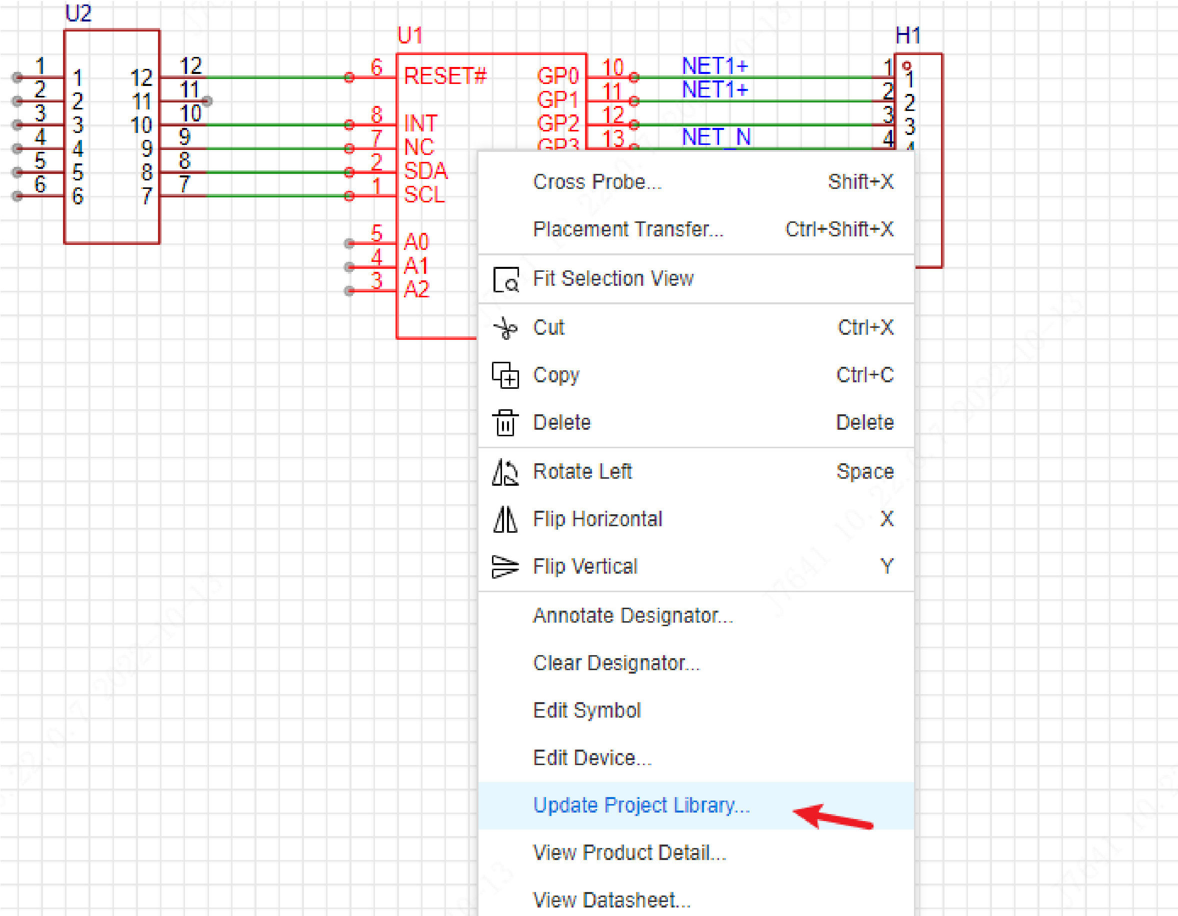
After modifying the footprint and symbols, there is no need to delete the footprint in the PCB, just update the modified footprint or device to the design drawing

How to operate:

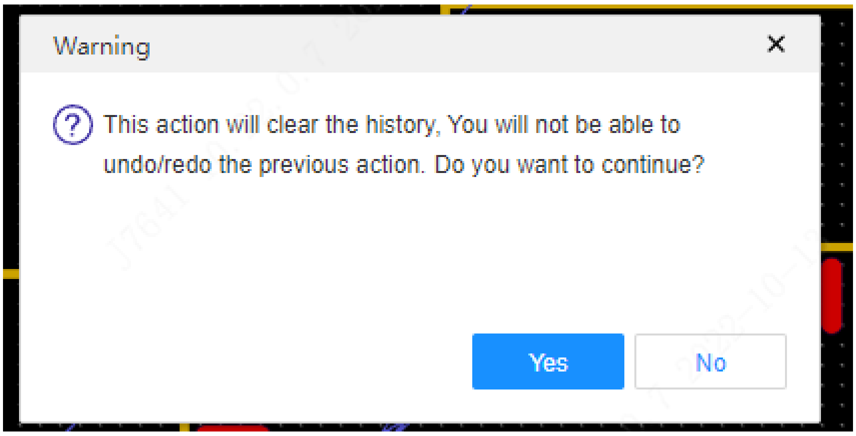
- Top Menu - Design - Update Project Library



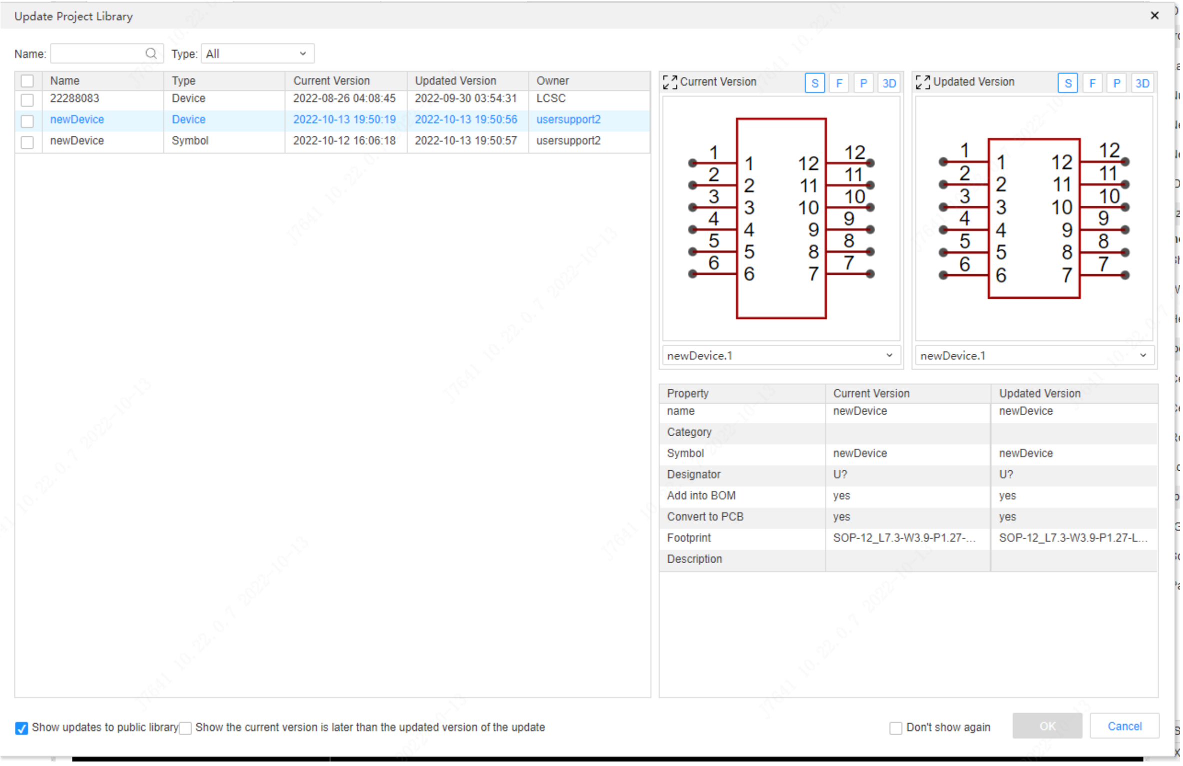
- or right-click the component - Update Project Library



will open the project library update dialog, it will list all libraries which can be updated



you can update what libraries as you want



Project Drawing Settings

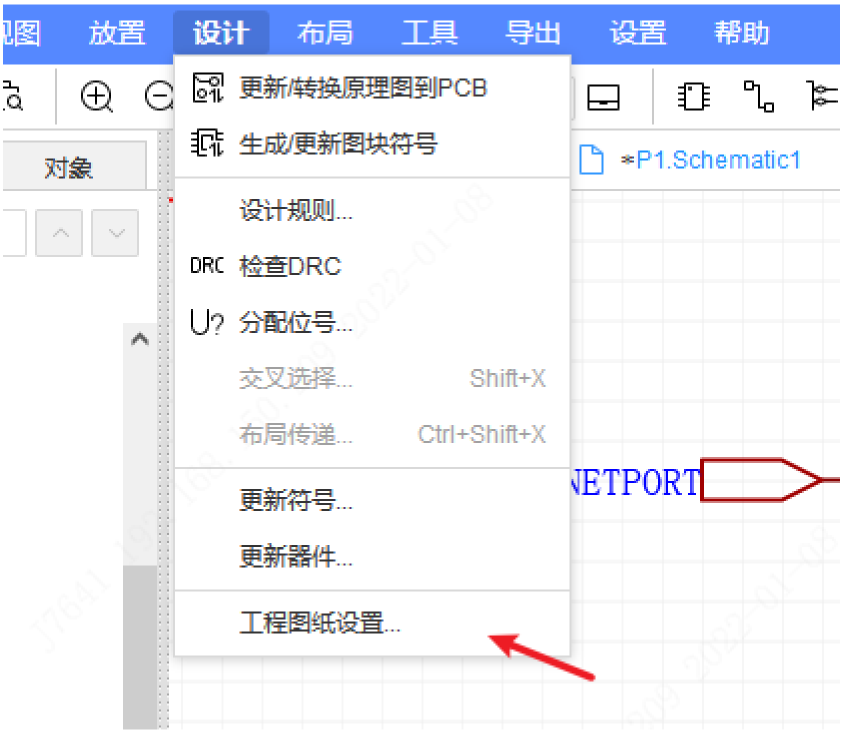
In a new project, EasyEDA Pro will set a drawing symbol to the drawing page by default.

If you need to set a custom drawing template for each new project, you can set the default drawing symbol in "Top Menu - Settings - Drawings".

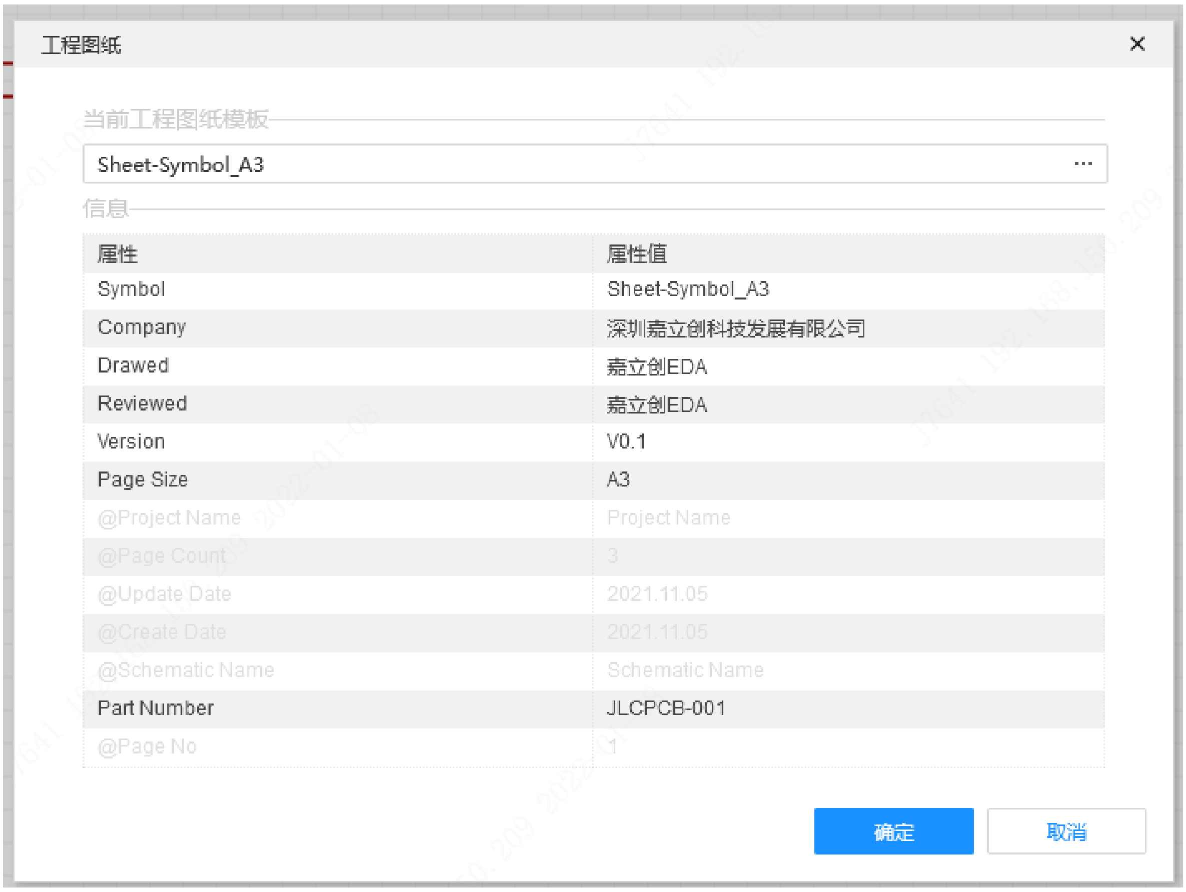
After the project is created, the drawing template of the project will be saved, and the next time a new drawing page is created, the drawing will be created according to the drawing information of the project. So we set the template information of engineering drawings.

Current engineering drawing template

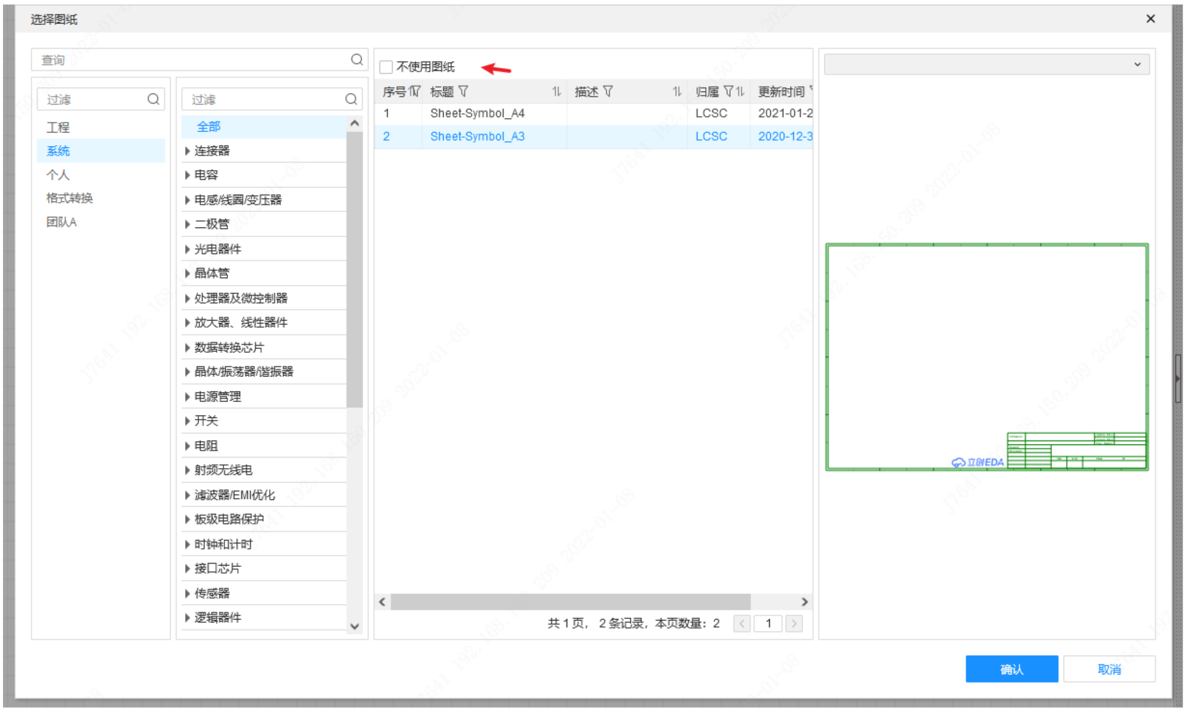
Entry: Top Menu - Design - Drawing Settings



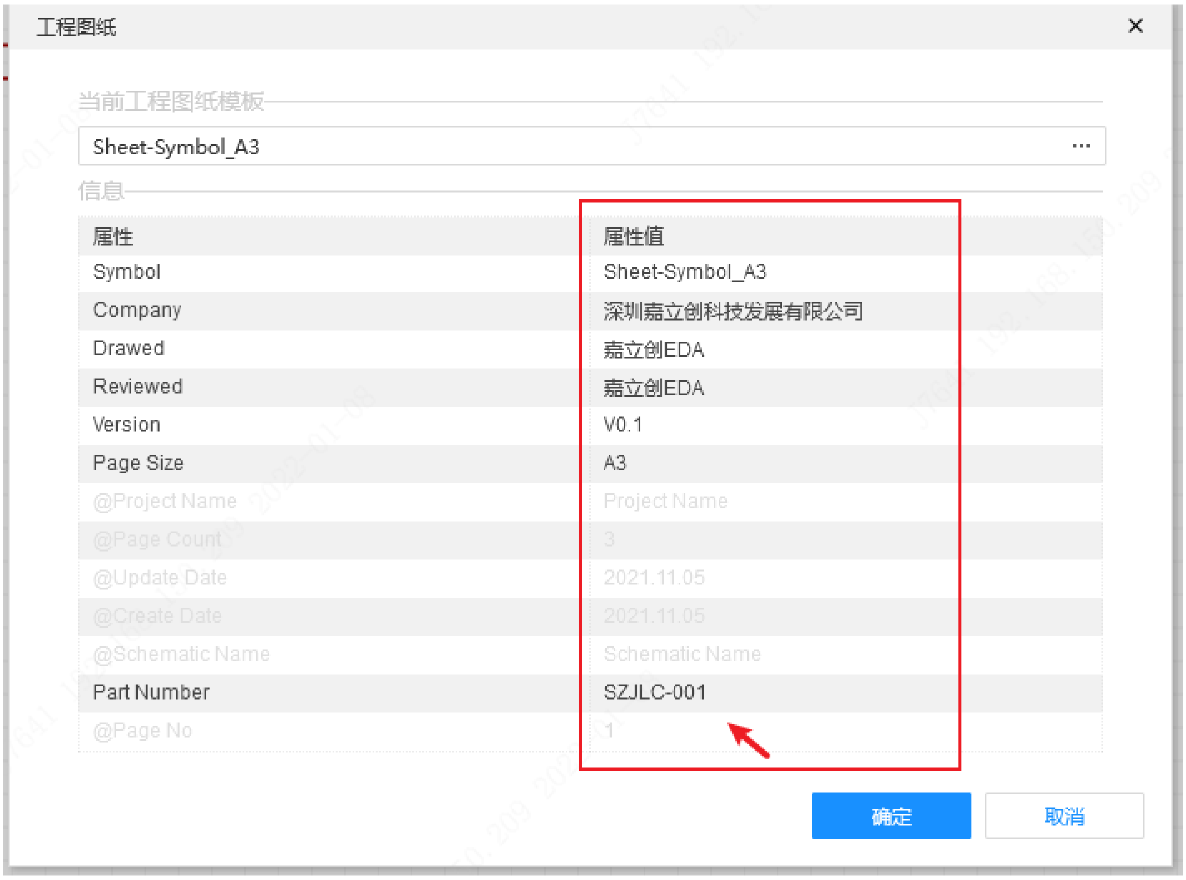
Drawing Sheet Settings dialog box:



Drawing Template: Click to select the required new template. If the subsequent drawing pages do not require drawings, check "Do not use drawings" in the opened drawing selection dialog box.

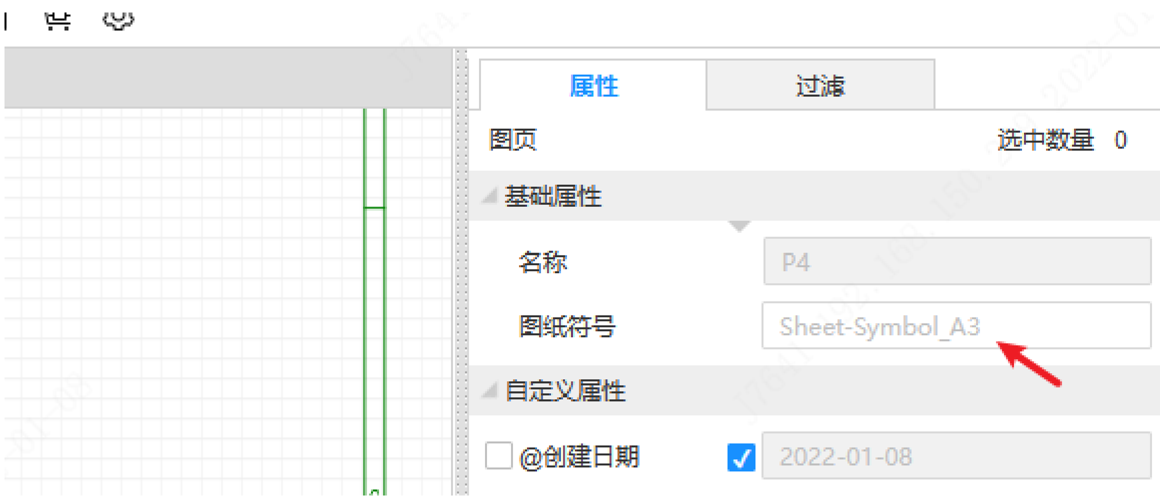


Drawing properties: Here are the properties that can be modified according to the drawing template. Modify the property values on the right side. The next time you create a drawing page, the drawing values will be generated based on the values set here.

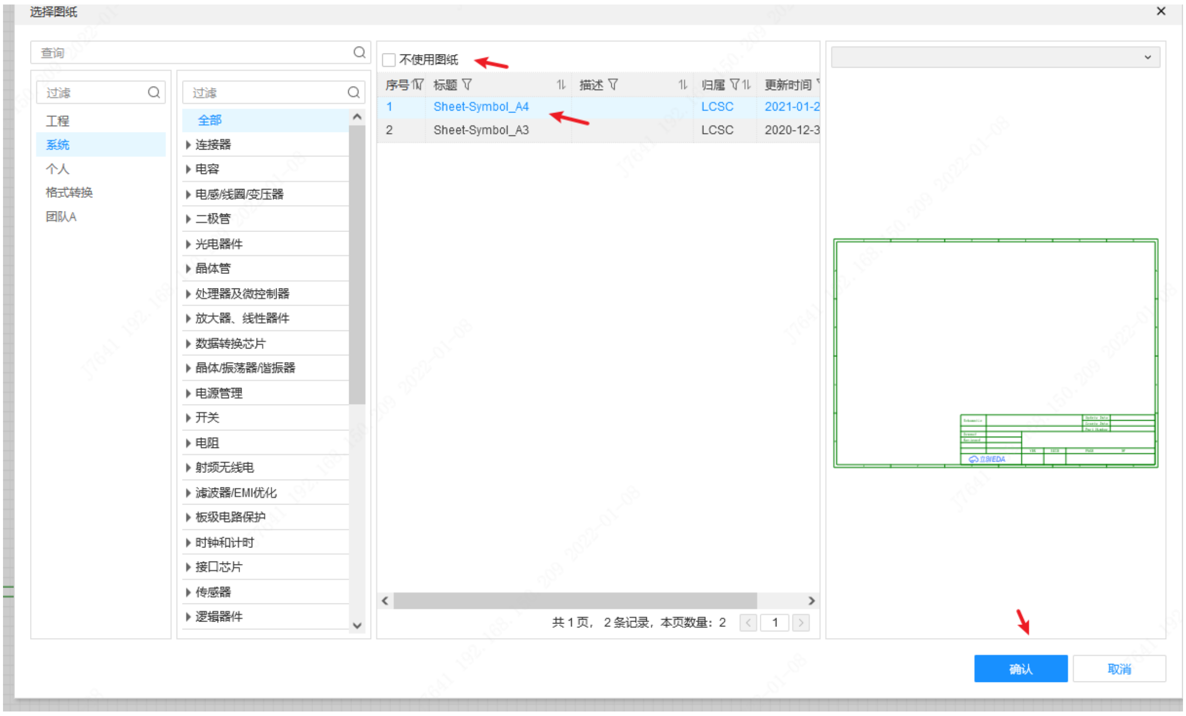


current page drawing template

If you need to change the drawing template of the current drawing page, you only need to set a new drawing symbol in the property panel on the right side of the canvas.



After selecting a new drawing, the application will take effect on the current drawing page.

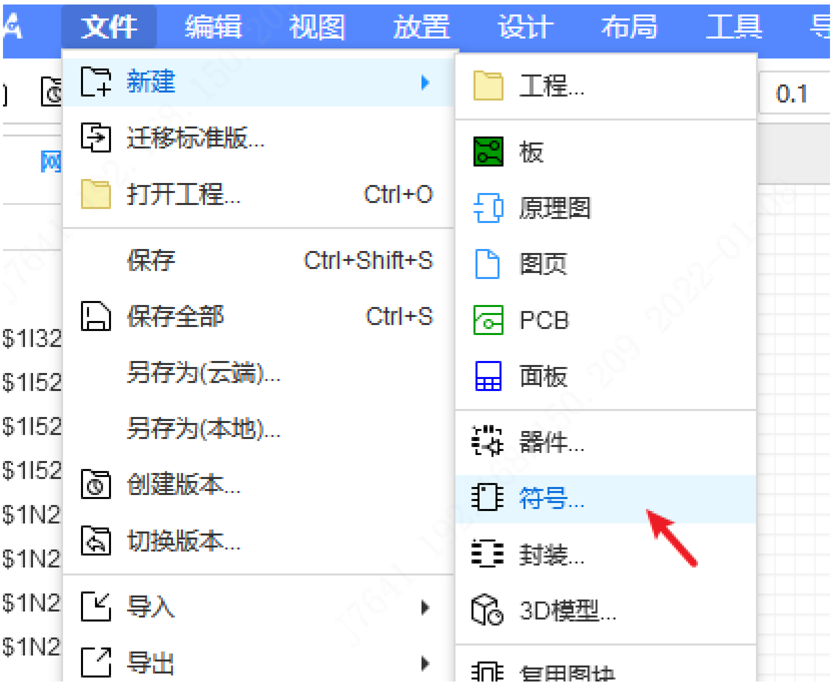


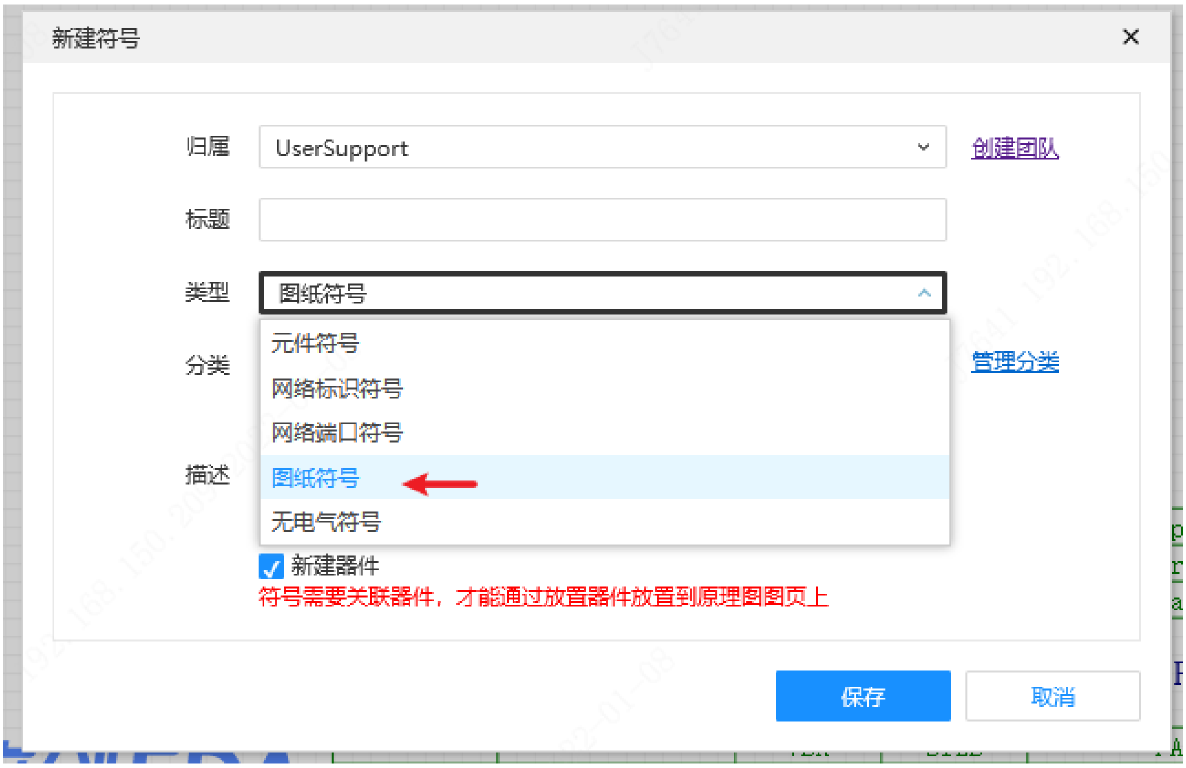
The drawing rePlace of the current drawing page is only valid for the current drawing page, and the next new drawing page will be created according to the engineering drawing settings.

Create custom drawing template

If there is no desired drawing template, you can create a new drawing symbol for association.

Entrance: Top menu - New - Symbol, choose Drawing Symbol for symbol type.





After drawing the drawing symbol, you need to create a device binding drawing symbol with the same name.

After completion, you can find and use it in system settings, engineering drawing settings, and single-page drawing switching.

Align

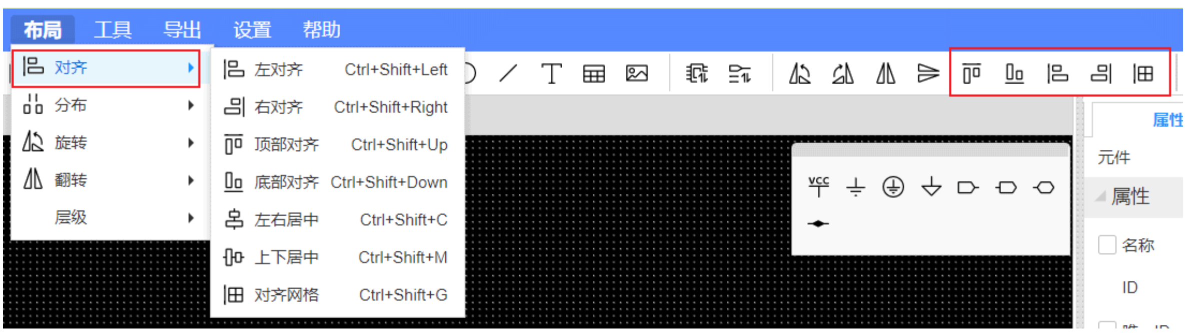
Note: Tables, symbol pins, and wires cannot be aligned during schematic alignment operations.

operate:

- Select Device - Top Menu - Layout - Align
- Find the alignment icon in the top right corner of the top toolbar

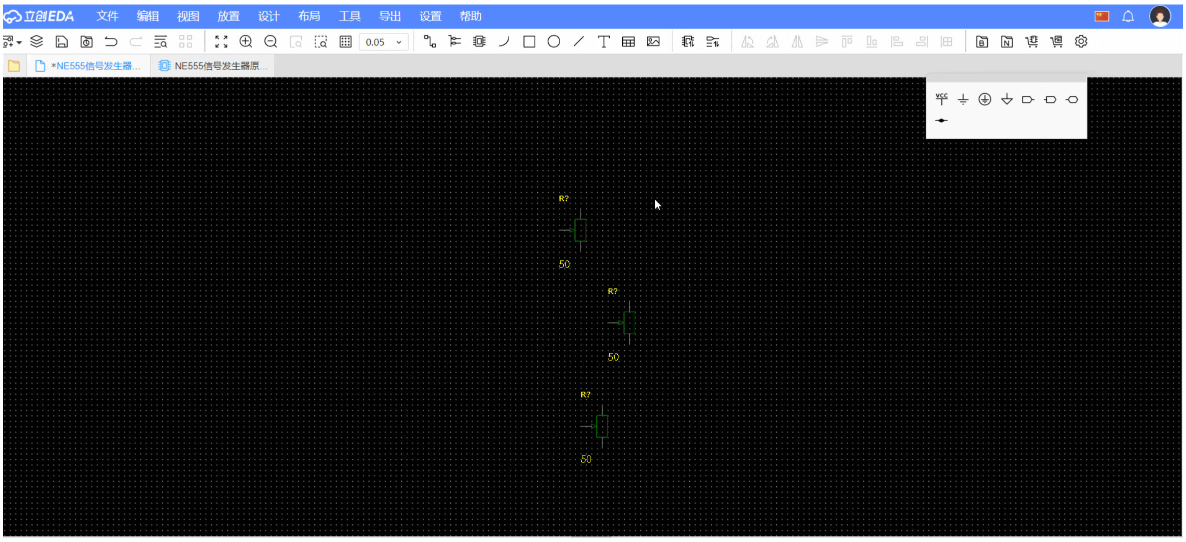
hot key

- **Left Align**: `Ctrl` + `Shift` + `←`
- **Right alignment**: `Ctrl` + `Shift` + `→`
- **Top Align**: `Ctrl` + `Shift` + `↑`
- **Bottom Align**: `Ctrl` + `Shift` + `↓`
- **Center left and right**: `Ctrl` + `Shift` + `C`
- **Center up and down**: `Ctrl` + `Shift` + `M`
- **Snap to Grid**: `Ctrl` + `Shift` + `G`



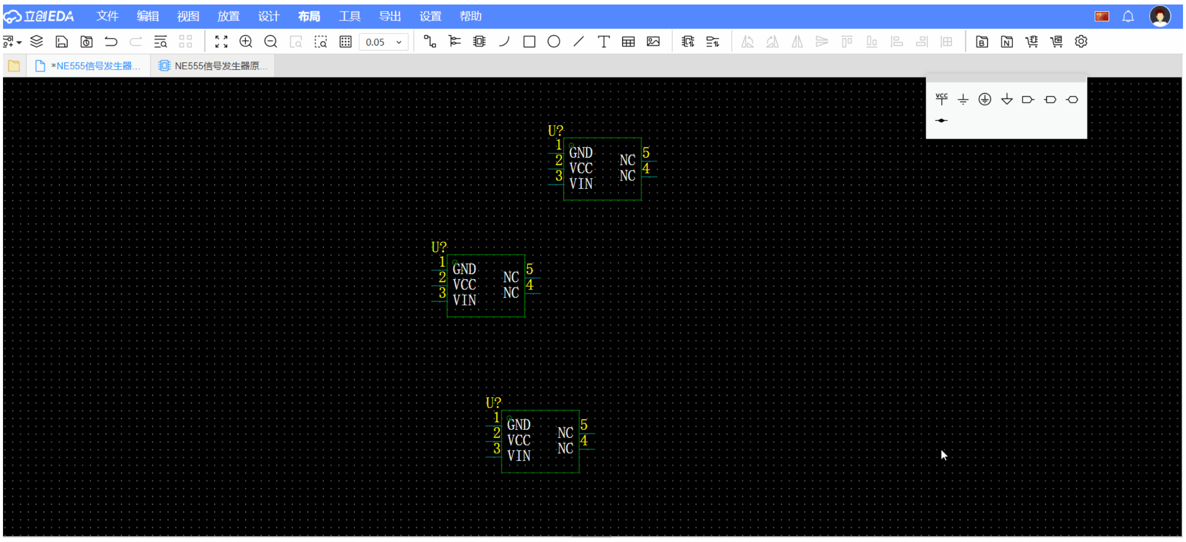
left align

Align Left is to align the leftmost device among the selected devices.



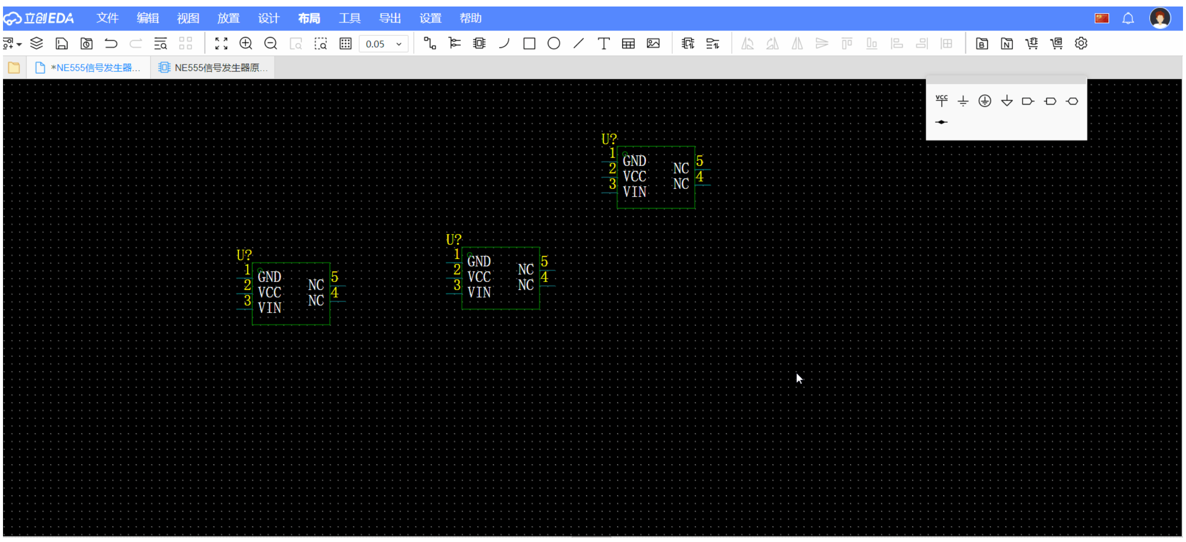
right align

Right alignment is to align the rightmost device among the selected devices.



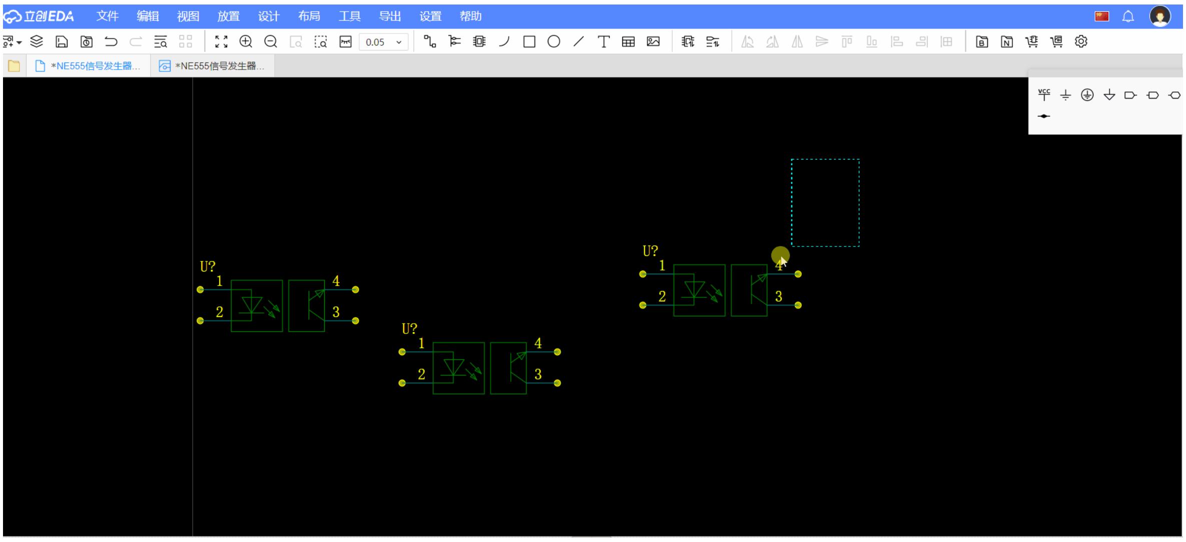
top align

Top alignment is to align the topmost device among the selected devices.



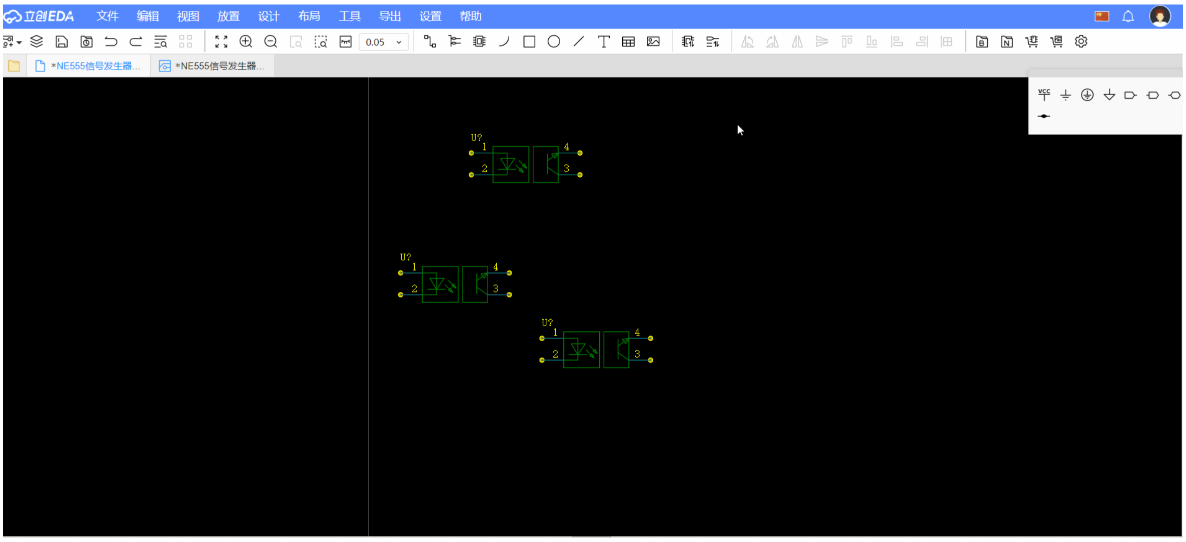
bottom align

Bottom alignment is to align the bottommost device among the selected devices as a reference point.



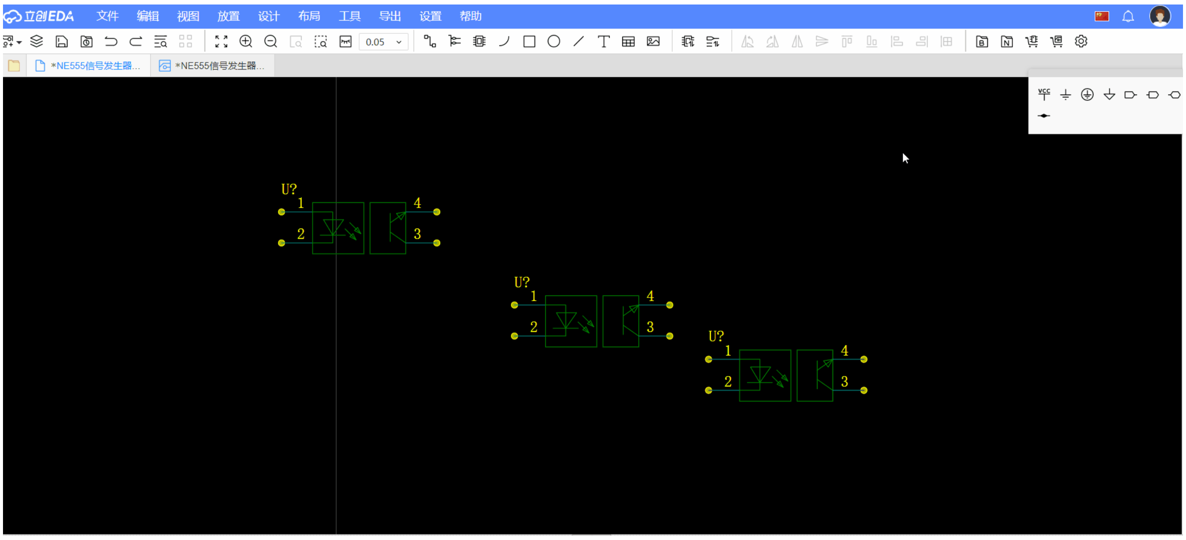
Centered left and right

Align left and right center is to find the middle point alignment on the left and right sides between the selected devices.



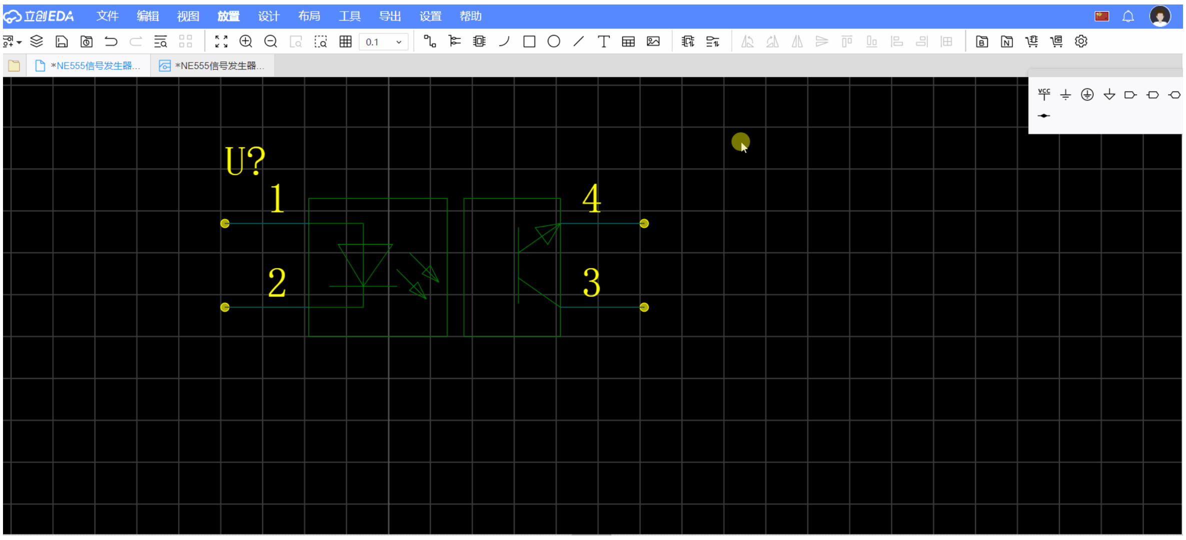
Center up and down

Line center alignment is to align the upper and lower middle points between the selected devices.



snap to grid

Align the pins of the selected device in the grid of the schematic settings.



distribute

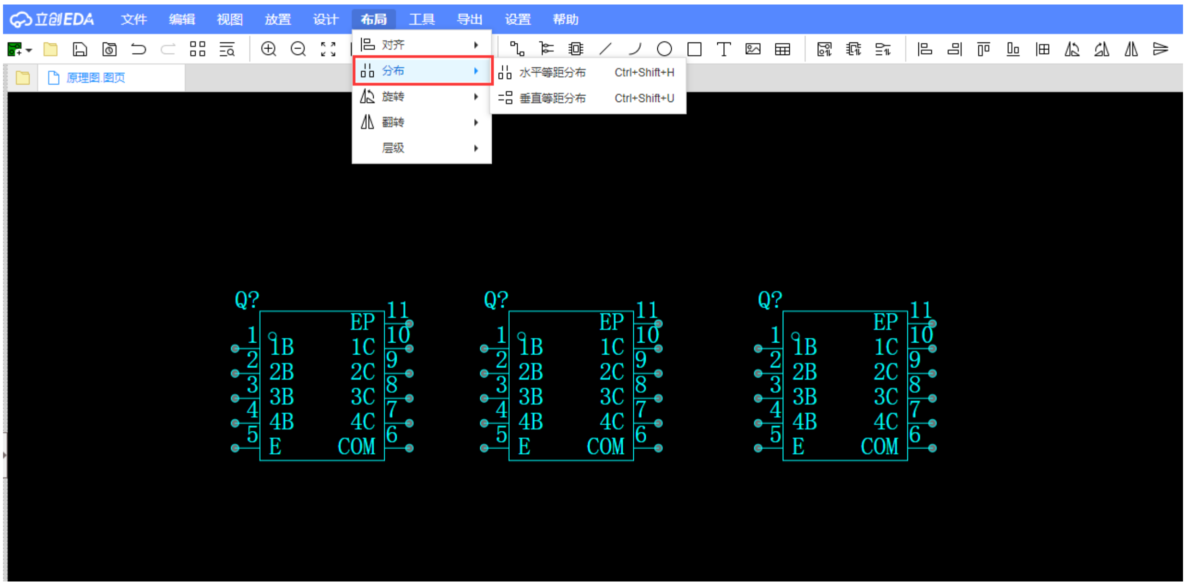
The selected multiple devices are arranged in an equidistant distribution. The premise of the equidistant arrangement and distribution is that more than two devices can be arranged and distributed.

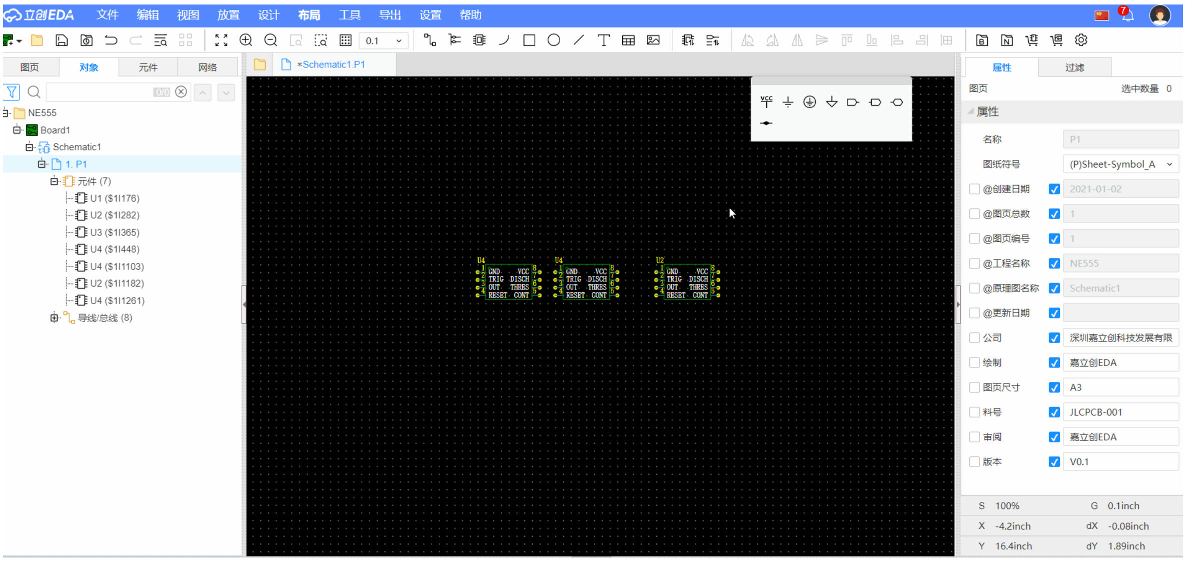
Horizontal equidistant distribution

Arrange the selected horizontal devices equally spaced.

Steps:

- Select the devices to be arranged and distributed - Top menu - Layout - Distribution - Horizontal equidistant distribution
- Select the device to be arranged and distributed - Shortcut keys `Ctrl` + `Shift` + `H`



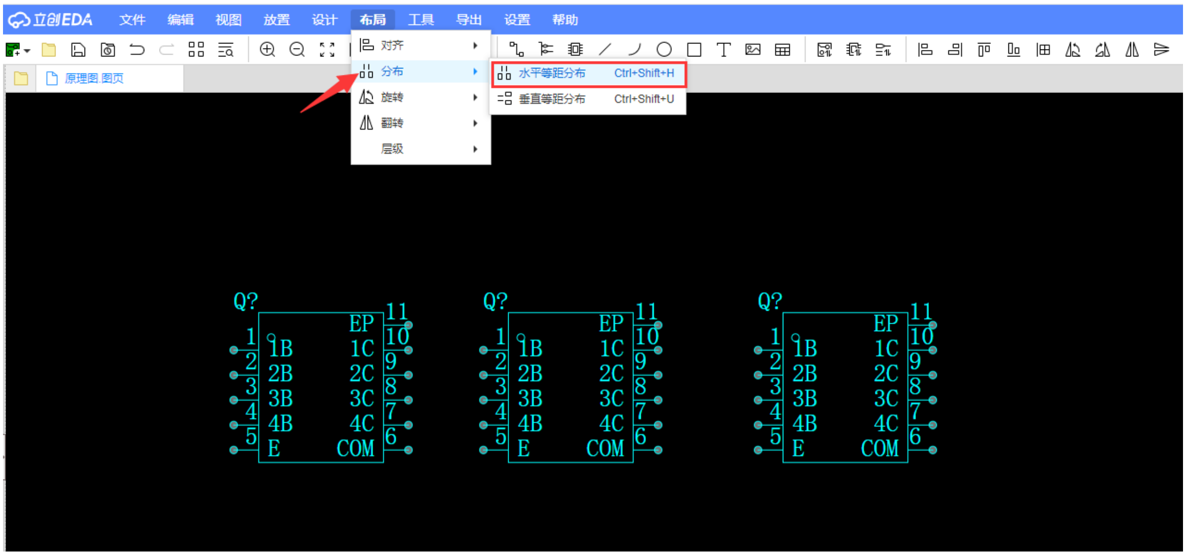


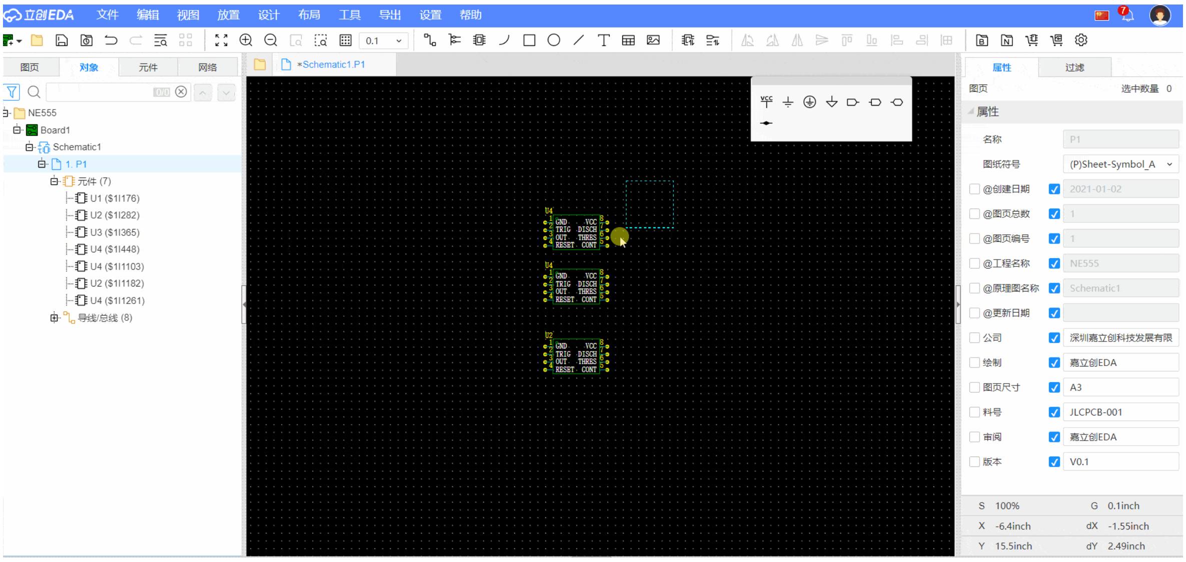
Vertical equidistant distribution

Arrange the selected vertically oriented devices in an equidistant distribution.

Steps:

- Select the devices to be arranged and distributed - Top menu - Layout - Distribution - Vertical equidistant distribution
- Select the device to be arranged and distributed - Shortcut keys `Ctrl` + `Shift` + `U`





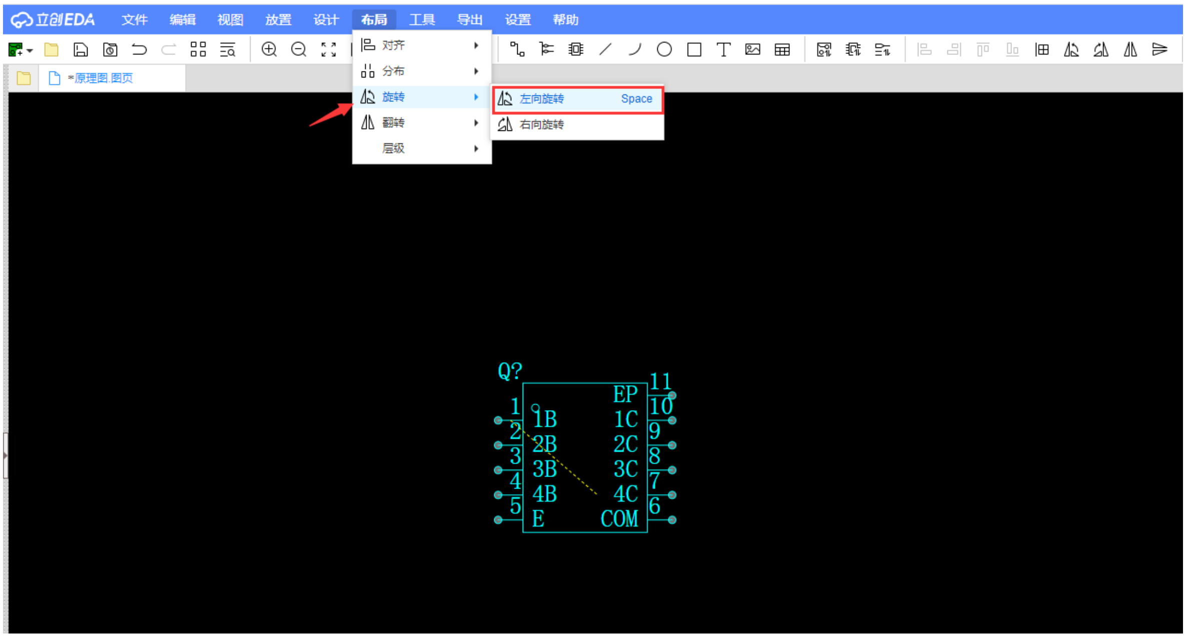
rotate

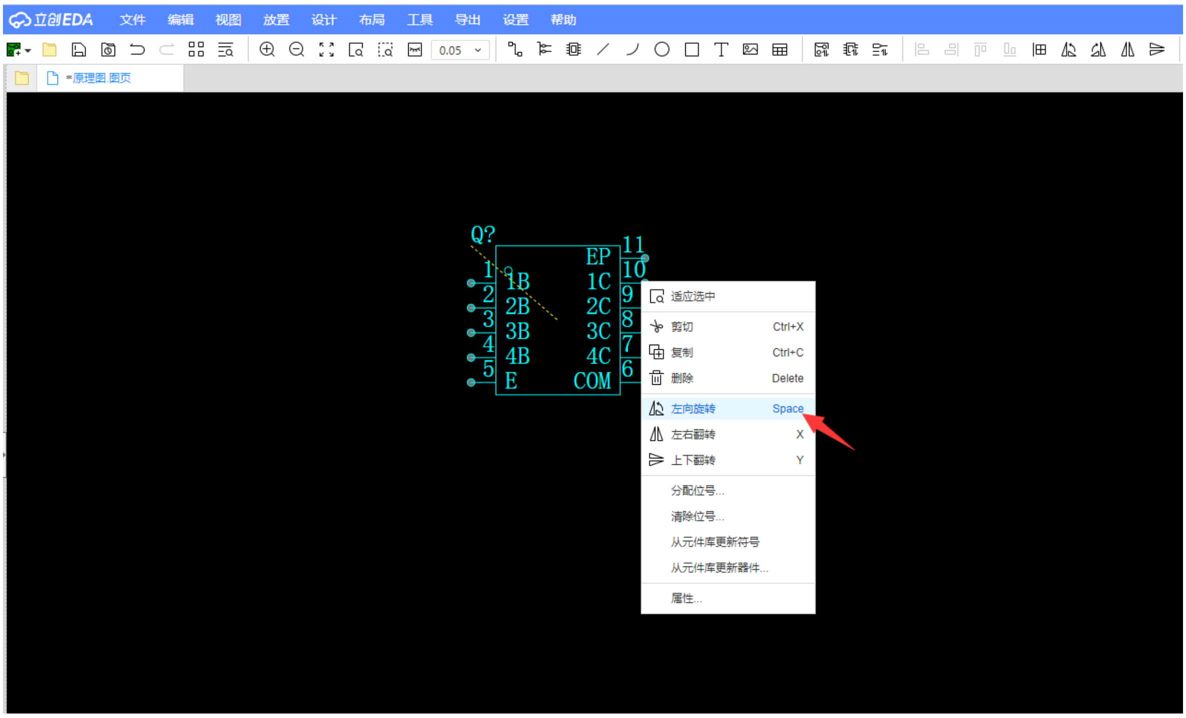
Rotation in EasyEDA Pro does not support drag rotation, it can only be rotated in the selected state.

Note: Schematic does not support 45 degree rotation

Steps:

- .Select Device - Layout - Rotate
- Keyboard `Space` key to rotate
- right click



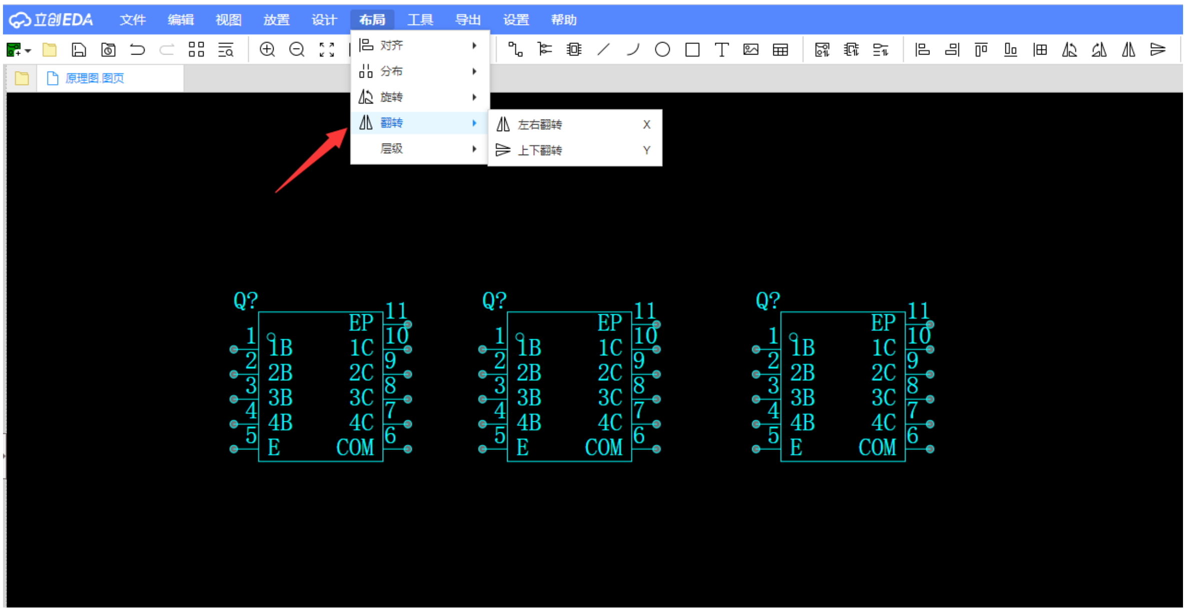


flip

Flip pictures, text, etc., devices and wires do not support flipping.

Steps:

- Top Menu - Layout - Flip
- Shortcut keys X / Y

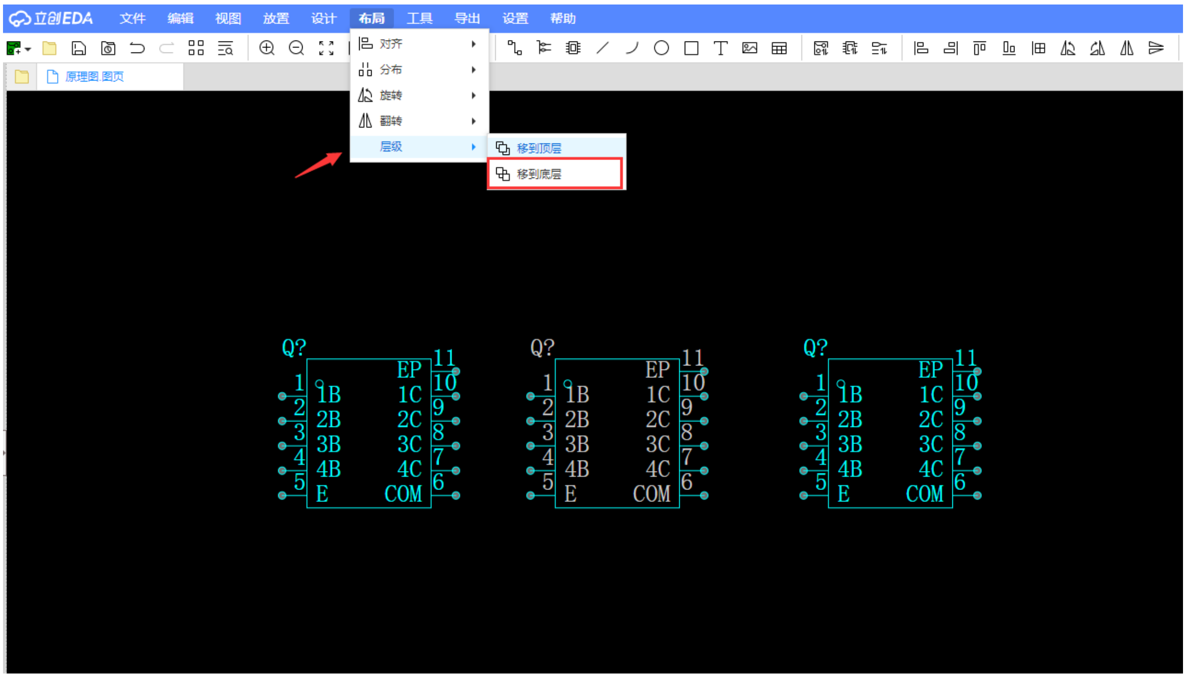


level

Element Place level modification.

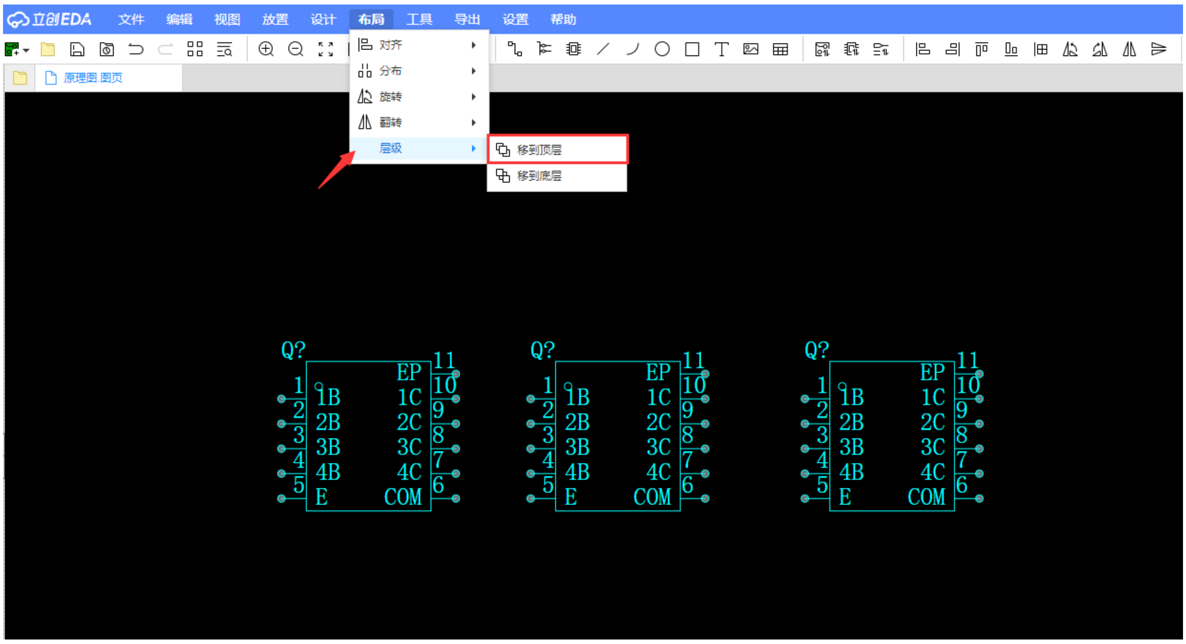
move to the bottom layer

Select the element to be moved - Top Menu - Layout - Hierarchy - Move to Bottom



move to top level

Select the element to be moved - Top Menu - Layout - Hierarchy - Move to Top



log

Inspection tips and backup information for ERC electrical rules are displayed in the log in the bottom panel

Top Menu - Tools - Logs

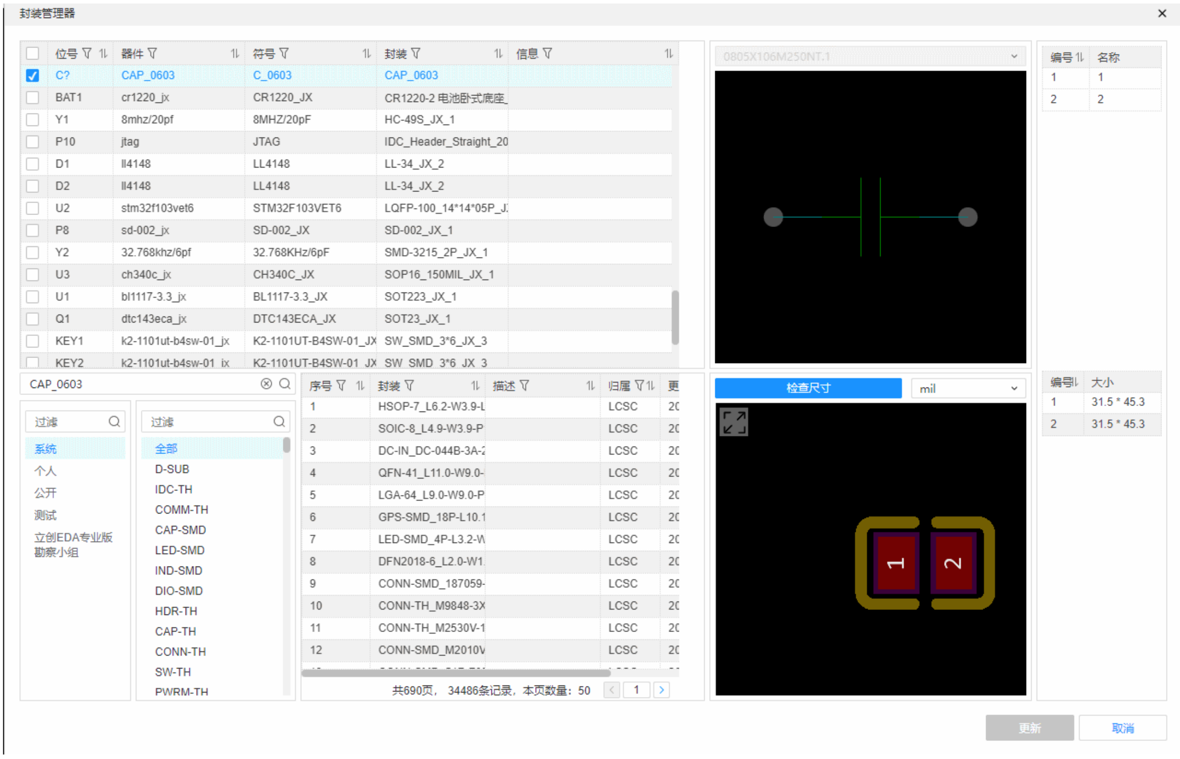
footprint Manager

footprint manager

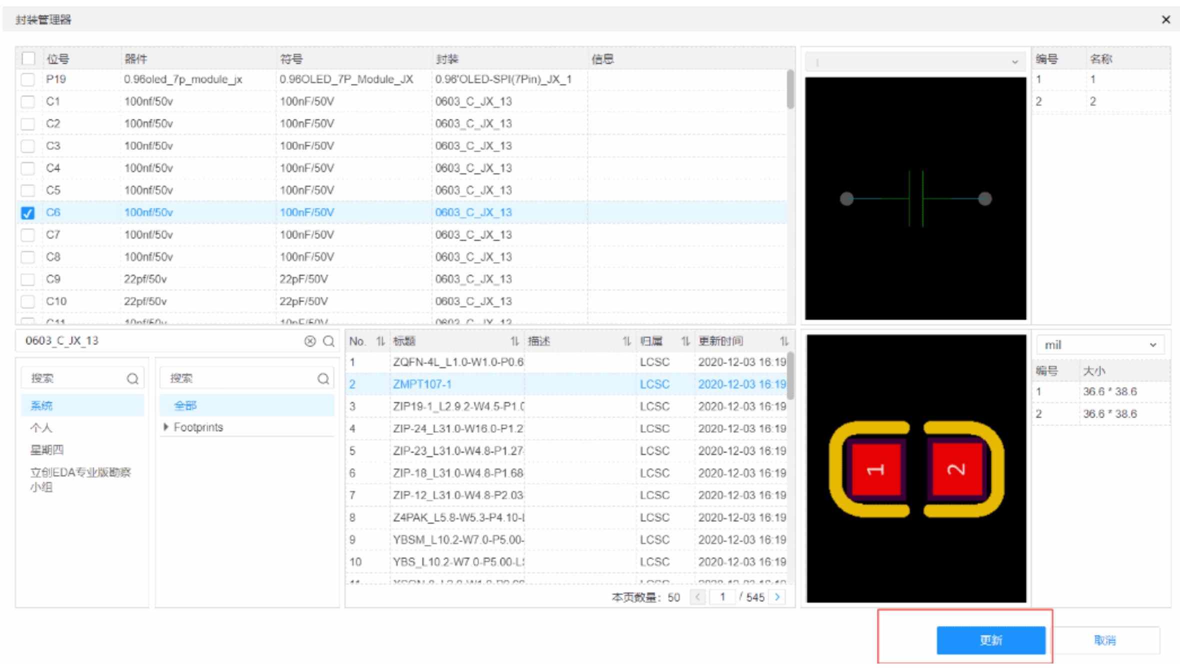
Modify the footprint in batches. If there are many devices in the same footprint in the schematic, then modifying the footprint in batches will greatly reduce your workload.

Open the footprint manager in the schematic interface, the top menu - tools - footprint manager

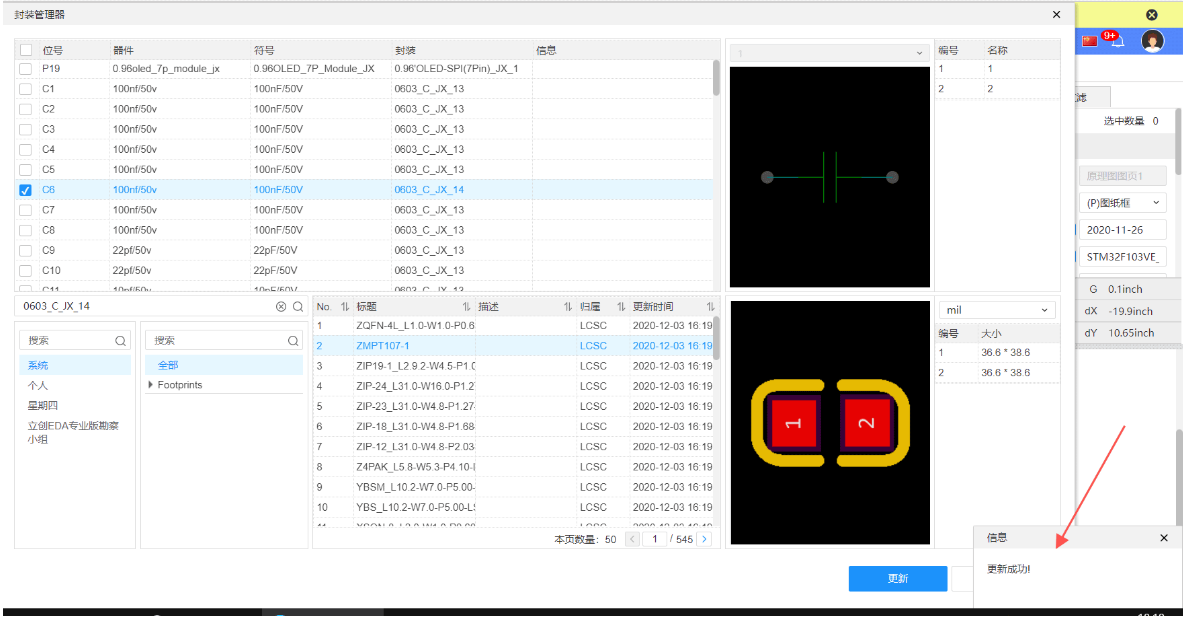
After opening, you can see the symbols and footprints of all devices under the project. The upper side is the symbol, and the lower side is the footprint.



Select the device to be modified and click Update to update the latest footprint to the entire project.



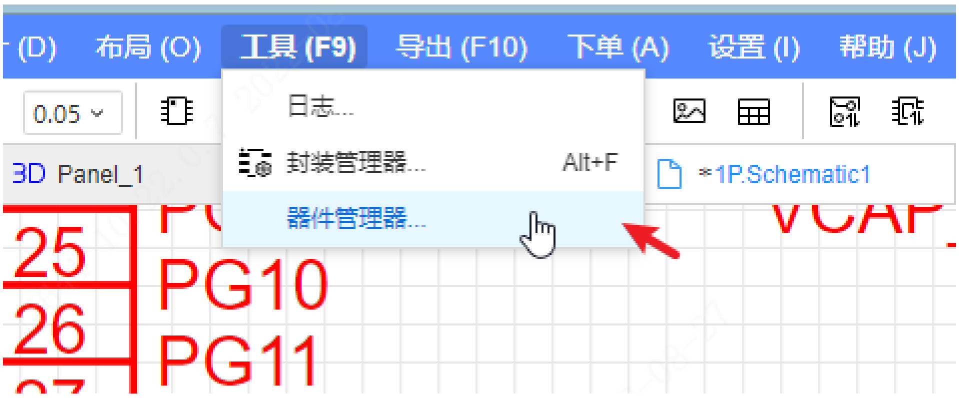
When the update is complete, a prompt will appear in the lower right corner.



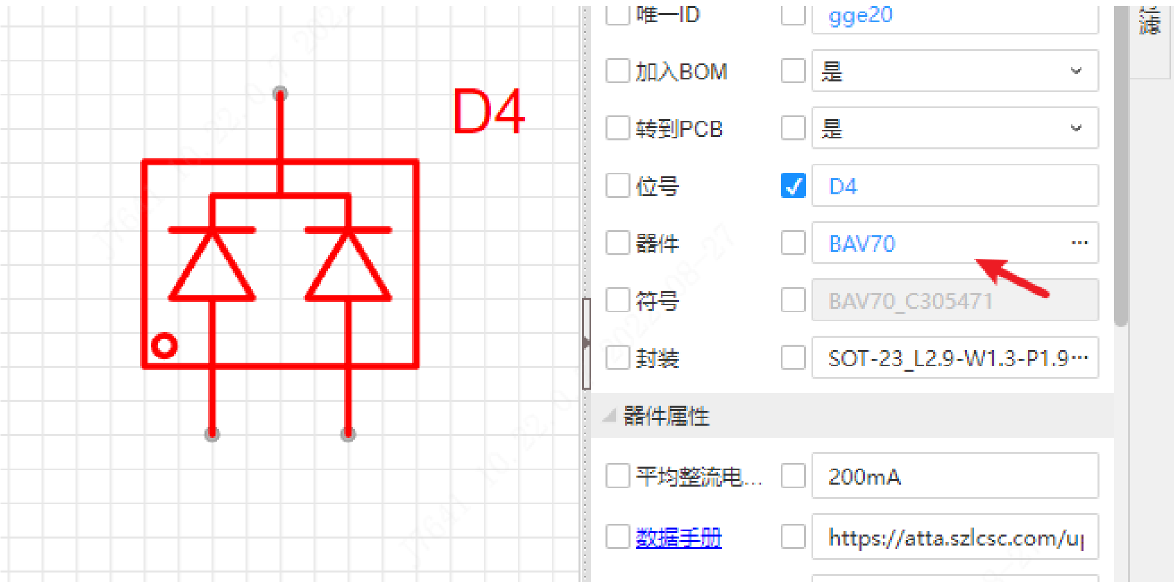
Device Manager

EasyEDA supports device replacement through the device manager. It is convenient to modify the material properties of the device, and it is convenient to output a BOM with a unified specification.

Entry: Top Menu - Tools - Device Manager



Or: select the device - right property panel - click the device property input box



Click to open the Device Manager dialog. After selecting the device on the left, search for the target device to be replaced, and replace it directly after checking.

Fig.154

Before replacement, you can preview the symbol and package, physical map, and properties of the target device.

After the replacement, all attributes except the tag number and unique ID will be replaced. Substitution of symbols and packages is not currently supported.

Export BOM

Export BOM

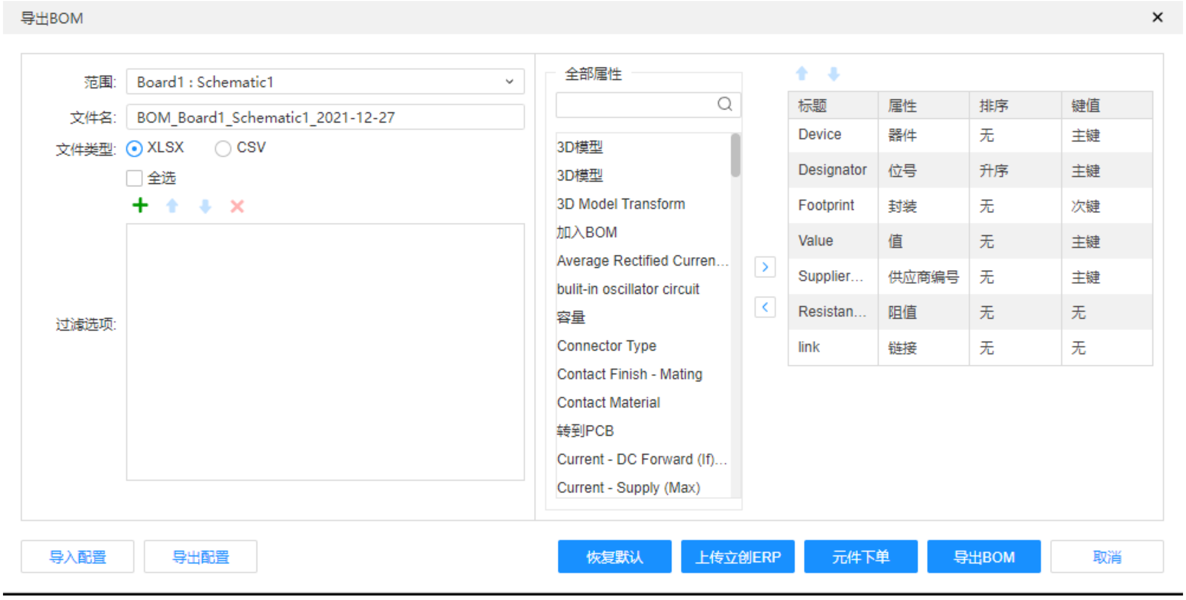
EasyEDA Pro supports separate PCB export BOM (Bill of Materials) so that you can purchase the required parts.

- Top Menu - File - Export - Bill of Materials BOM
- Top Menu - Manufacturing - Bill of Materials BOM

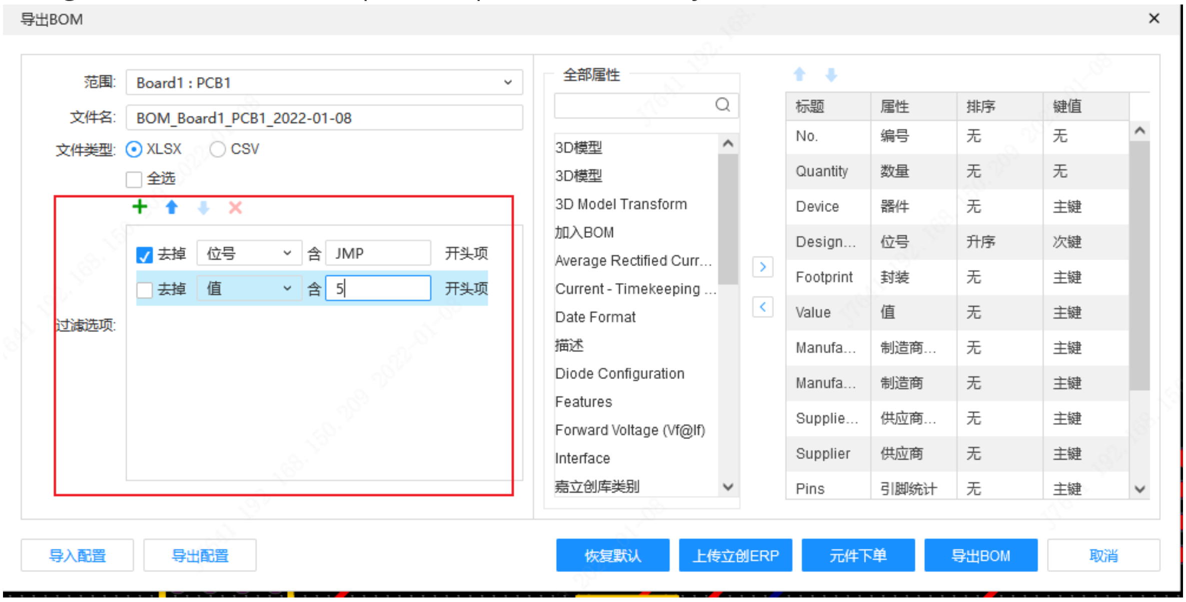
Scope: Select the PCB of the exported project or the BOM of the schematic

Filename: The filename of the exported BOM

File Type: Only XLSX and CSV formats are supported

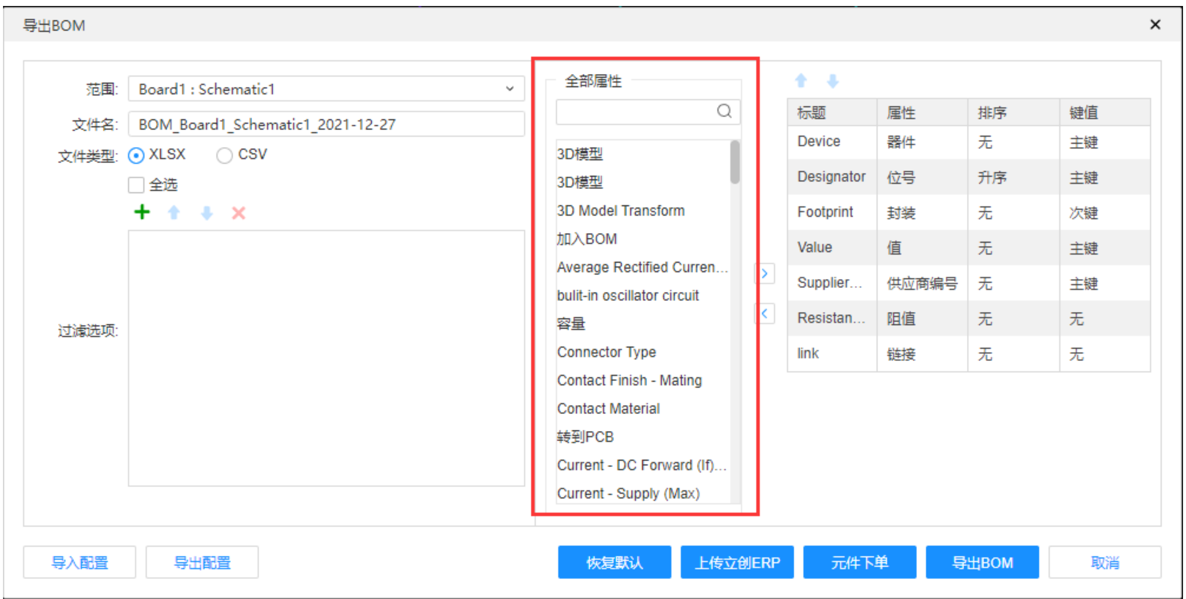


Filter options: Support adding filter rules to filter unnecessary components. The filter configuration will be saved in personal preferences and synchronized in the cloud.

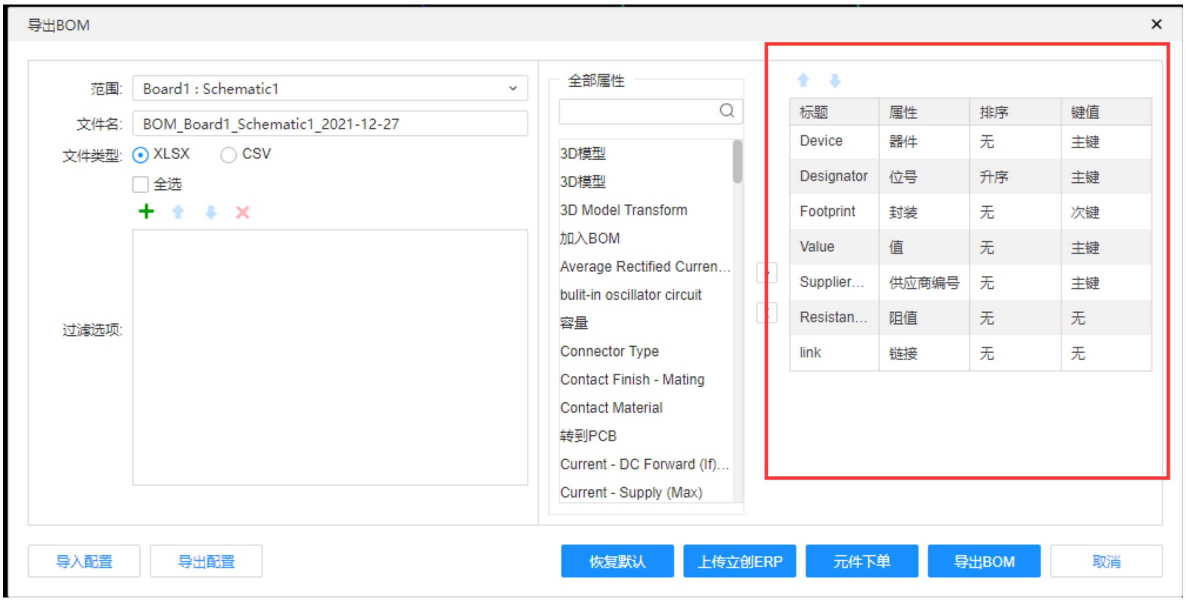


All attributes:

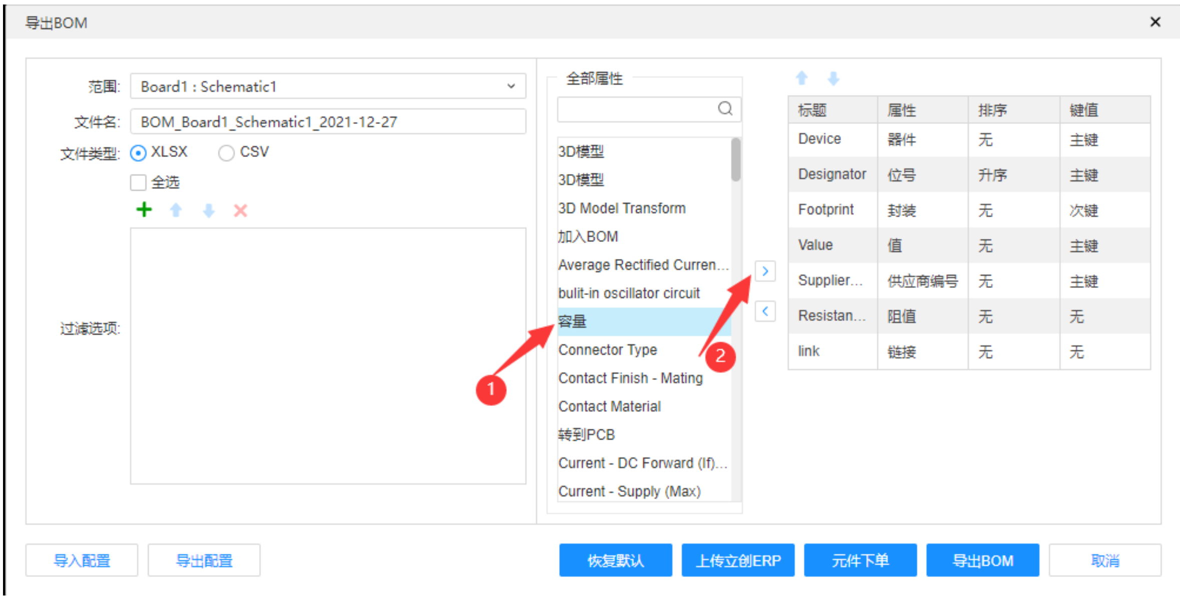
In the middle is the type of BOM or the properties of the device.



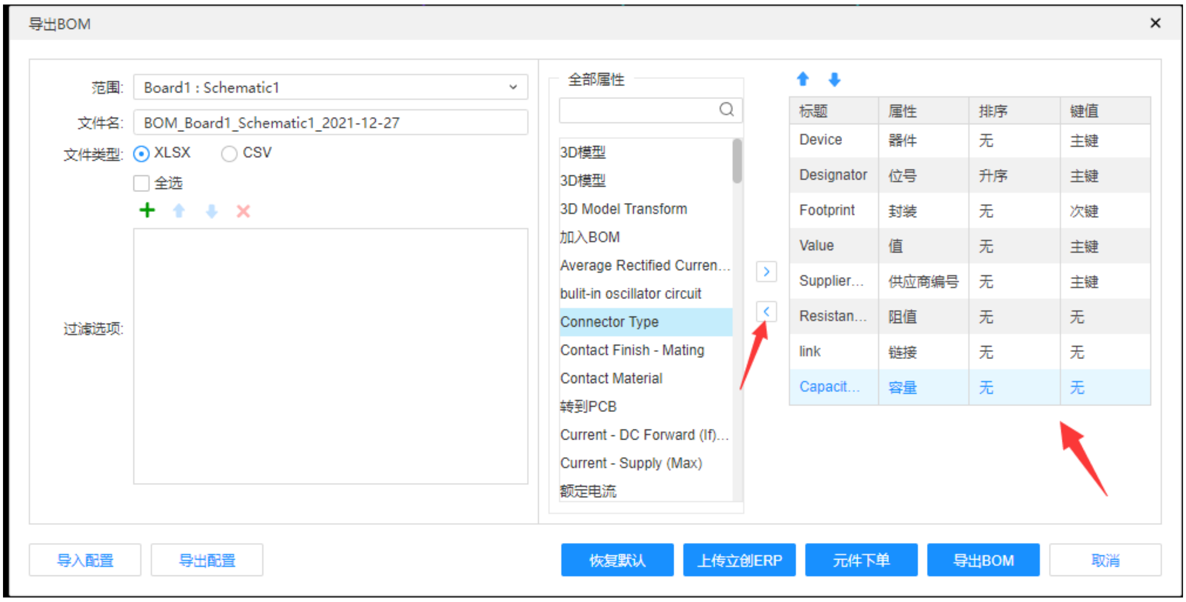
On the right is the option to export the contents of the BOM.



Select the content to be exported and click Add to add it into the BOM.



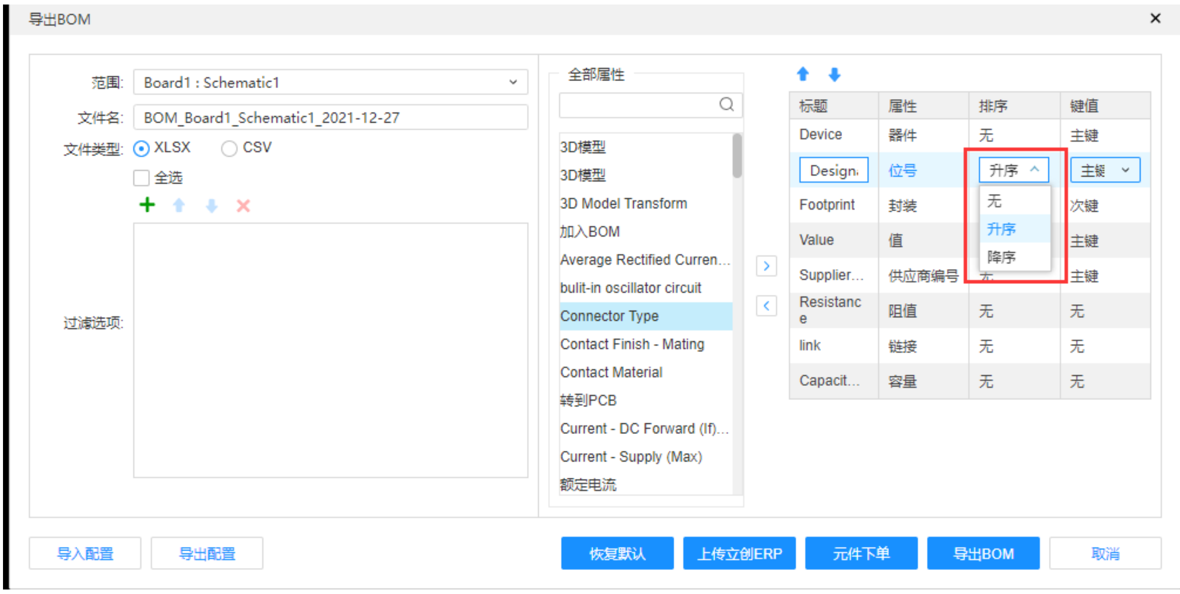
The removal operation is also the same operation. Just select the category to be removed on the right, click the small arrow, and the selected category can be removed from the BOM.



To add a type of sorting, you need to double-click the content to be changed on the right, or select the type and click the up and down arrows at the top to change the order.

- **Title**: The title of the exported BOM;
- **Attribute**: the relevant attribute type of the device;
- **sort**: export the sorting order of BOM properties;

Key value



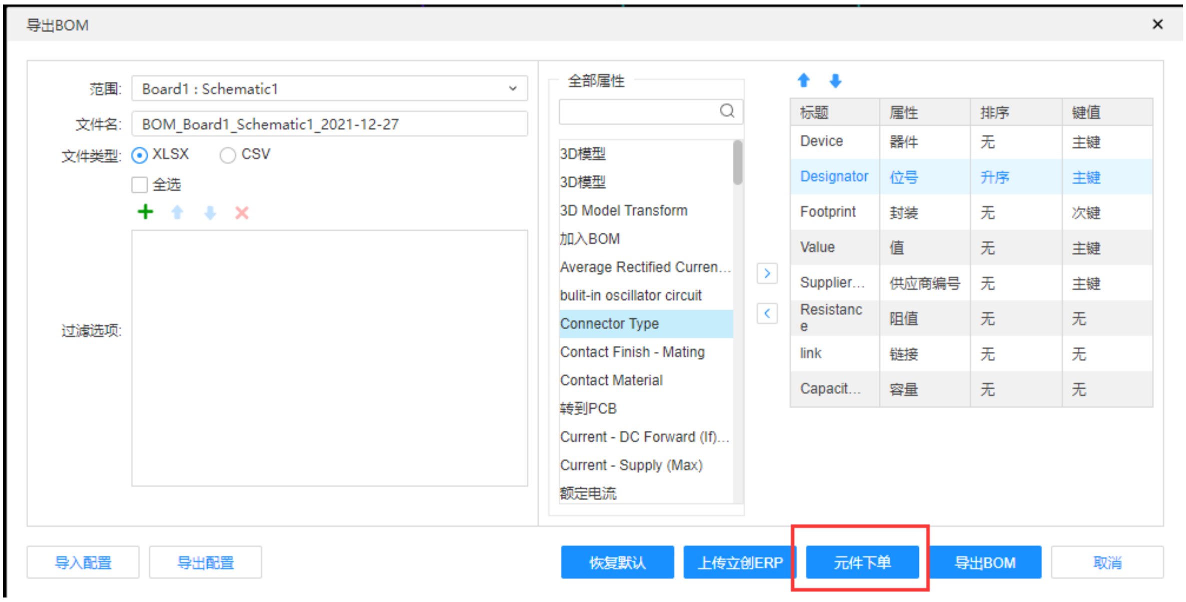
- **Primary key**: When exporting the same attribute to the BOM table, the values are arranged separately;
- **Secondary key**: When exporting the same attribute to the BOM table, the values are merged and arranged;

Exported BOM

A	B	C	D
No.	Device	Designator	Footprint
1	0.5a/6v	F1	0805_L_JX_1
2	100nf/50v	C2, C4, C5, C6, C7, C8, C9	0603_C_JX_13
3	bl1117-3.3_jx	U1	SOT223_JX_1
4	mini-usb_jx	P1, P2	USB_Mini_B_Fe_JX_2
5	1.5k/1%	R1	0603_R_JX_11
6	red/led	LED0	0603_D_JX_3
7	4.7k/1%	R2, R3, R4	0603_R_JX_11
8	green/led	LED1	0603_D_JX_3
9	blue/led	LED2	0603_D_JX_3
10	0r/1%	R5	0603_C_JX_13
11	114148	D1, D2	LL-34_JX_2
12	0r/1%	R6	0603_R_JX_11
13	8mhz/20pf	Y1	HC-49S_JX_1
14	10pf/50v	C14, C13	0603_C_JX_13
15	32.768khz/6pf	Y2	SMD-3215_2P_JX_1
16	k2-1101ut-b4sw-01_j	WKUP1, KEY1, KEY2, RESE	SW_SMD_3*6_JX_3

Parts order

EasyEDA Pro supports BOM ordering. On the BOM export interface, select components to place an order, and the system will generate BOM data. Click OK to jump to the BOM matching interface of LCSC.



LCSC part store: https://lcsc.com

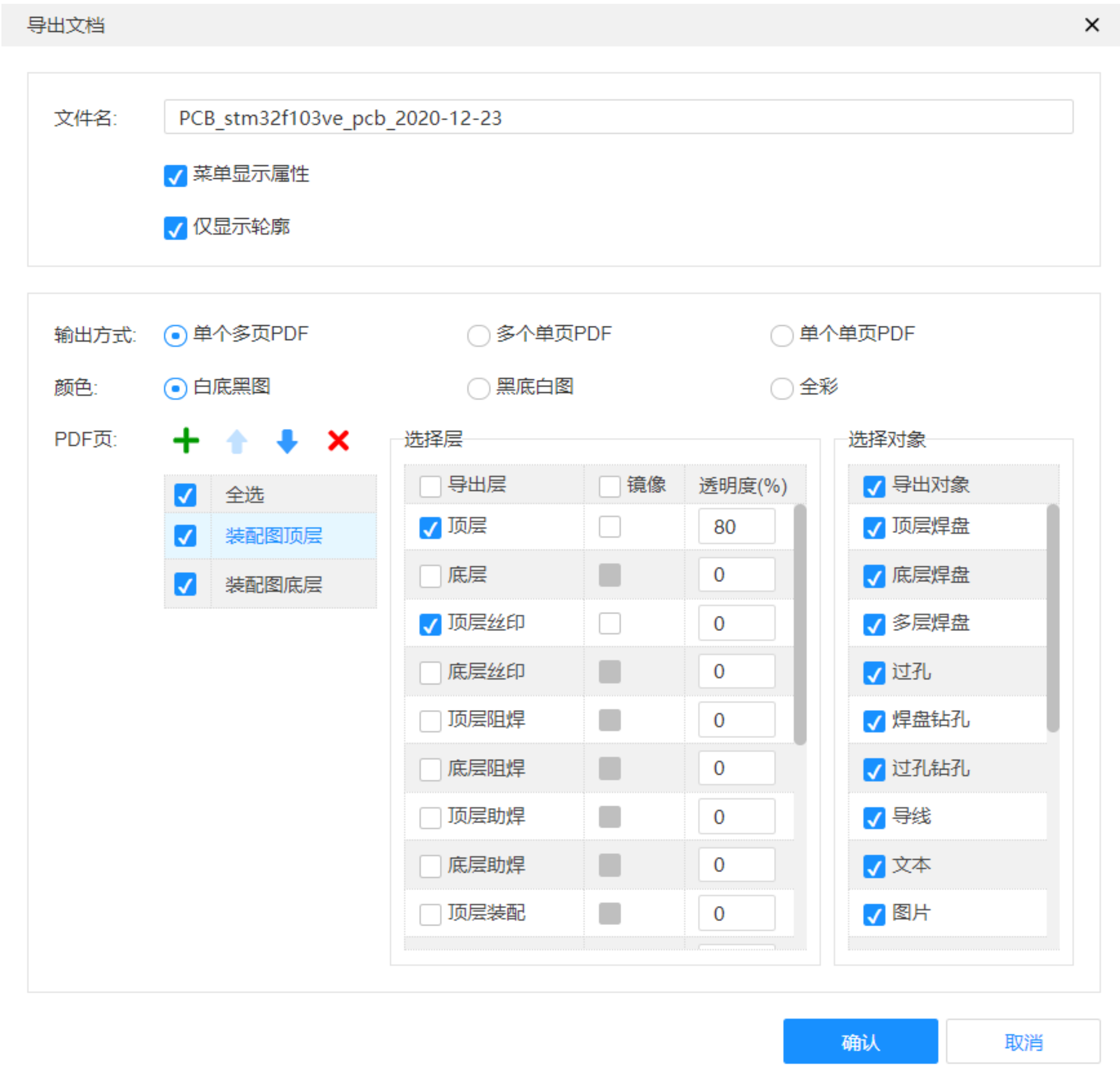
Export PDF/Image

The export PDF functionality for schematics and PCBs is different, but the operation is similar. It can support exporting layers and objects separately, and set export mirroring and transparency.

- **Top Menu - Export - PDF/Image**

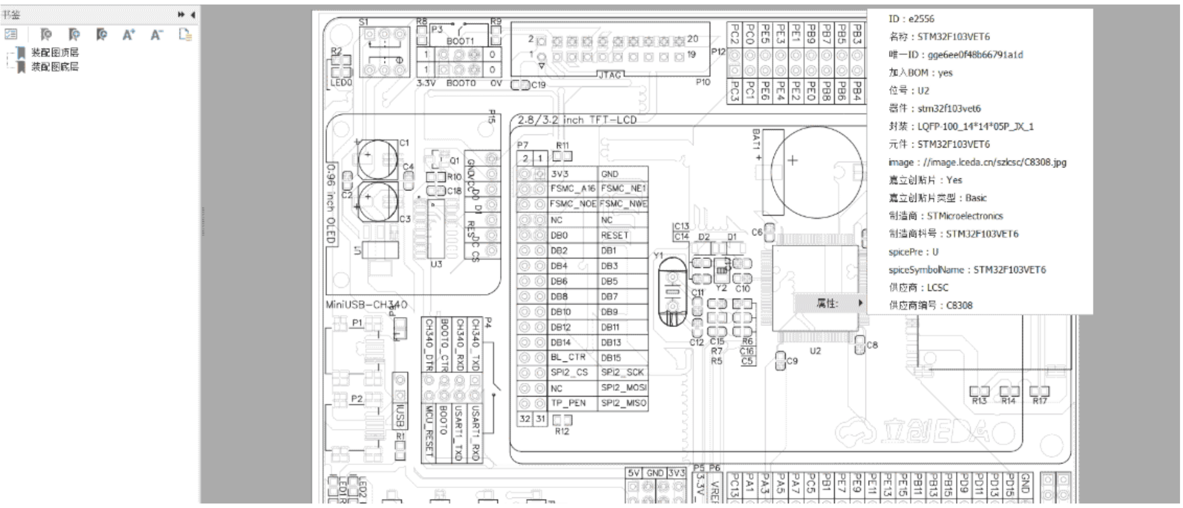
Popup Settings Popup

The top of the pop-up window is the file name of the exported PDF.



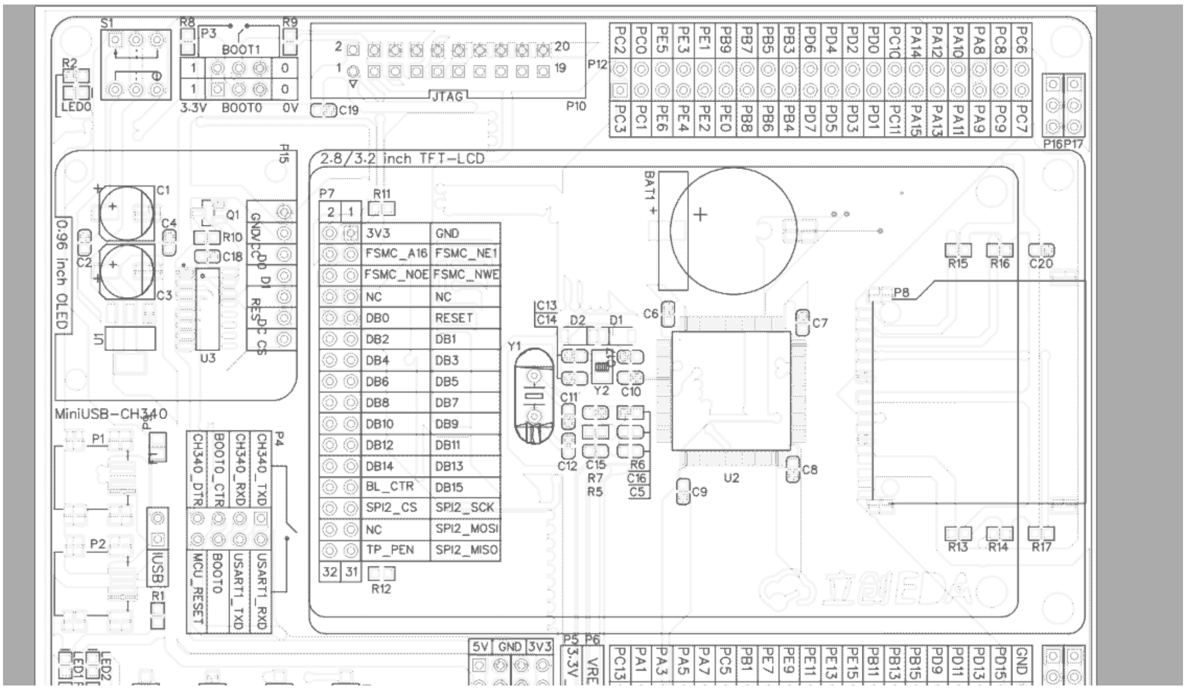
Menu Properties

After selection, the exported PDF is opened in a PDF reader, click Device Yes to view the properties of the device.



Show outline only

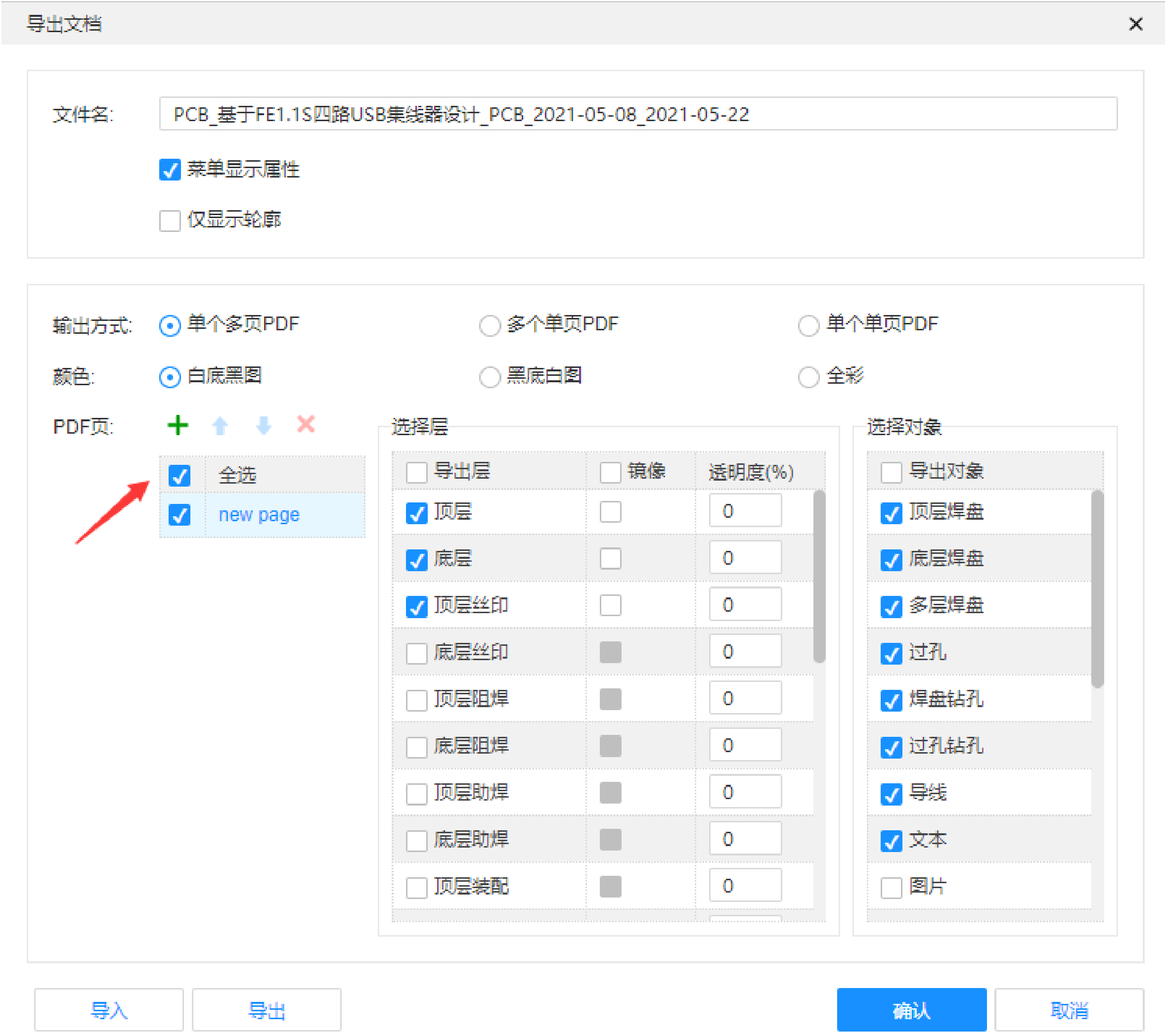
After selecting, the pads, wires, and outline primitives of the exported PDF will only display outlines.

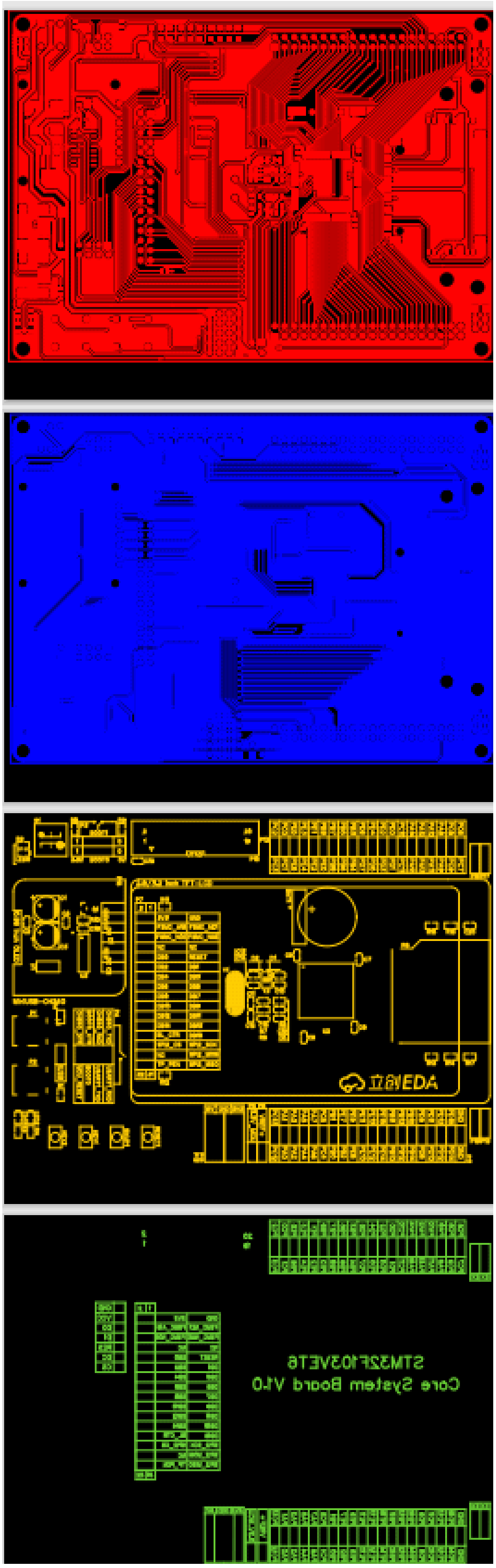


Single Multi-Page PDF: Export multiple file multi-page drawings.

And you can select the exported layers.

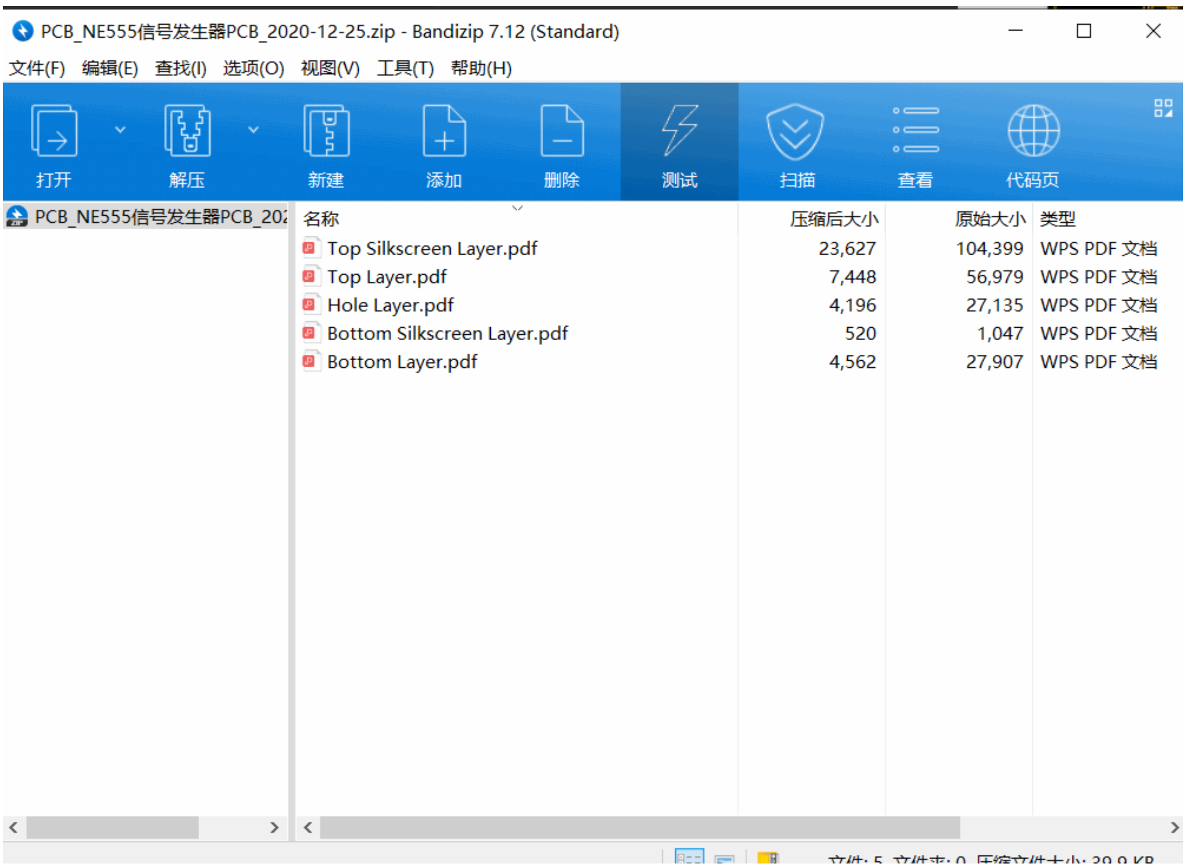
导出文档
文件名:
PCB_stm32f103ve_pcb_2020-12-23
菜单显示属性
仅显示轮廓
输出方式:
单个多页PDF
多个单页PDF
单个单页PDF
颜色:
白底黑图
黑底白图
全彩
PDF页:
全选
装配图顶层
装配图底层
选择层
导出层
镜像
透明度(%)
顶层
80
底层
0
顶层丝印
0
底层丝印
0
顶层阻焊
0
底层阻焊
0
顶层助焊
0
底层助焊
0
顶层装配
0
选择对象
导出对象
顶层焊盘
底层焊盘
多层焊盘
过孔
焊盘钻孔
过孔钻孔
导线
文本
图片
确认
取消


导出文档
文件名:
PCB_基于FE1.1S四路USB集线器设计_PCB_2021-05-08_2021-05-22
菜单显示属性
仅显示轮廓
输出方式:
单个多页PDF
多个单页PDF
单个单页PDF
颜色:
白底黑图
黑底白图
全彩
PDF页:
全选
new page
选择层
导出层
镜像
透明度(%)
顶层
底层
顶层丝印
底层丝印
顶层阻焊
底层阻焊
顶层助焊
底层助焊
顶层装配
0
选择对象
导出对象
顶层焊盘
底层焊盘
多层焊盘
过孔
焊盘钻孔
过孔钻孔
导线
文本
图片
导入
导出
确认
取消

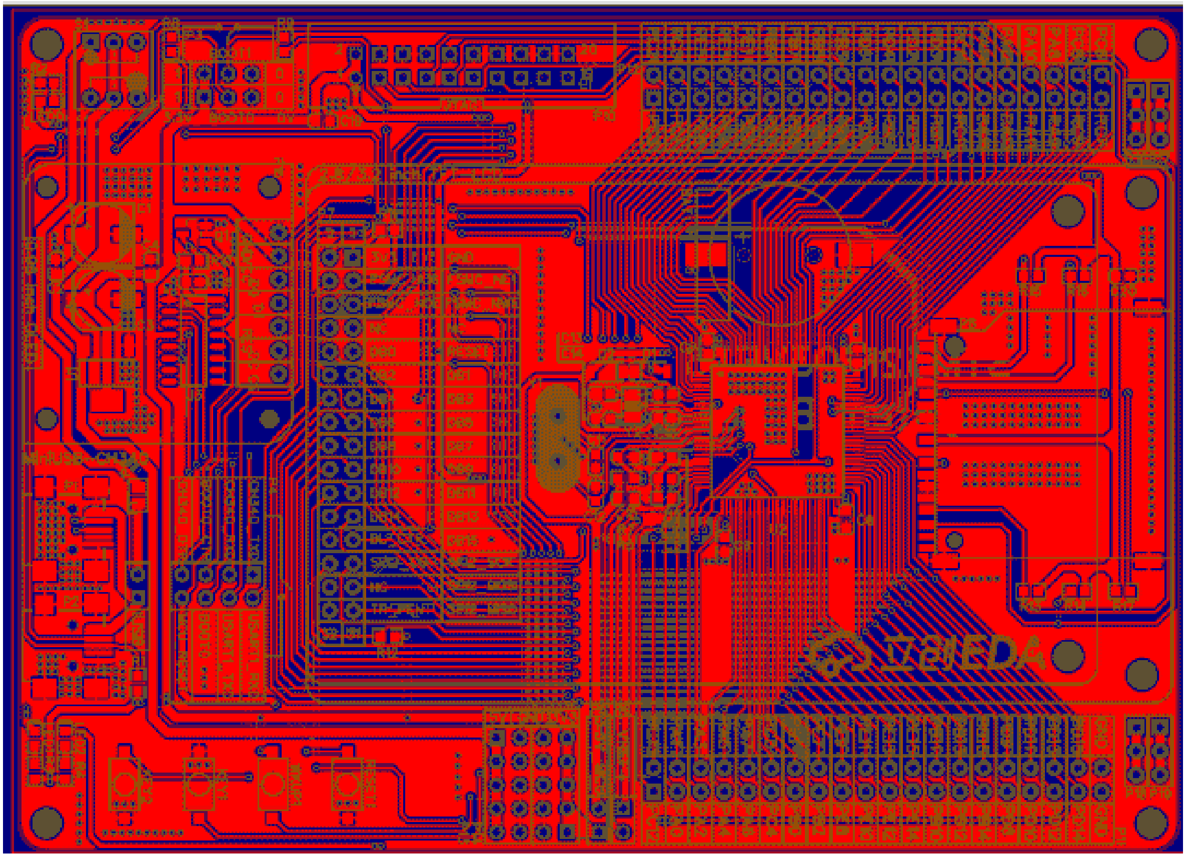


Multiple single-page PDFs: Export the layers of the PCB as a separate file and export it as a compressed footprint. You can select the layers to be exported, mirror images, and adjust the transparency, and select the exported objects.


导出文档
文件名: PCB_stm32f103ve_pcb_2020-12-23
菜单显示属性
仅显示轮廓
输出方式: 单个多页PDF 多个单页PDF 单个单页PDF
颜色: 白底黑图 黑底白图 全彩
选择层
导出层 镜像 透明度(%)
顶层 0
底层 0
顶层丝印 0
底层丝印 0
顶层阻焊 0
底层阻焊 0
顶层助焊 0
底层助焊 0
顶层装配 0
选择对象
导出对象
顶层焊盘
底层焊盘
多层焊盘
过孔
焊盘钻孔
过孔钻孔
导线
文本
图片
确认
取消


PCB_NE555信号发生器PCB_2020-12-25.zip - Bandizip 7.12 (Standard)
文件(F) 编辑(E) 查找(I) 选项(O) 视图(V) 工具(T) 帮助(H)
打开 解压 新建 添加 删除 测试 扫描 查看 代码页
PCB_NE555信号发生器PCB_202
名称 压缩后大小 原始大小 类型
Top Silkscreen Layer.pdf 23,627 104,399 WPS PDF 文档
Top Layer.pdf 7,448 56,979 WPS PDF 文档
Hole Layer.pdf 4,196 27,135 WPS PDF 文档
Bottom Silkscreen Layer.pdf 520 1,047 WPS PDF 文档
Bottom Layer.pdf 4,562 27,907 WPS PDF 文档

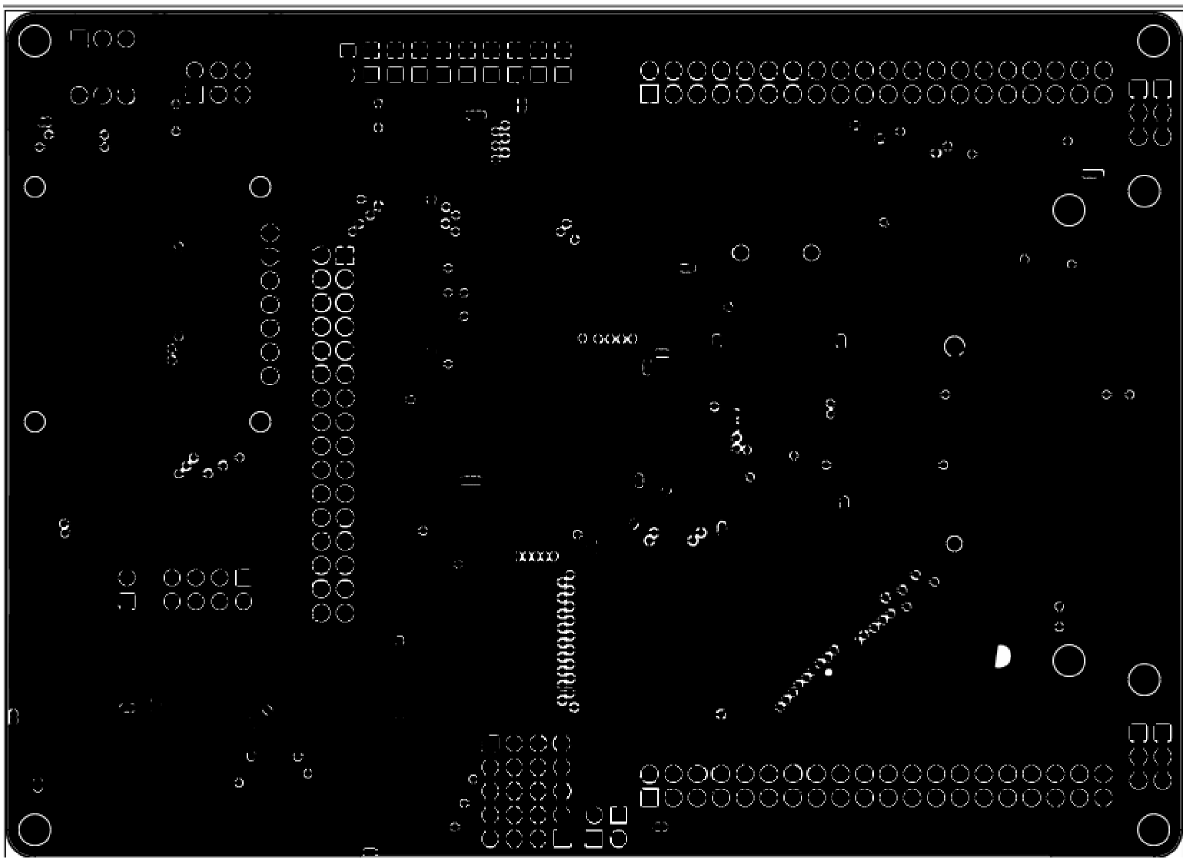
Single single-page PDF: Export all layers as a drawing page, you can also choose the exported layers, transparency adjustment, and exported object selection.



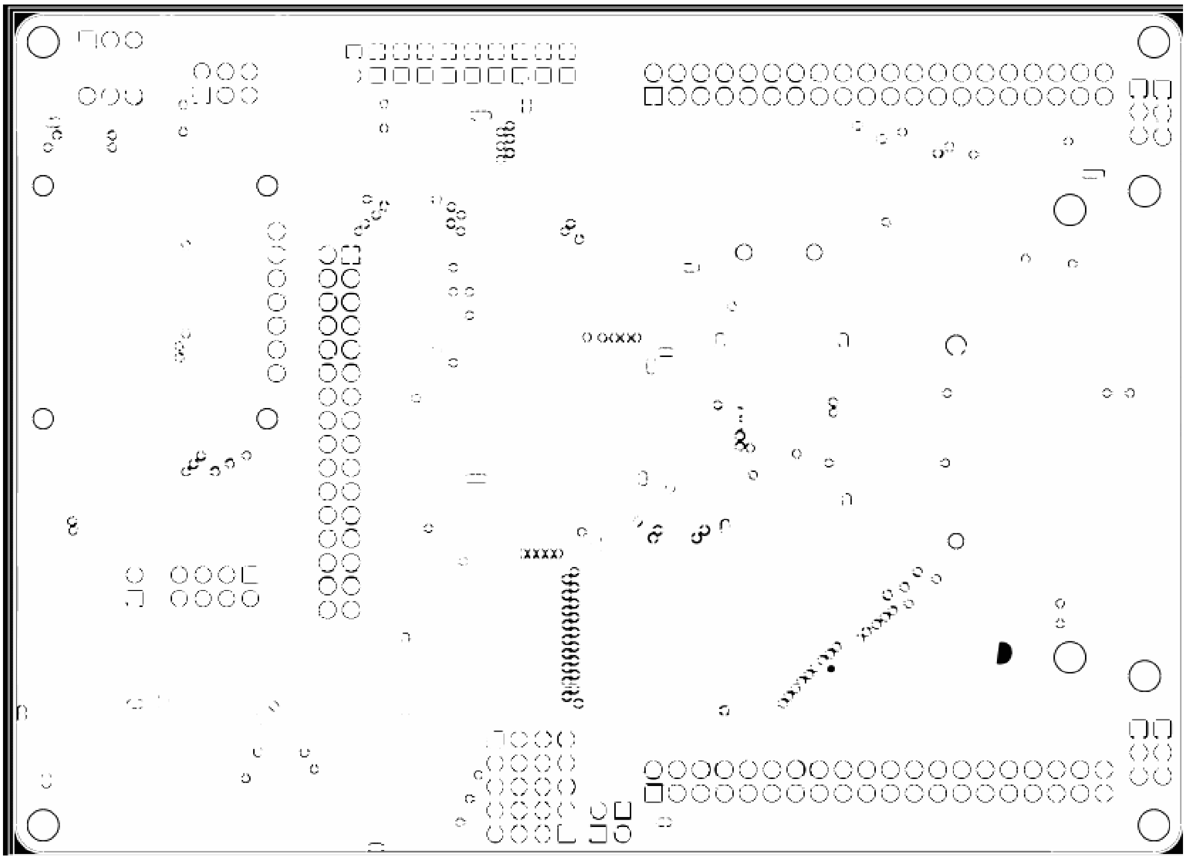
color

The color of the exported PDF image can be selected.

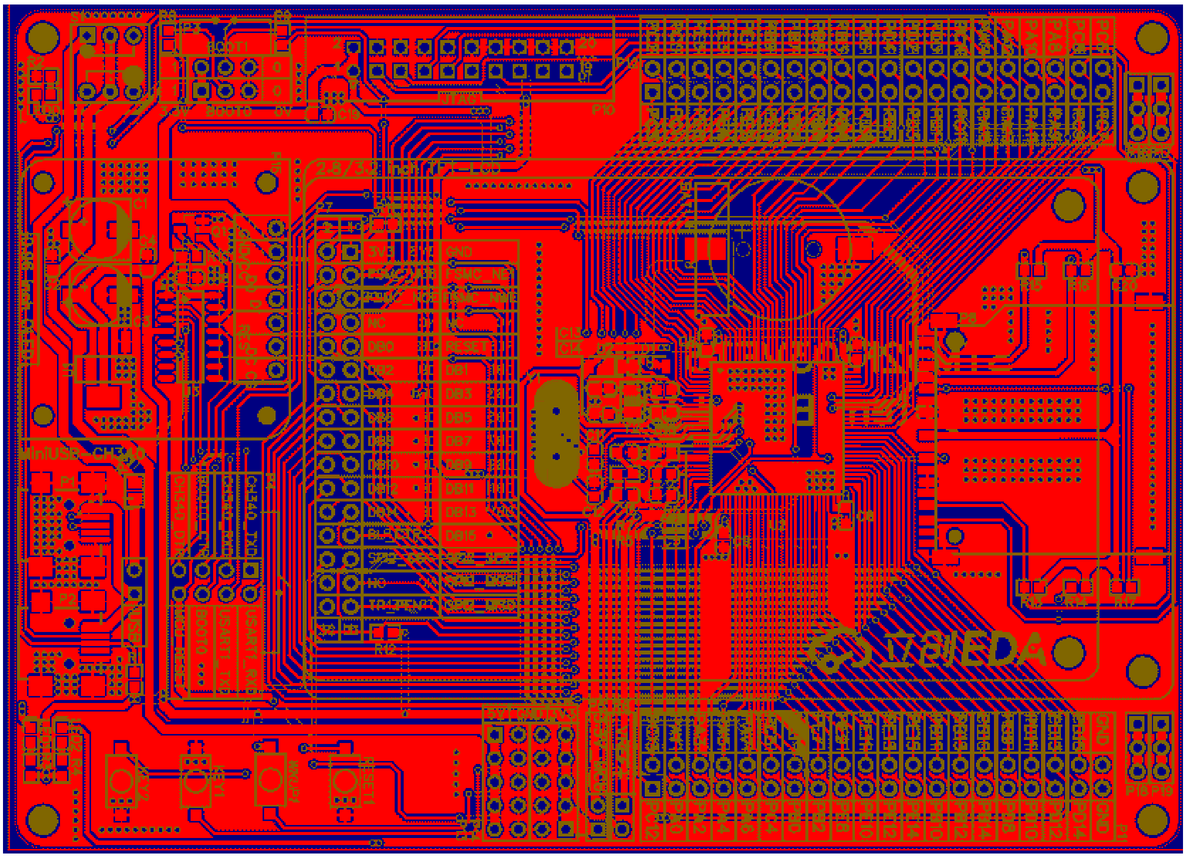
Black image on white background



White on black background



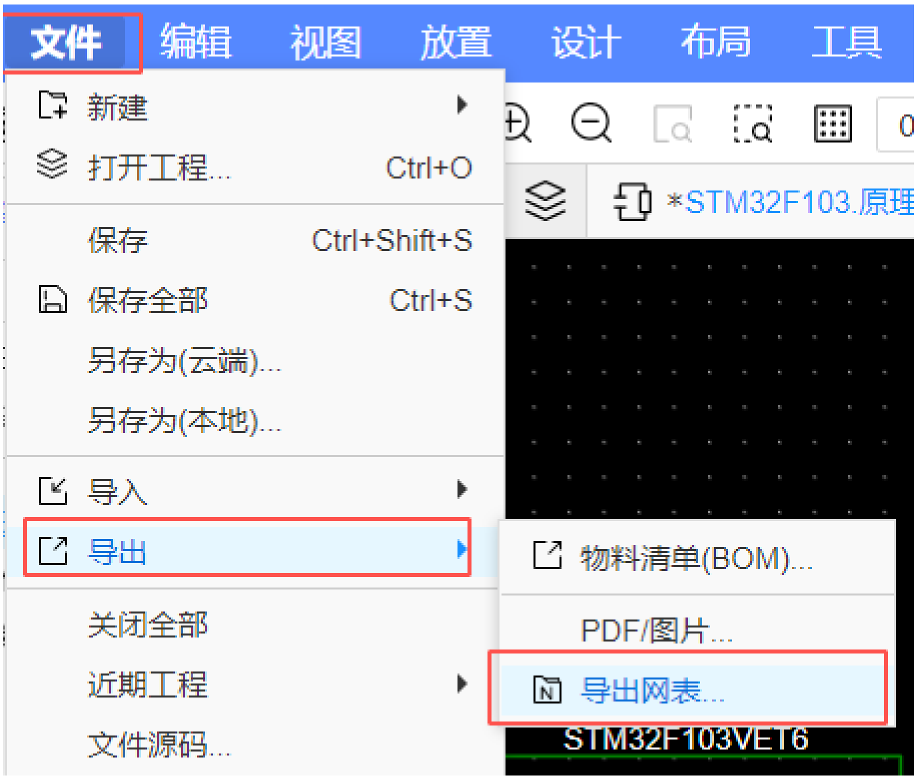
Full Color



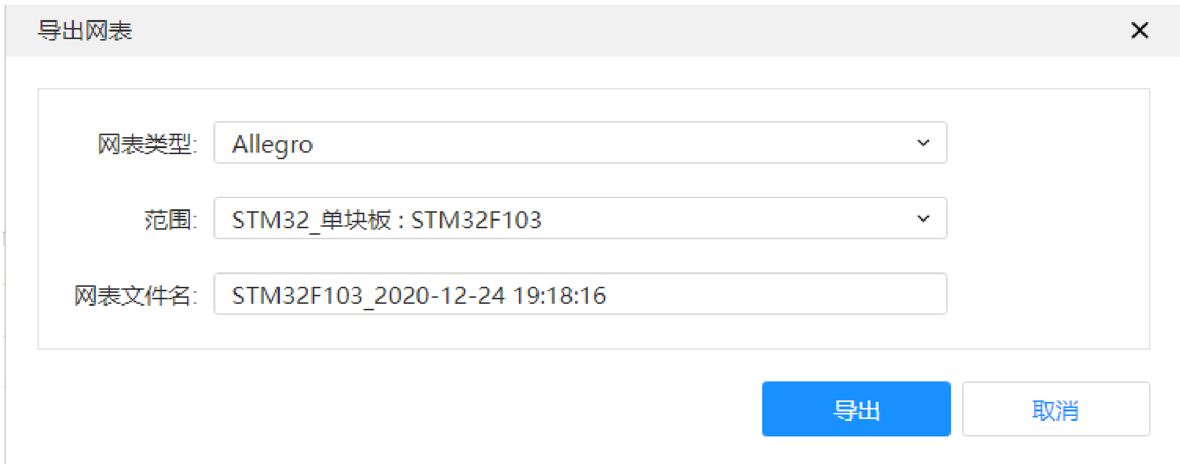
Export Netlist

EasyEDA Pro currently only supports exporting the netlist file formats of Allegro and EasyEDA Pro. You can export a single document netlist or a netlist for the entire project.

- **Top Menu - File - Export - Export Netlist**



The pop-up window allows you to modify the export netlist type and the page range for which the netlist needs to be exported.



Order Parts

Order Parts

1. Design the schematic diagram and PCB in EasyEDA Pro.
2. Open the schematic diagram, click "**Top Menu - Export - Bill of Material (BOM)**" or "**Top Menu - File - Bill of Material (BOM)**" to export the BOM table.
3. Click on the top **menu bar - Export - Order Parts** or the far right side of the top toolbar **Order Parts** to open LCSC Mall, upload the exported BOM to purchase components.

Note: *If you need to patch in JLCPCB, the SMD components need to be purchased when JLCPCB SMT order is placed. The purchase of components in LCSC Mall and JLCPCB is independent, including inventory and price. *

PCB order

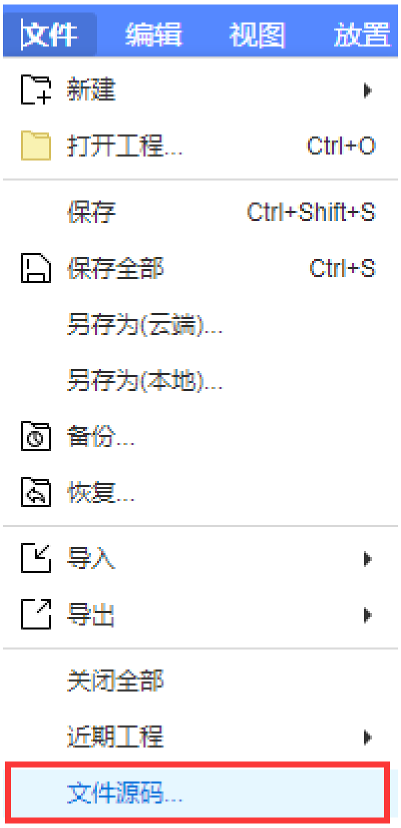
1. Open the PCB, click the "**Top Menu - Export - Generate PCB Fabrication File**" menu to generate Gerber.

2. Click **Top Menu - Export - PCB Order**, open the Jiali Chuang order interface, upload and export to generate the manufacturing file Gerber.
3. Go to JLCPCB https://jlcpcb.com, you can log in directly with the EasyEDA Pro account, no need to register again.

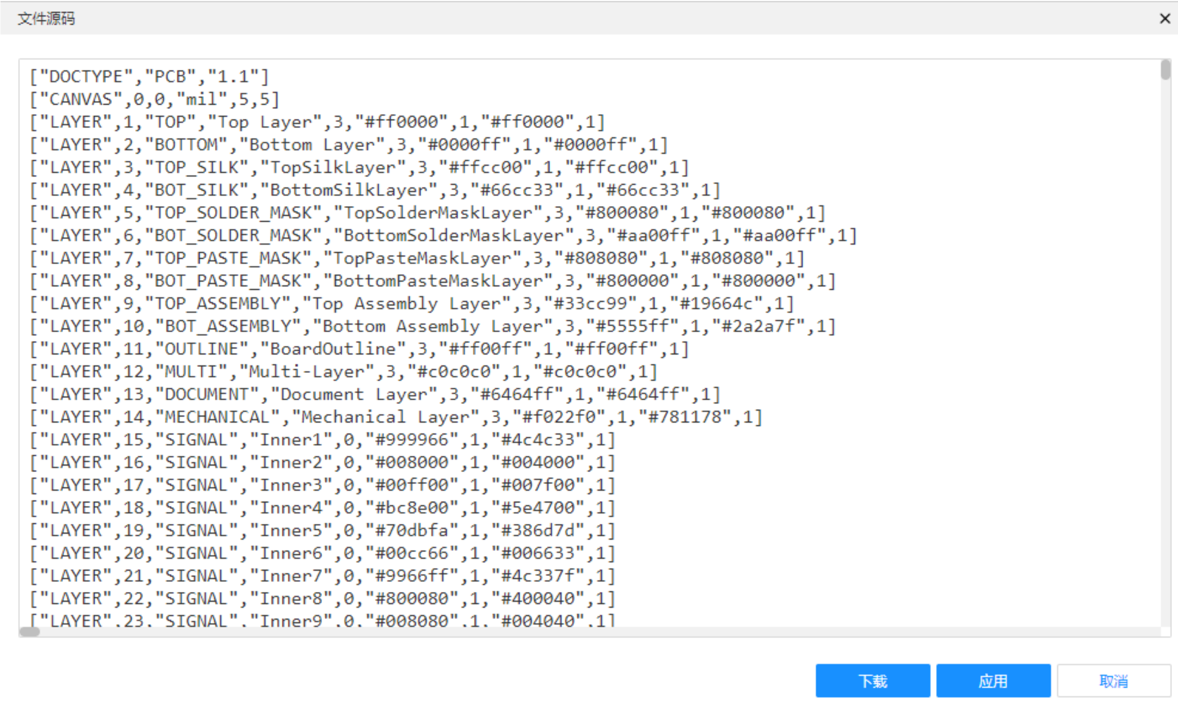
File Source

EasyEDA Pro supports exporting or editing the source code of files.

- Top Menu - File Source



After clicking Download, you can download the file source code of the project, and then you can open the file through a text editor or browser. You can also copy the code in the dialog to edit it in the editor.

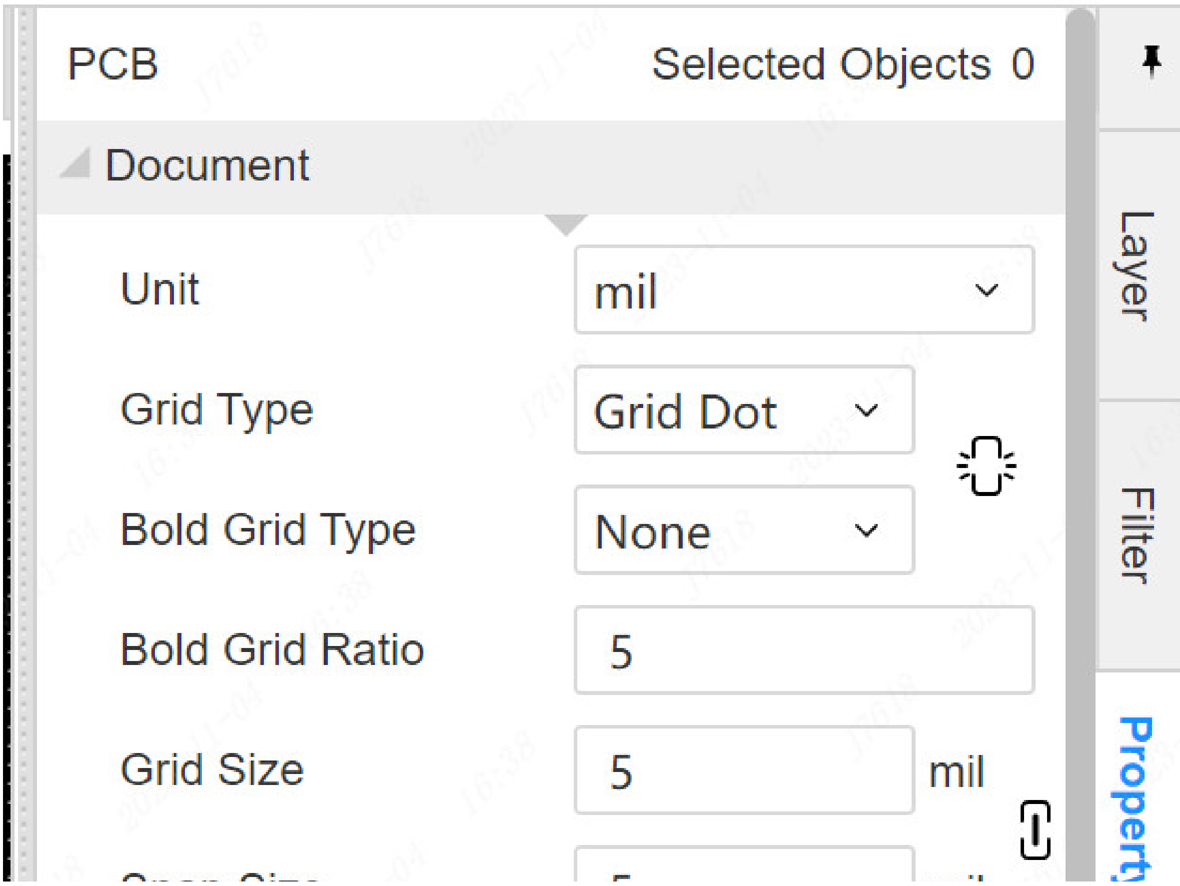


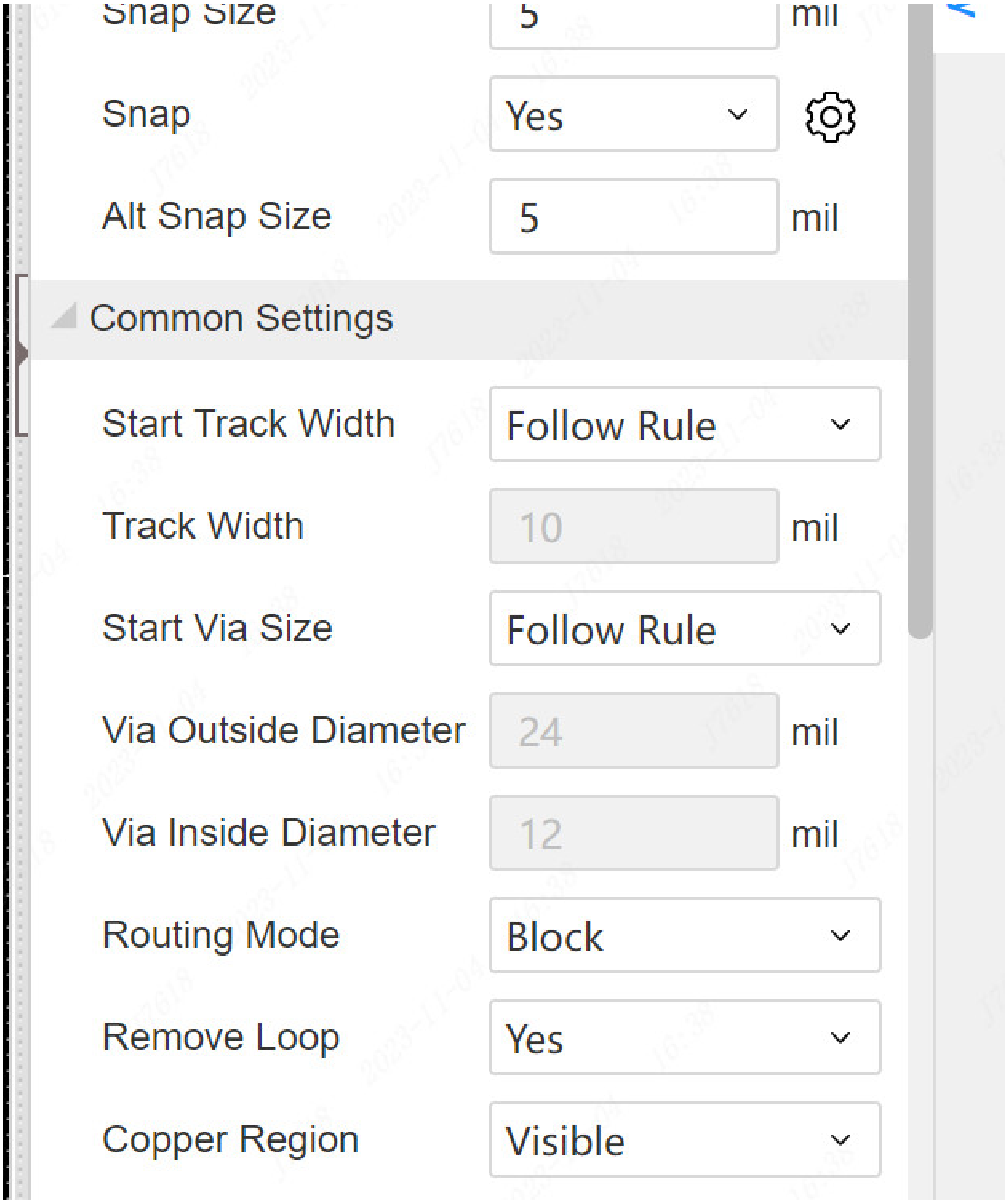
PCB Design

PCB Setting

Canvas property

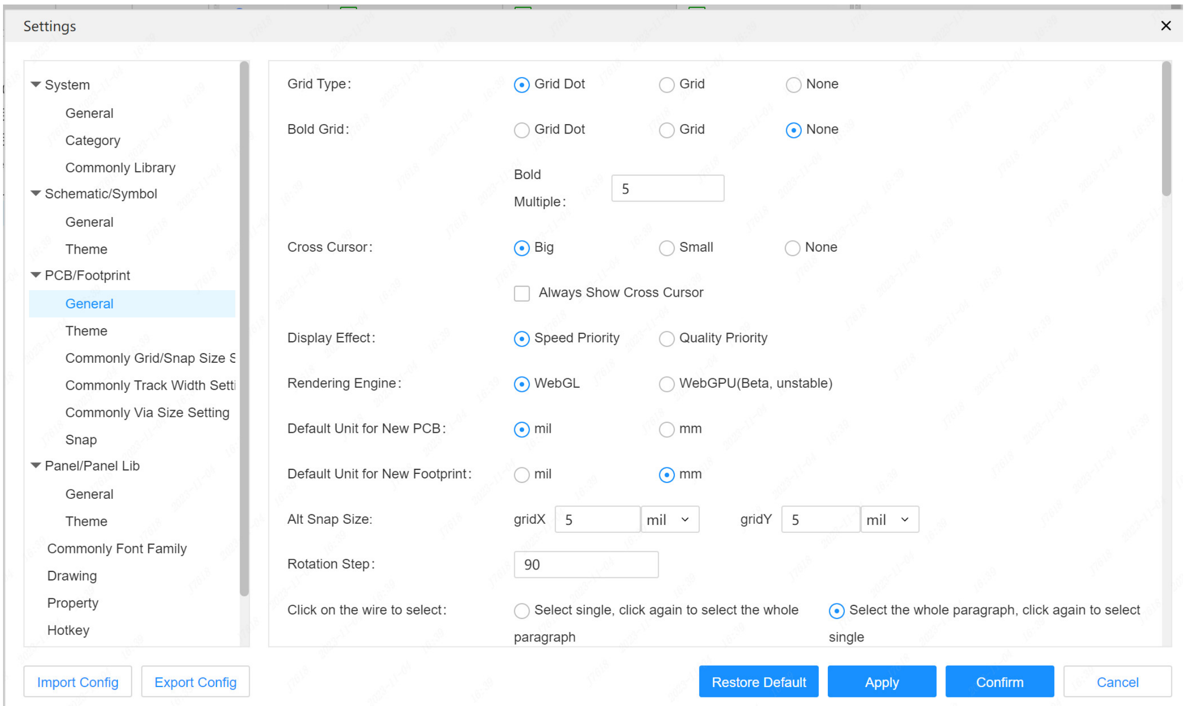
After opening the PCB, you can set the commonly used settings on the right attribute panel. These settings are set on the top menu, right -click menu, and filtering TAB can be modified.The setting parameters can be saved into the PCB. The next time you open the PCB, the parameters of the settings are given priority.





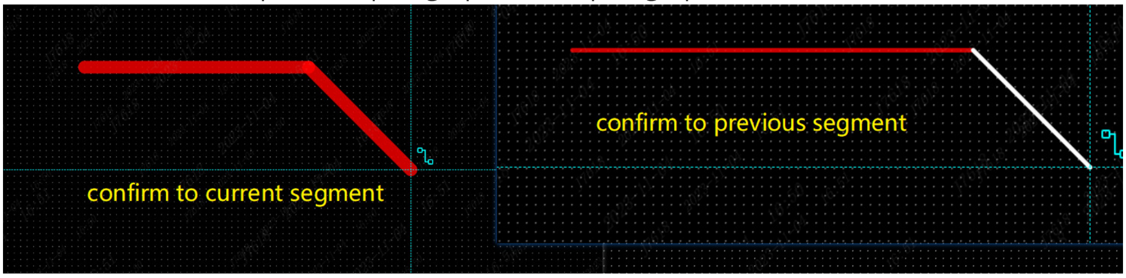
PCB preference settings

Many of the operation -related settings in PCB can basically set up -PCB -general settings to view and modify its attributes.

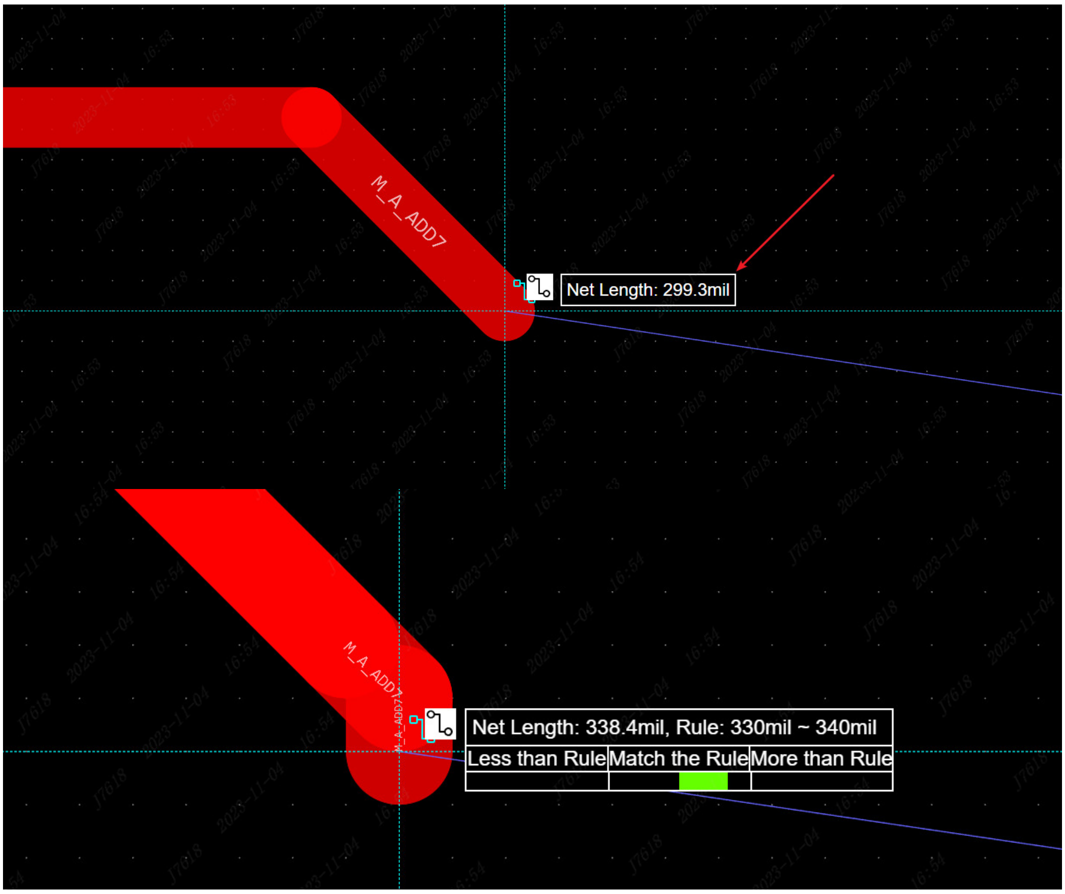


- **Grid Type**: Grid Dot,Grid,none
- **Bold Grid**: Grid Dot,Grid,none.After the bold mesh setting, the display effect will be thicker than the ordinary grid. You can adjust the color and transparency in the PCB theme settings
- **Bold Multiple**: For the ordinary grid size of each thick multiplayer, the bold grid is displayed
- **Cross Cursor**: Change the size of the cursor in the PCB editor.
- **Always Show Cross Cursor**: After the selection, the mouse will always be the shape of the cross cursor.
- **Display Effect**: The priority will improve the performance of the editor, but the quality decreases.The quality priority is to open anti -aliasing. After opening, the display will be relatively smooth and have certain demand for computer configuration.
- **Rendering Engine**: Webgl, webgpu.WebGPU is currently a test version, and the performance is unstable.
- **Default Unit for New PCB**: Amend the new PCB unit here.
- **Default Unit for New Footprint**: Modify the newly -built unit here.
- **Alt Snap Size**: ALT adsorption size (applying this parameter according to ALT adsorption during drawing)
- **Rotation Step**: The angle setting of the device rotation.
- **Click on the wire to select**: Divided into selecting a single segment, click the entire section and select the entire section, and click again to select the single segment.Here will affect the selection range of the first selected wire and contour object, and click again to switch to another range of selection.
- **Copy/Cut**: Select the reference point when copying the object.

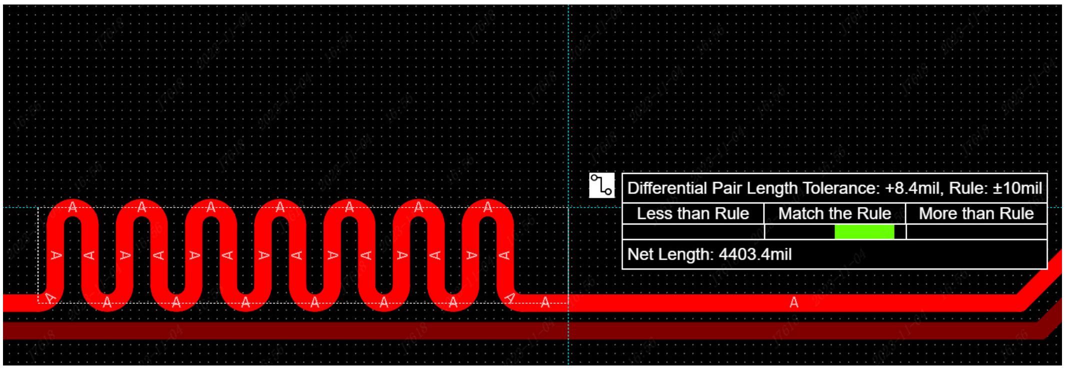
- **Confirm track segment while routing**: Each time you add a wire segment when the wiring The quantity is confirmed to add multiple paragraphs at one time in the current paragraph, and confirm that the previous paragraph adds a paragraph



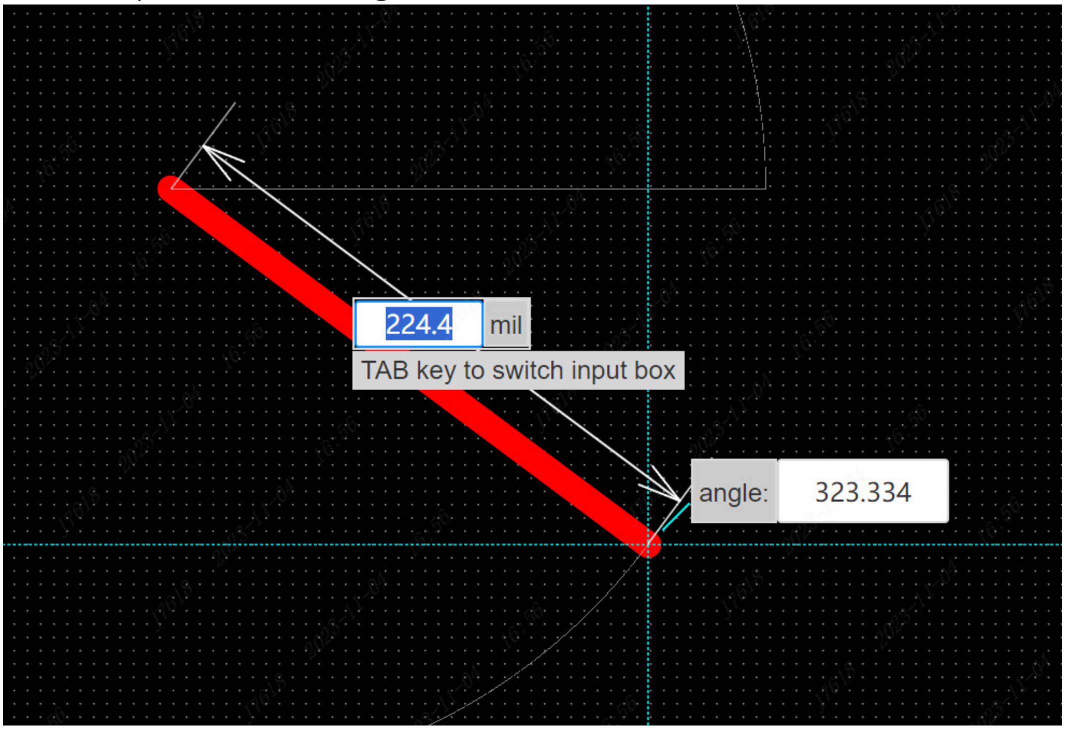
- **End primitives drawing**: To set the draw mode, right -click is to end the drawing or the right -click menu. Ctrl+right -click will be the opposite behavior.
- **Add via method while route**: Add at the previous turning point, and fixed the position of the perfunction at the previous wiring turning point position; add the current turning point, follow the mouse to move, and confirm the location after adding a new inflection point.
- **Rotation Objects**: Rotate the single object. When multiple objects are selected for rotation, each object is rotated according to the respective center; the overall rotation is selected.
- **Start Track Width for New Document**: Set whether the width of the wire when the new PCB is set up follows the rules.
- **Default font type for component properties**: After setting the schematic diagram to the PCB, the component's attributes such as the default font type and high attribute information displayed in PCB.
- **Start Via Size for New Document**: Whether the perforated size follows the rules when setting a new PCB.
- **Show Number or Net**
    - ***Show Pad Number***: Show the number of the pad of the device.
    - ***Show Pad Net Name***: Display and hide the network of pads of the device.
    - ***Show Track Net Name***: Hide and display the network on the wire.
    - ***Show Via Net Name***: Display and hide the network of holes.
- **Real Time Display**
    - ***Display Real-Time Net Length While Route or Stretch Track***: After turning on, when the wiring, drag, stretching wire, and long adjustment, the network length information is displayed in real time. If the network sets the network length rules, it will also show the network length range



- ***Display Real-Time Differential Pair Length Tolerance While Route or Stretch Track***: After turning on, when the differential pair, drag, stretch the wire, and adjust the long adjustment of the different time, display the differential pairs of length error information.



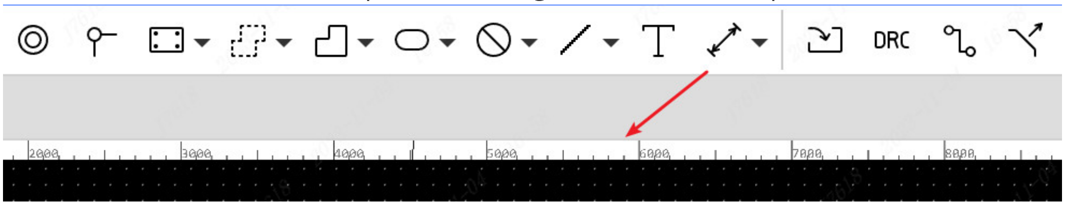
- ***Draw Using Dynamic Input Box***: Draw a folding line or polygon in the current free angle of the line and the angle of the arc free angle, the rectangular, and the round mode. The dynamic input box will appear in this function. The directly input parameter is used to lock the parameter according to the TAB or ENTER.



- **Mouse Hover**
  - ***Highlight the Net When Hovering Over the Net***: When the mouse hovers on a certain wire, pad, and pores, the wires, pads, and pores of the entire network are highlighted.
  - ***Highlight Object While Cursor Hover Object***: When the mouse hovers at a certain object, Gao Liang is high. At this time, holding the object in the mouse selection will be the same as the object of Gao Liang.
  - ***Display Object Property While Cursor Hover Object***: When the mouse hovers at a certain object, some attribute information of the object is displayed.

- **Route**
  - ***Prefer Place Blind/Burried Via while Change Layer on Routing***: When the pores are placed in the wiring switching layer, set the perforated or blinded holes for priority.
  - ***Auto Change Layer While Routing Vias***: After turning on, add holes during the wiring process and automatically switch to other layers at the same time.
  - ***Automatic fan-out when end of routing***: After turning on, when the chip pin wiring is turned on, it is given a distance from the pad.
  - ***Terminate Routing Automatically***: After clicking the other map of the same network, the wiring is ended, that is, the wiring is over.
  - ***Assign Net for Free Track***: When drawing a network -free wire, it will automatically allocate a new network for the free wire and the connected pads after ending the wiring.
  - ***Add Teardrop Automatically***: Draw a wire, drag a wire, pad, and a hole, and automatically add tears.
- **Drag/Move**

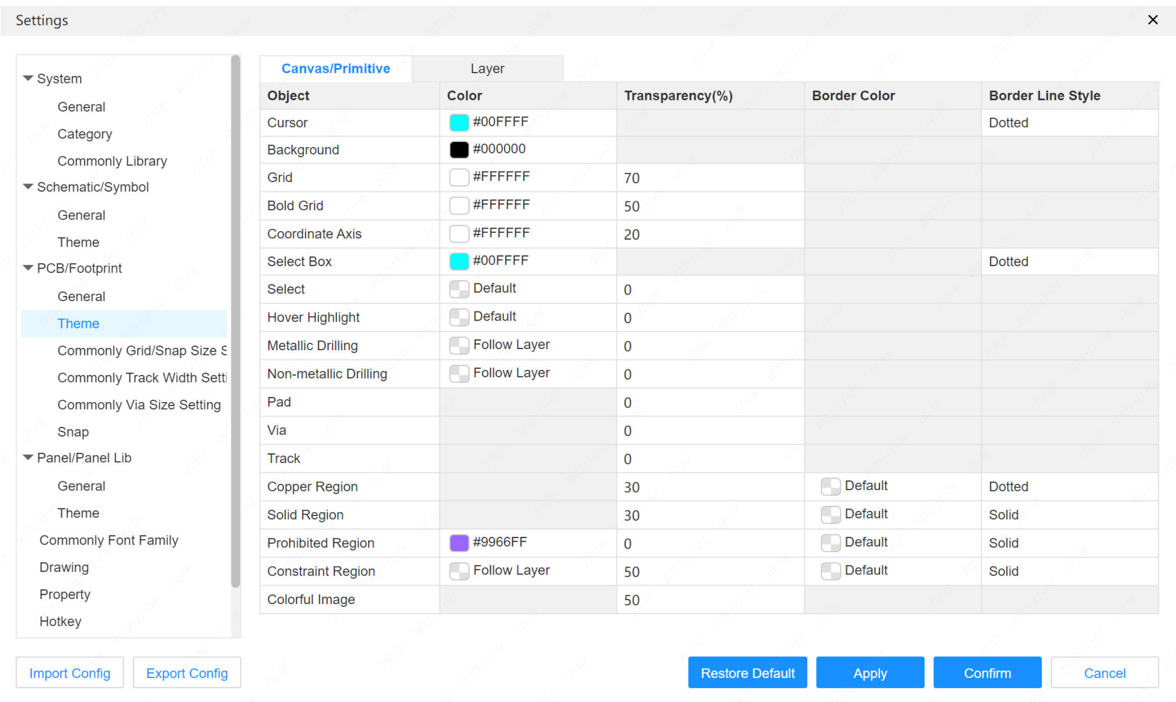
- ◦ ***Ctrl drag to copy***: Press and hold the ctrl key to drag the object, you can copy the new object。
  - ***Cursor snaps to center while dragging pads/vias***: Drag the pads/perforated, and the cursor automatically adsorb to the center and follow the mouse.
  - ***Cursor snaps to origin while dragging footprints***: Drag the package, automatically adsorb the cursor to the heart and follow the mouse.
  - ***Align in adjacent footprint/pads when mobile footprint/pads***: Drag the packaging/pad, if there are other pads and packaging next to it, it will automatically adsorb to the position of the alignment boundary when moving to its boundary.
- **Move the footprint, wire's following way**: When moving the device, the wire follow -up method is set.
- **Move the Via, wire's following way**: When moving the holes, the wire follow -up method is set.
- **Other**
  - ***Show Scale Ruler***: Close and open the scaling ruler below the top shortcut bar.

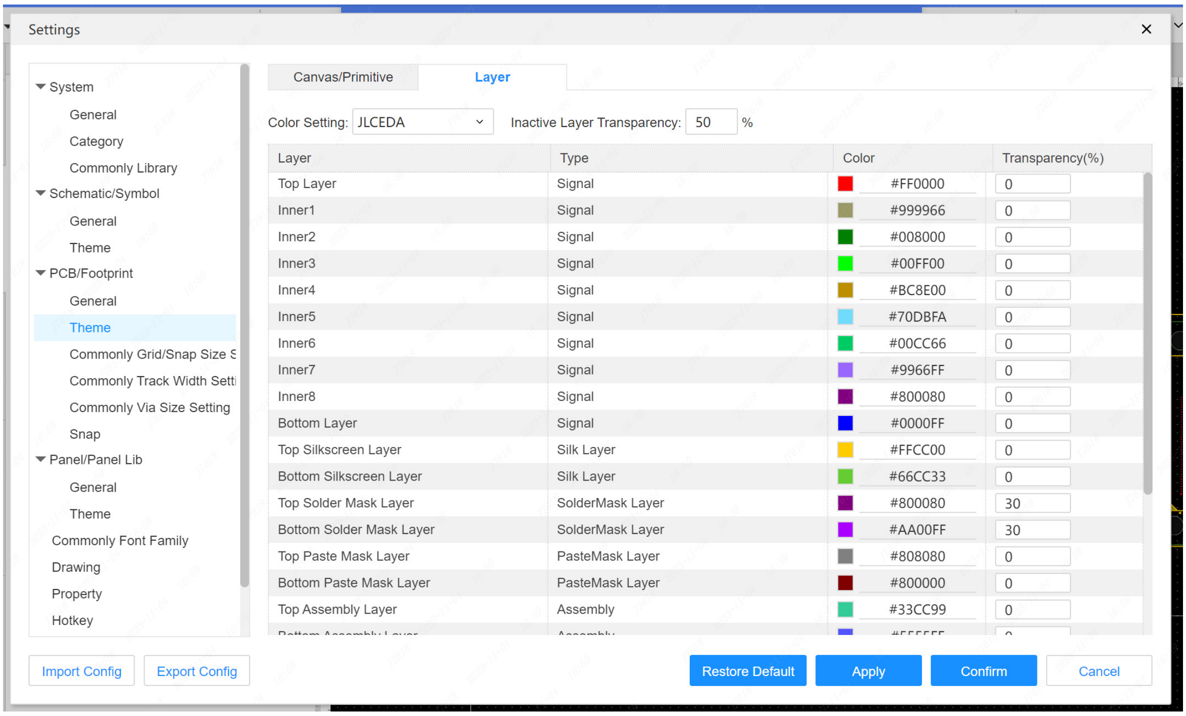


  - ***Check Net Connections Before Generating the Fabrication File***: When exporting the Gerber file, check whether the connection of the PCB network is completed.
  - ***Check DRC Before Generating the Fabrication File***: When exporting the Gerber file, check whether the DRC prompt is checked.
  - ***Reserve Designators when Pasting***: Stick the copy of the device to PCB, keep the original number.
  - ***Using JLC color silkscreen technology***: After enabling, you can use colored silk print -related functions.

Theme settings

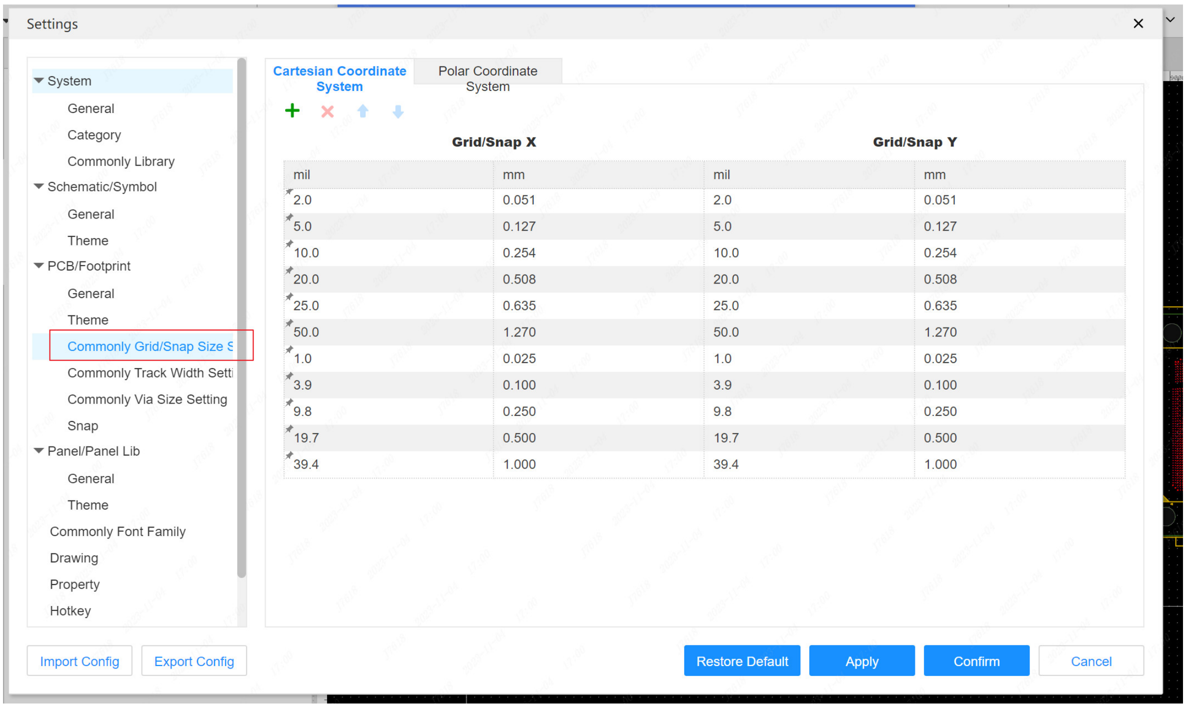
PCB interface theme settings, users can set some color configurations of the PCB interface according to their personal preferences, or they can choose the theme of other PCB software.





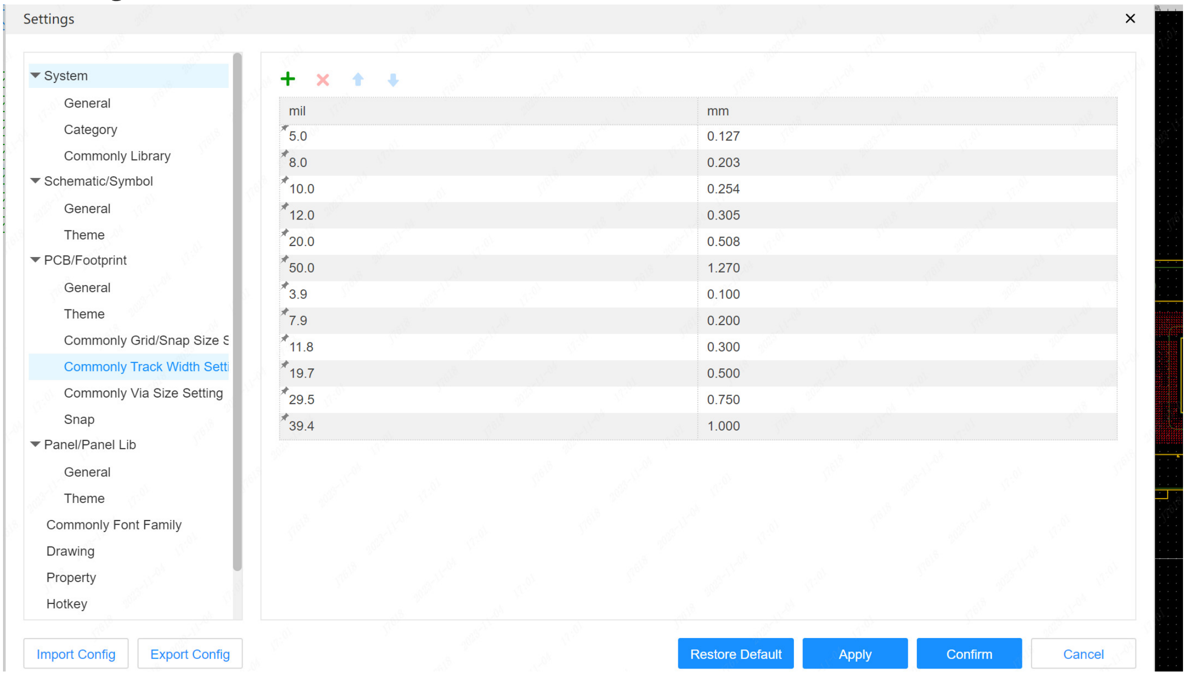
Commonly grid/snap size settings

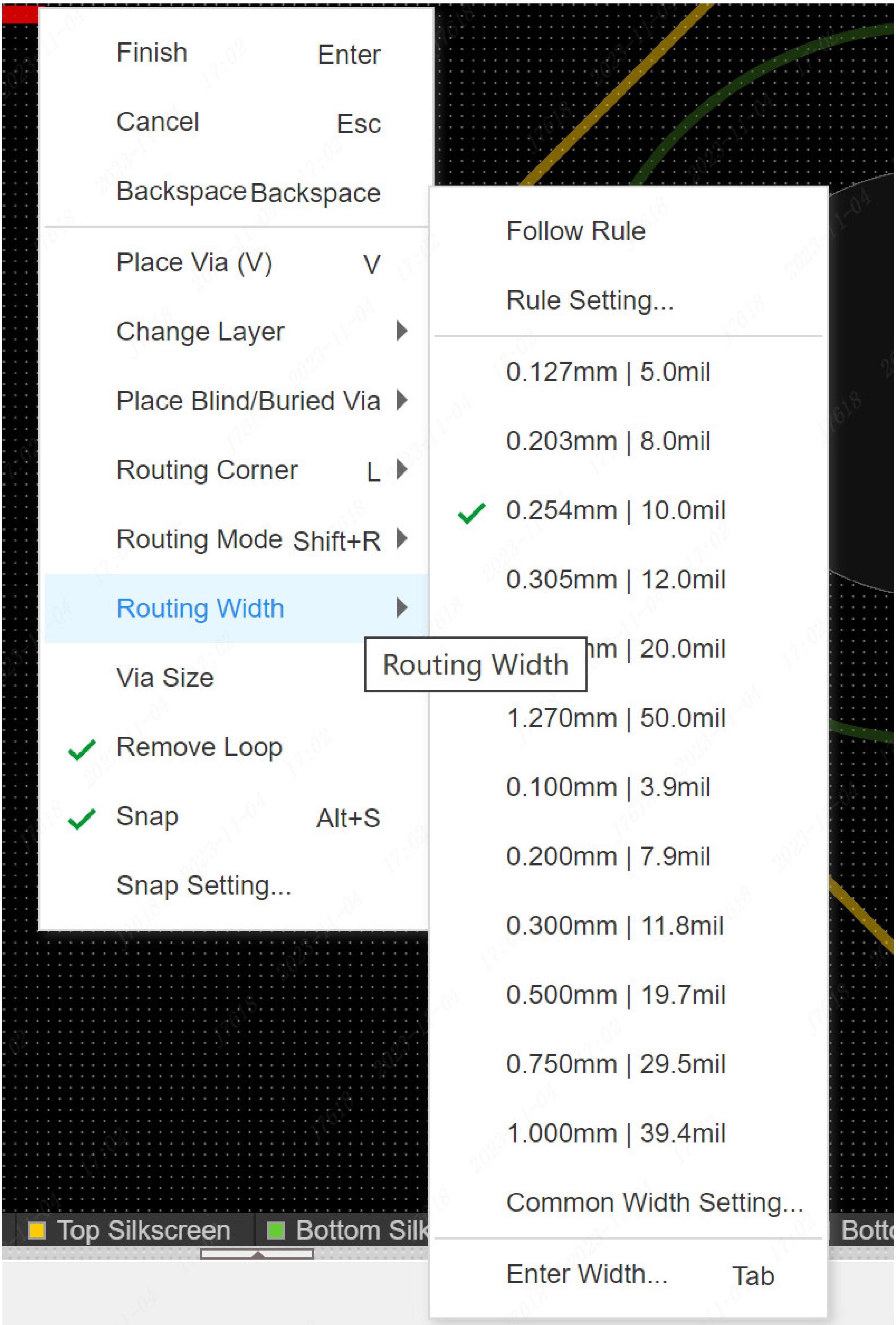
Set the size of the common right -angle coordinate system and polar coordinate grid, which is convenient for switching the size of the grid display when wiring.



Common track width settings

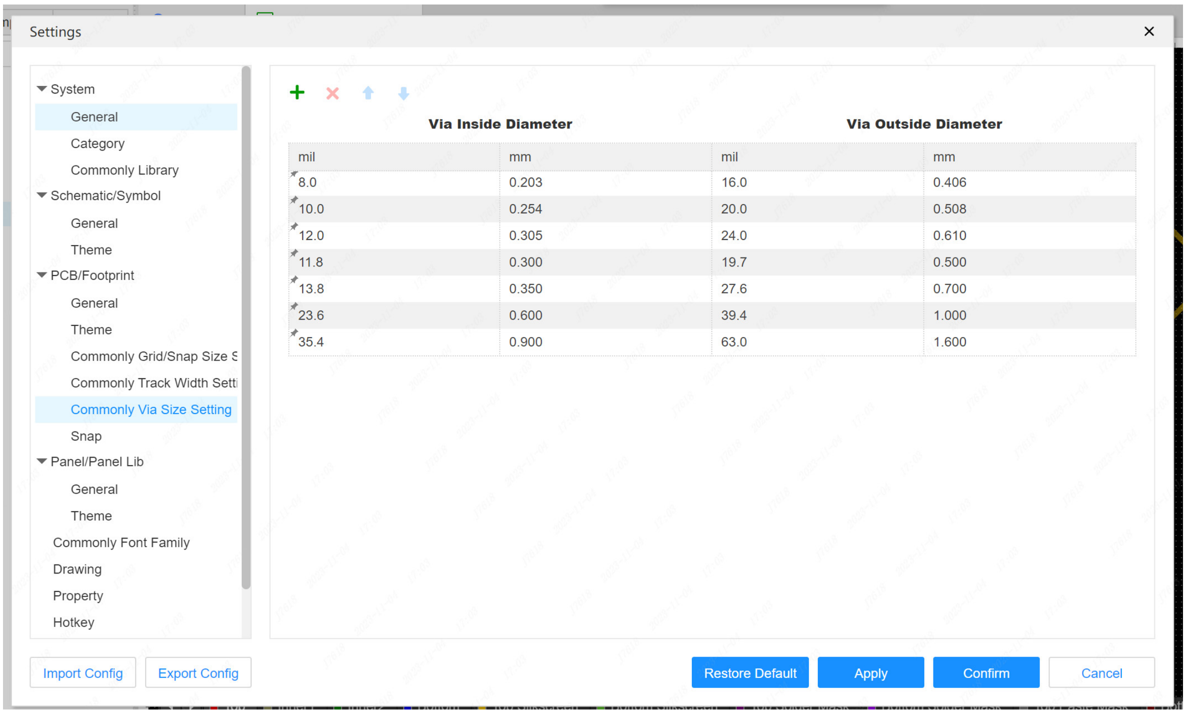
Set the common wire width of the wire, and you can quickly switch the width of the wire when the wiring.





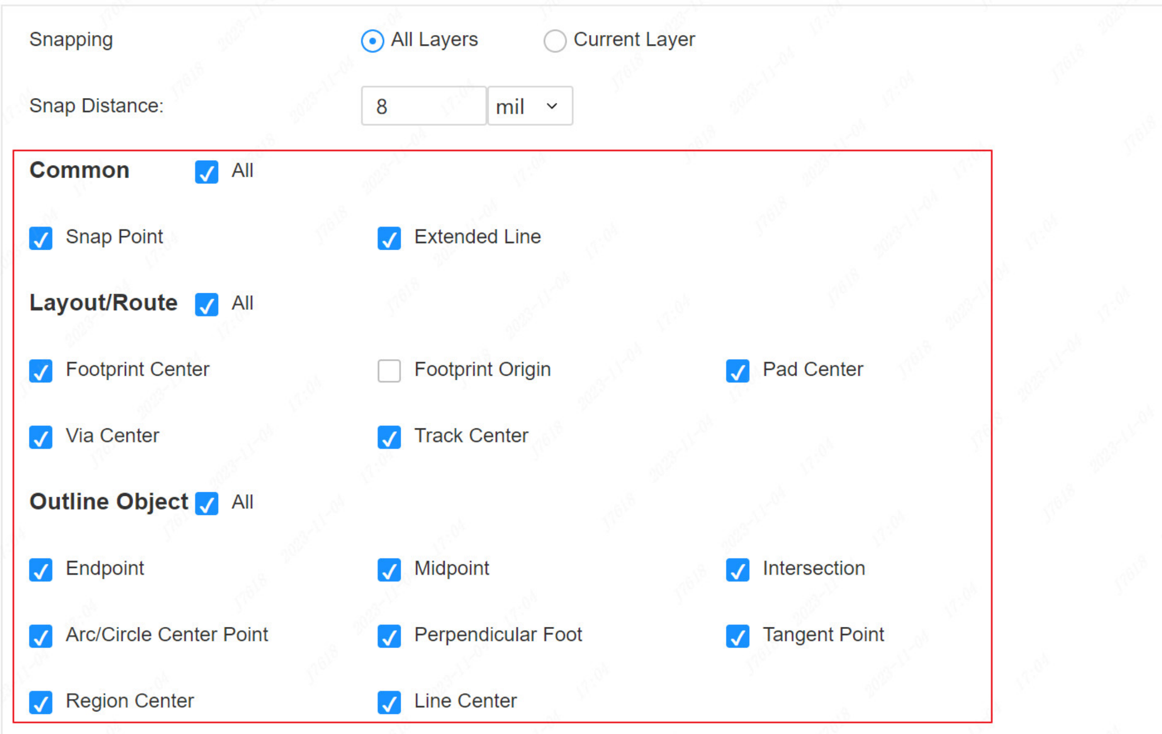
Common via size settings

Set the commonly used perforated size, which can be quickly switched to the hole when placing the hole.



Snap

- **Snapping*: You can set the adsorption only to take effect on the current activation layer, or take effect on all layers.
- **Snap Distance**: When the cursor moves to the adsorption point adsorption distance, it is automatically adsorbed to the adsorption point.
- **Adsorption item settings**: You can check the adsorption object you need here



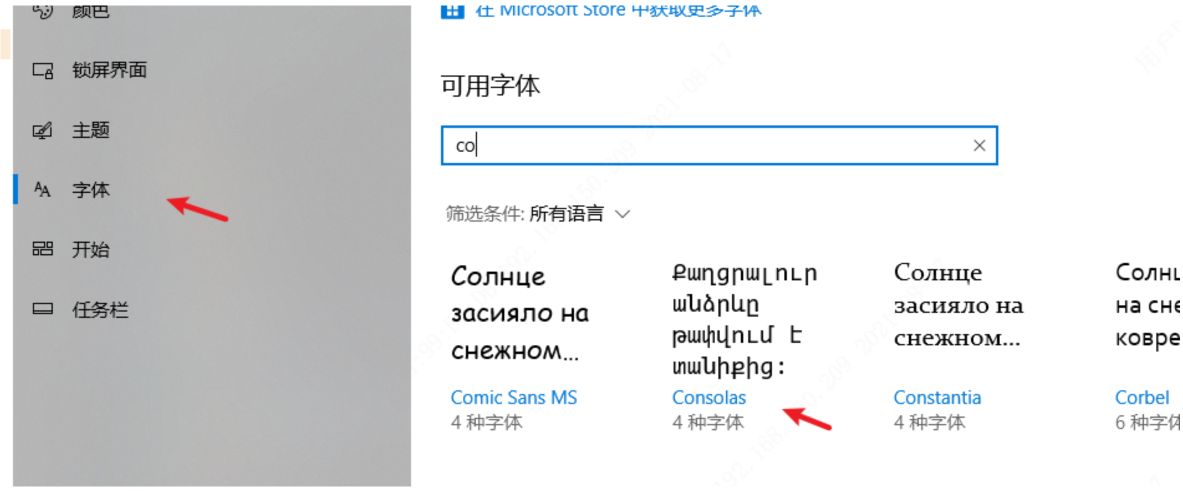
Add commonly used fonts

Support setting your own commonly used fonts.These fonts need to be installed on the local computer before being called by the editor, otherwise the editor will automatically use the default font provided by the browser for rendering.

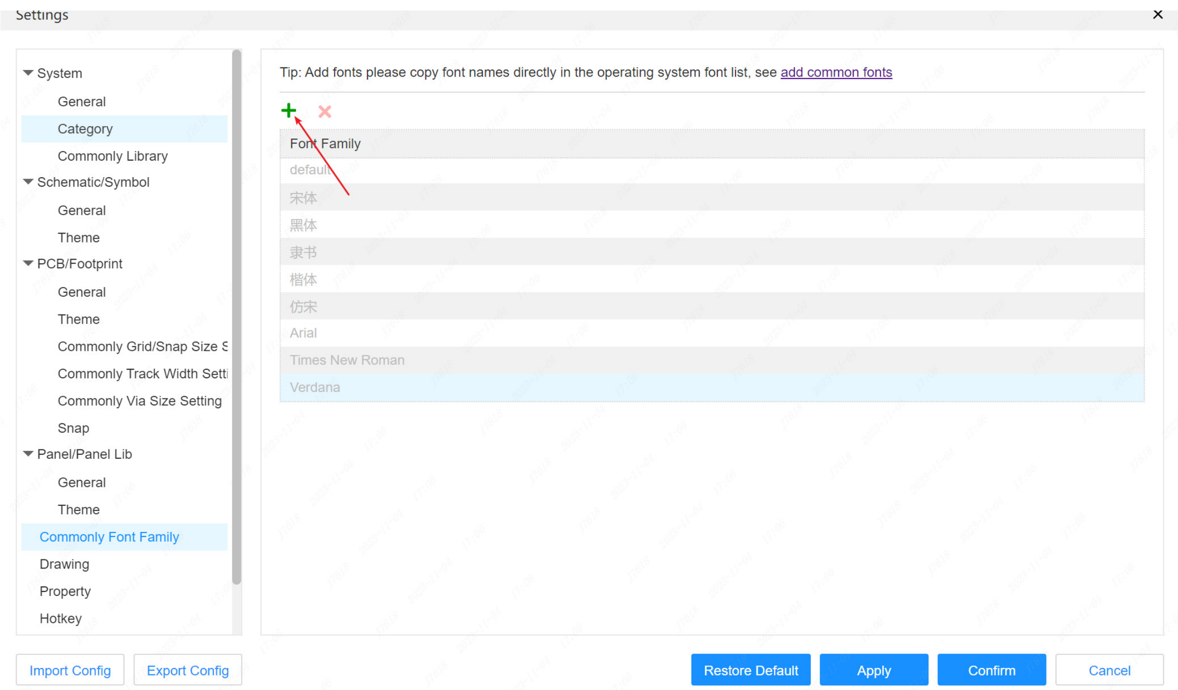
Add custom font:

1) First install the fonts you need in the local computer. If you already have fonts, you can ignore this step.

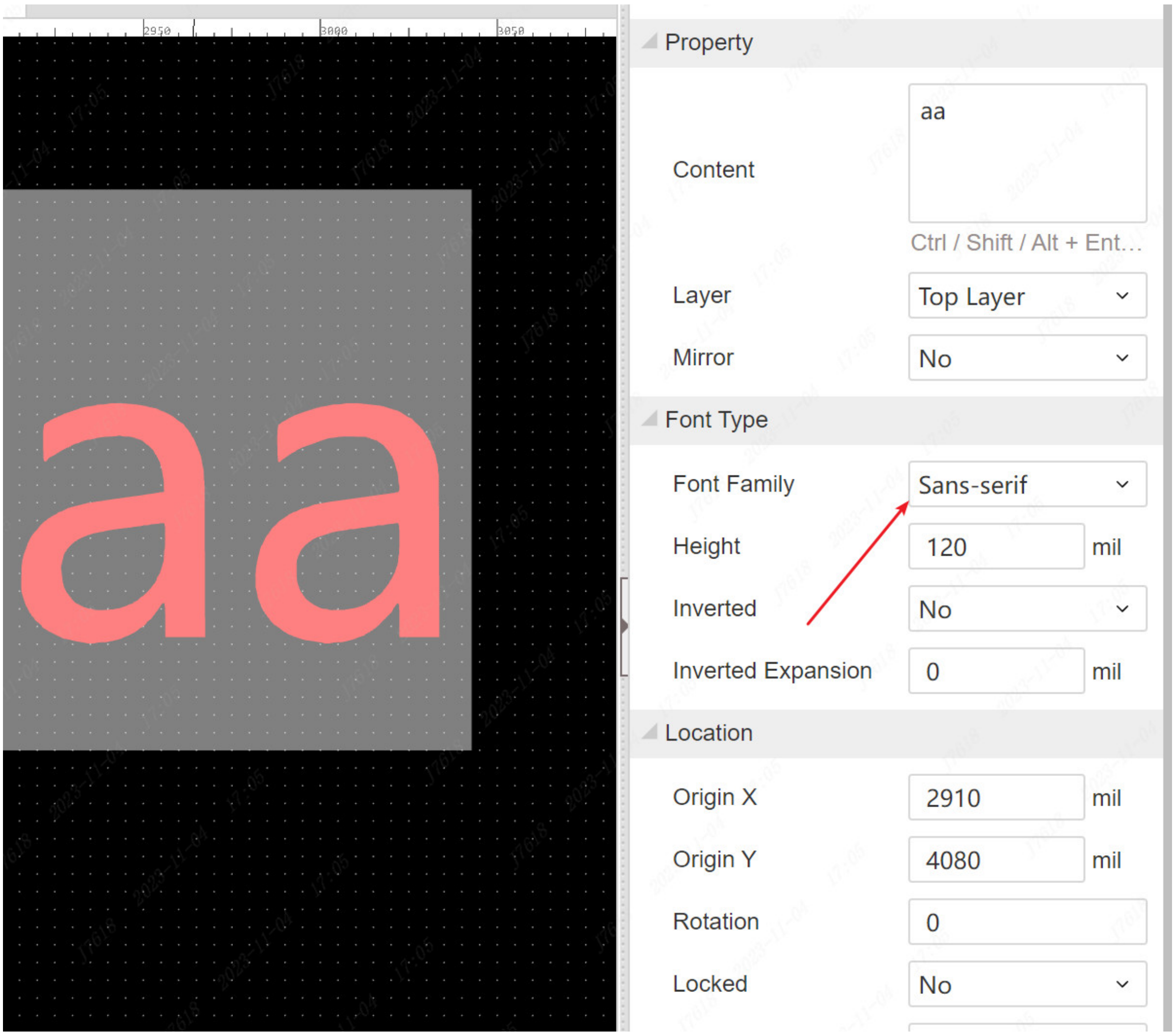
2) Taking the Windows system as an example, find the font settings in the system settings and get the font name.You need to get the accurate display name, which cannot be the file name of the font file.



3) Add a font according to the font name.



4) This font can be seen in the text font switching of the schematic or PCB. At this time, the font switching can be performed.



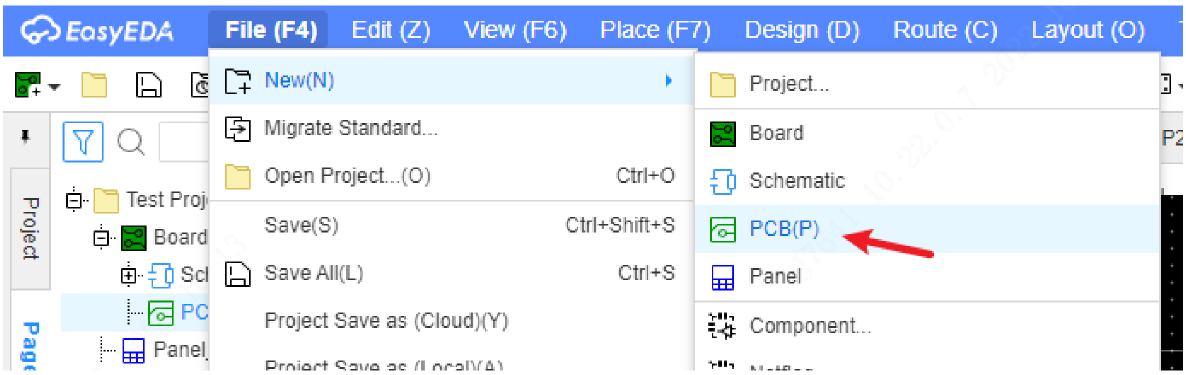
The fonts that are added can be used by schematic diagrams, panels, PCB, etc.

Create PCB

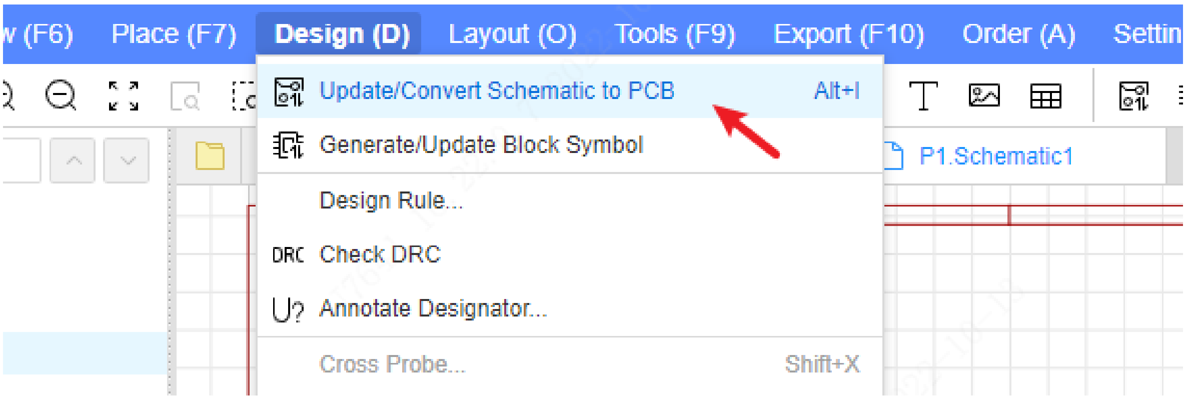
There are two ways to quickly create a PCB page in EasyEDA Pro to create a PCB interface.

After the project is created, a schematic diagram, drawing page, board file and PCB will be created by default in the project.

- Top Toolbar - Click - File - New - PCB



- Design in the schematic - update/convert to schematic to PCB to quickly generate PCB design interface board



EasyEDA Pro supports **PCB design without schematics**.

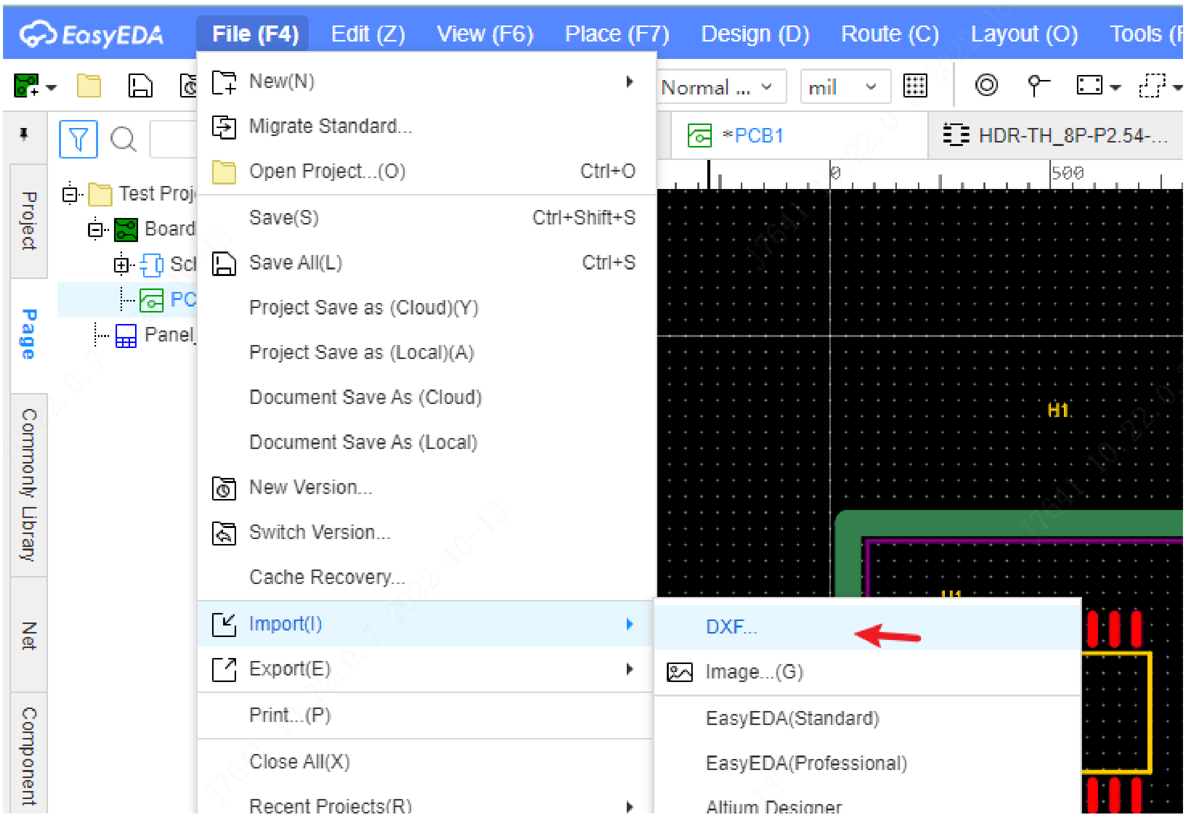
After creating a new PCB document, search and place the footprint you need in **"Component Library"** in the lower navigation panel. Custom properties can be added to each footprint to facilitate BOM export.

Import DXF

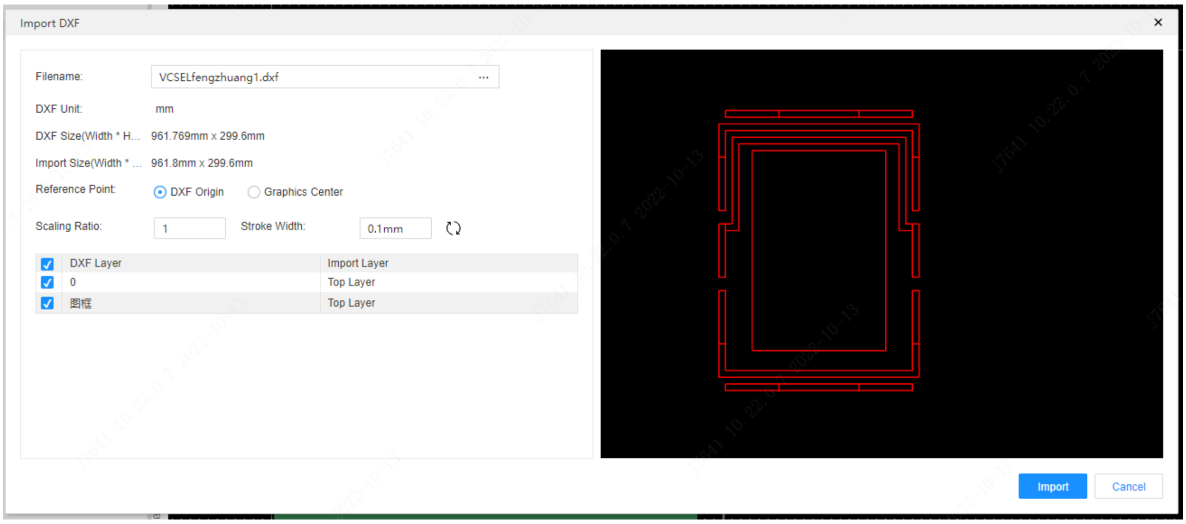
EasyEDA Pro supports importing DXF files as the border of the board, and can also be used to import DXF for other purposes.

Steps:

- Top Menu - File - Import - Import DXF

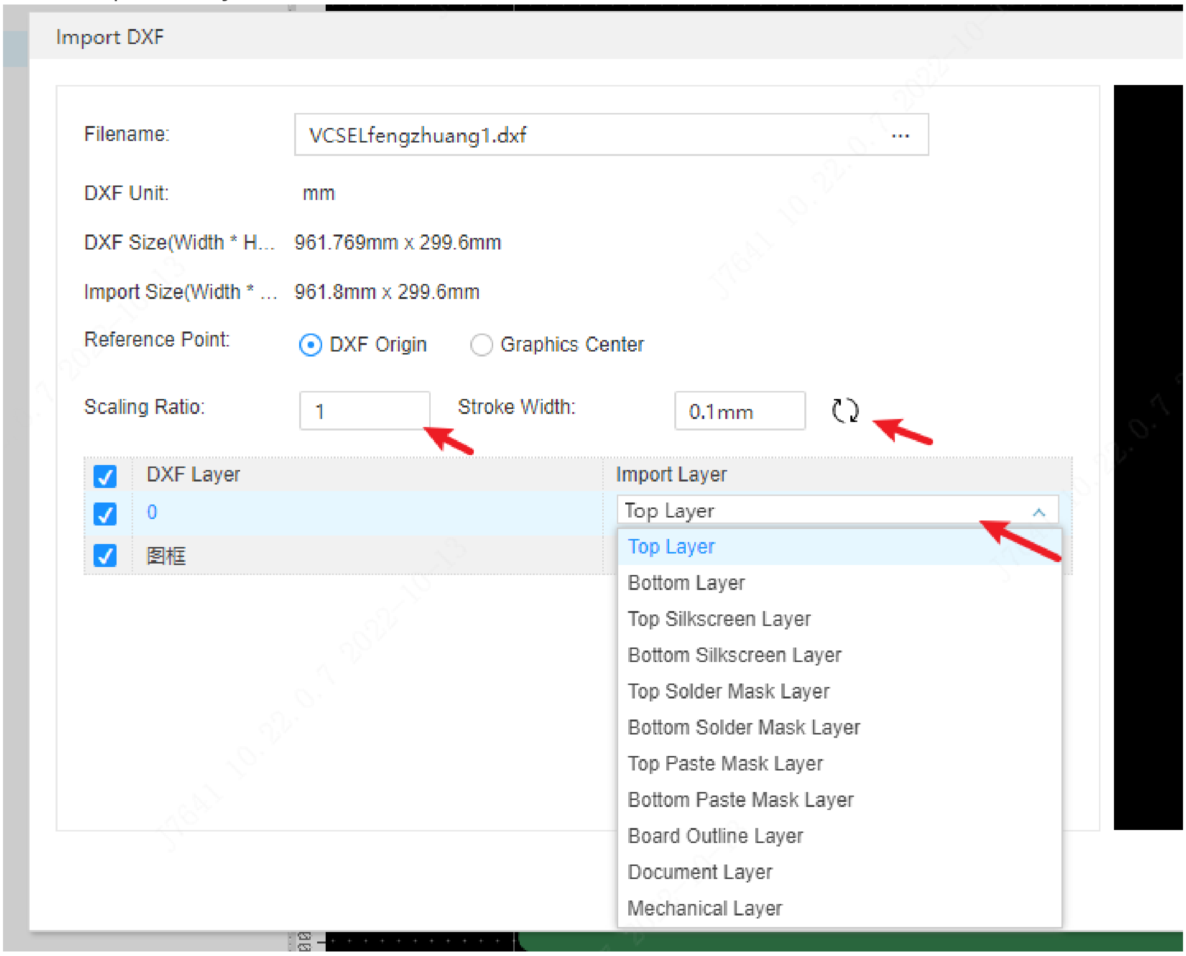


Options:



- **DXF unit**: default unit mm, cannot be modified
- **Scaling**: Imported DXF scaling modification
- **Line width**: The width of the imported DXF line, the default unit is inch, which cannot be modified.
- **width/height**: according to the size of the DXF file
- **Reference Point**: the reference point of the original DXF file and the center point of the DXF

Select imported layers

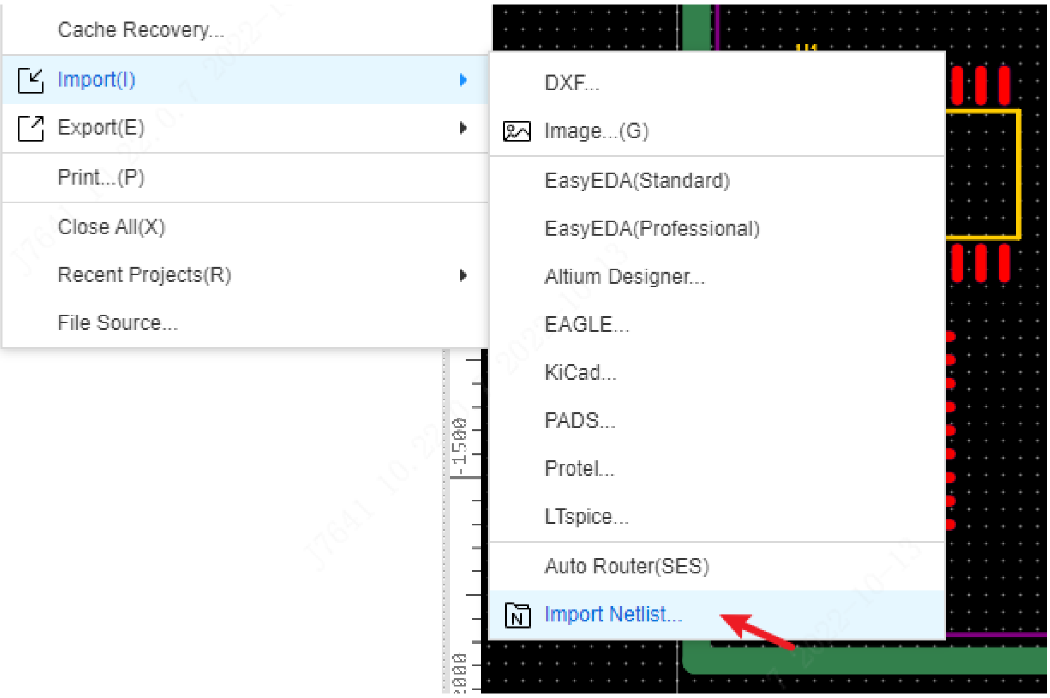


Import Netlist

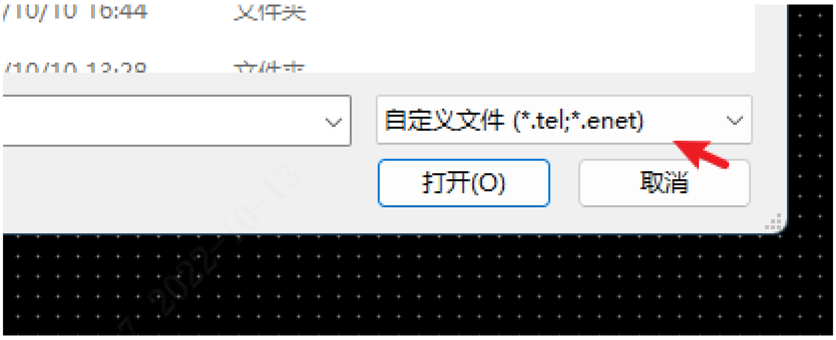
The netlist, also known as the netlist, is the representation of network connections and connections. The exported netlist file can be imported into the PCB, and the PCB will automatically generate the corresponding component information on the PCB according to the package information in the netlist.

Steps:

- Top Menu - File - Import - Netlist

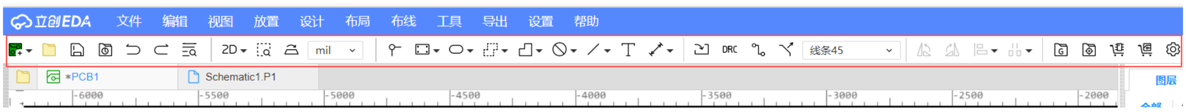


EasyEDA support two netlist format, EasyEDA (enet) and Allegro (tel)



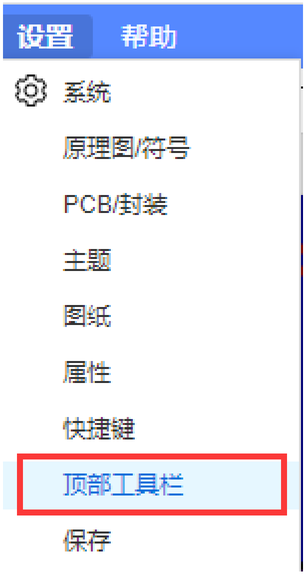
Top Panel - Toolbar

The toolbar in the top panel of the PCB is a shortcut bar for some commonly used tools

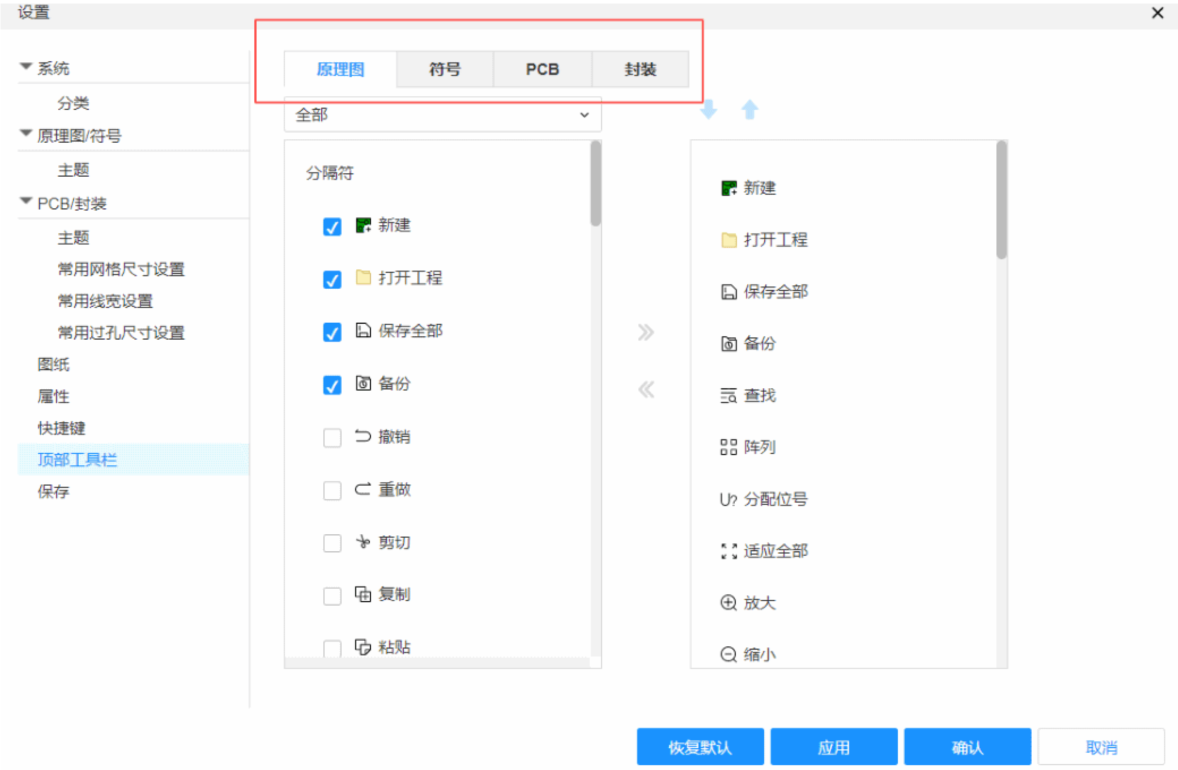


Set up the shortcut bar

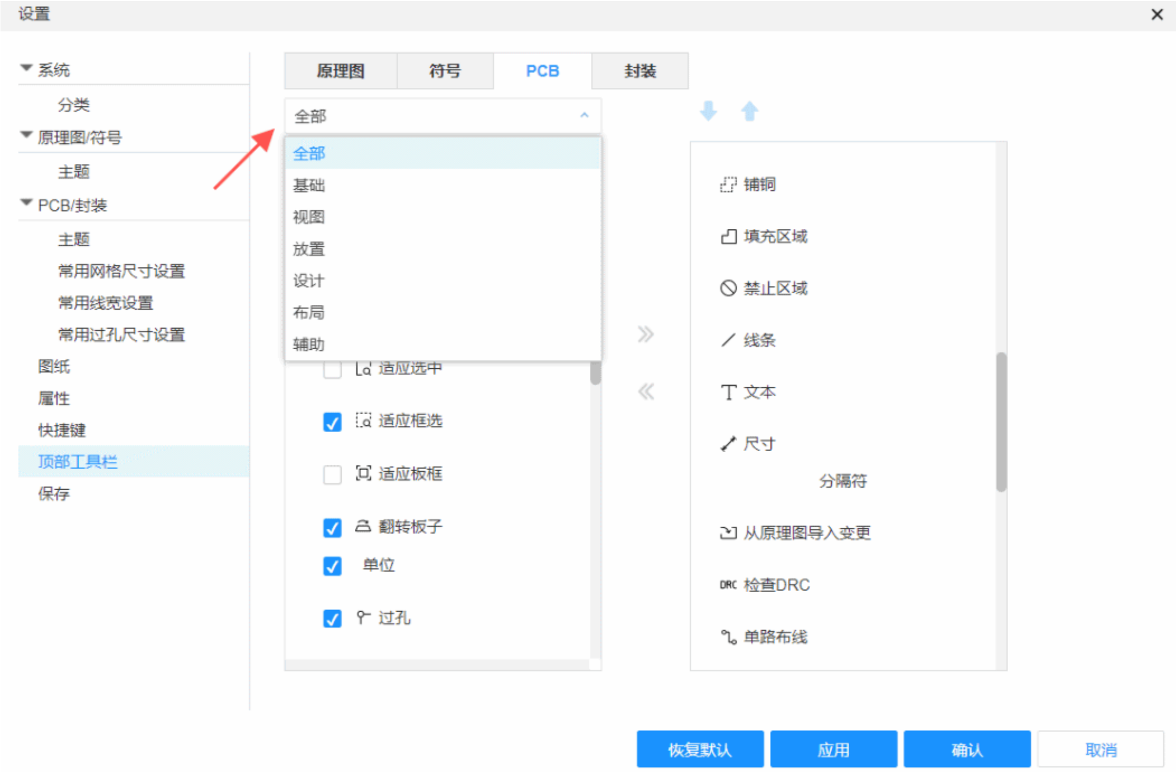
- Top Menu - Settings - Top Toolbar



In this pop-up window, you can set the top toolbar settings for schematics, symbols, PCBs, and footprints. The red box in the figure refers to the top menu of each interface.

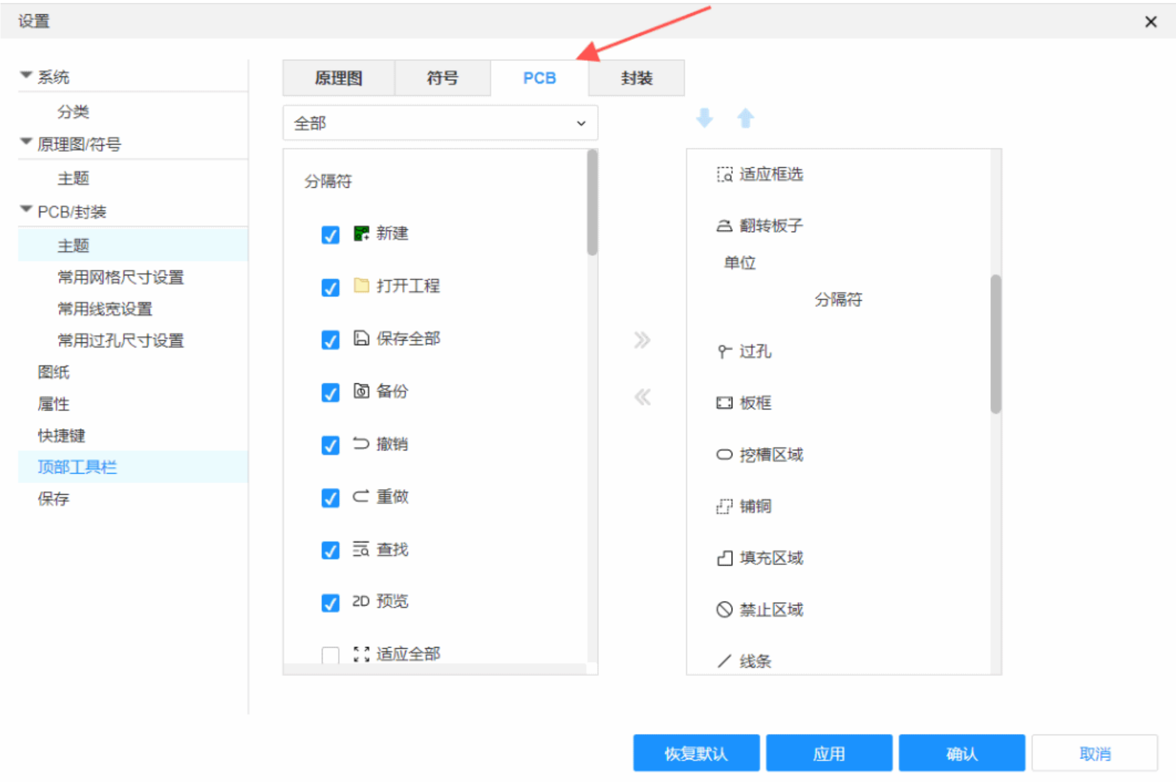


Click the checkbox to see a categorized display of tools

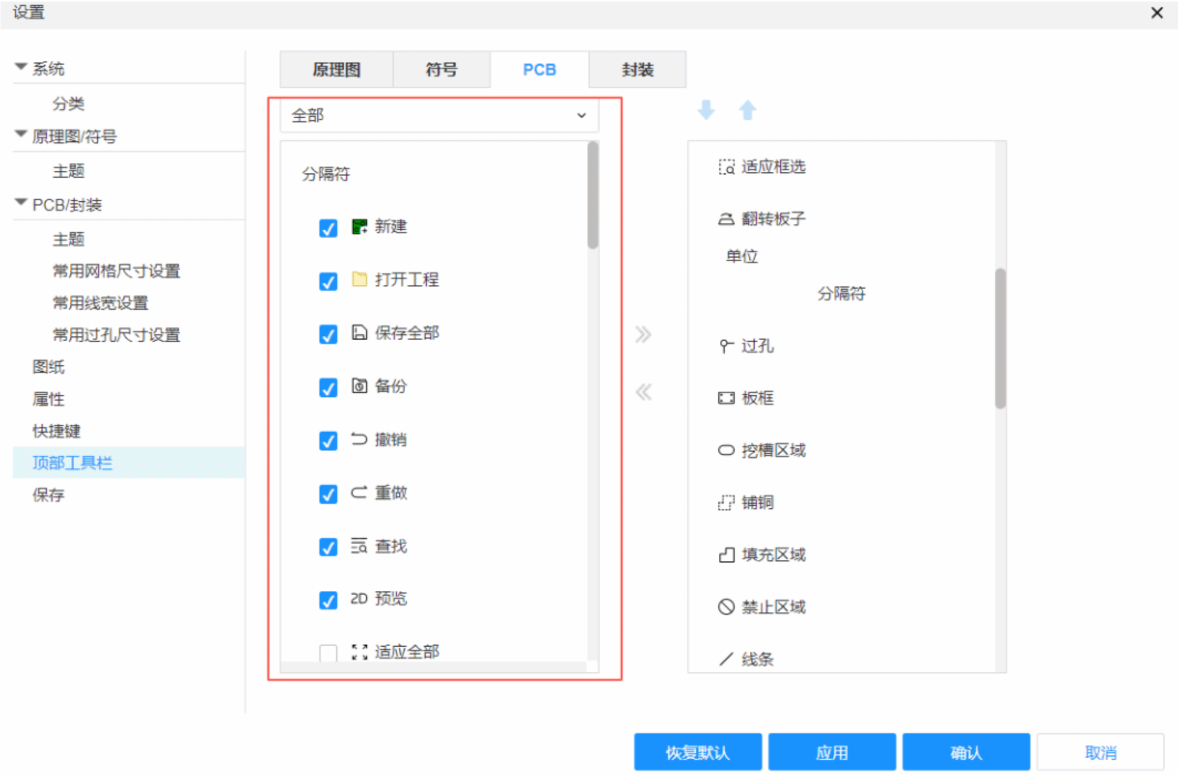


Customize the top toolbar

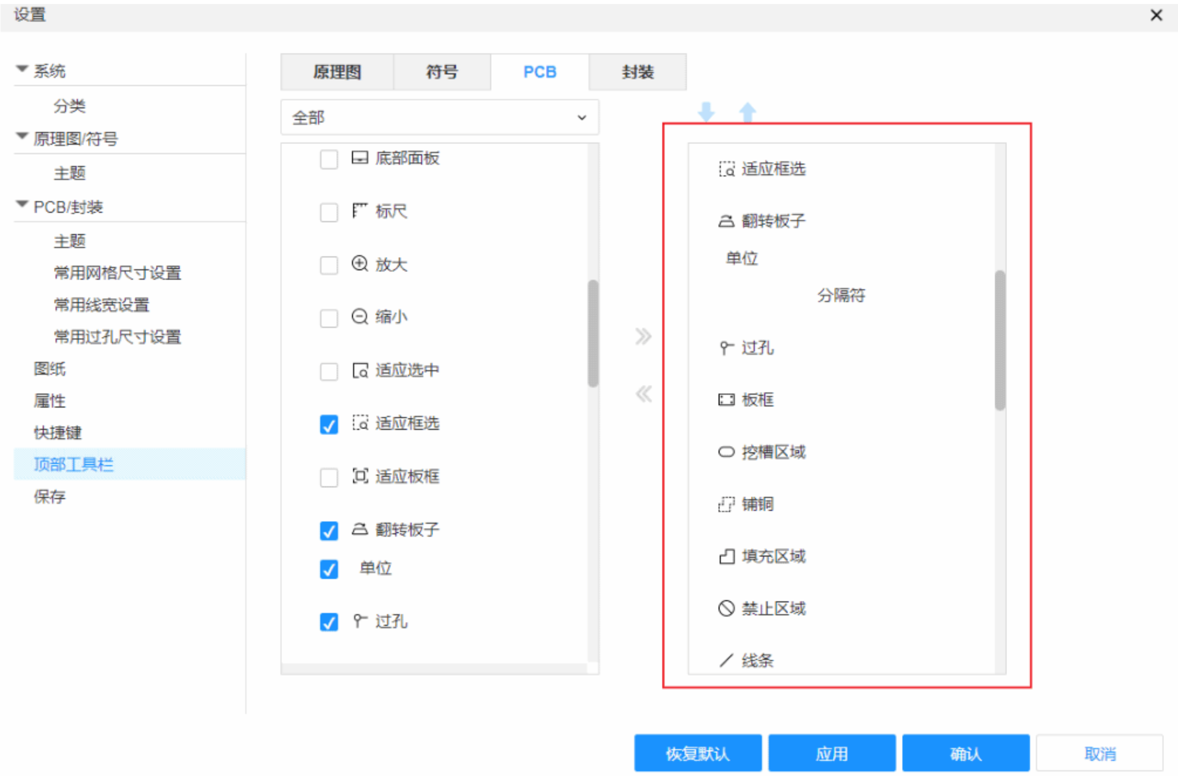
Click Settings on the top toolbar of the corresponding interface



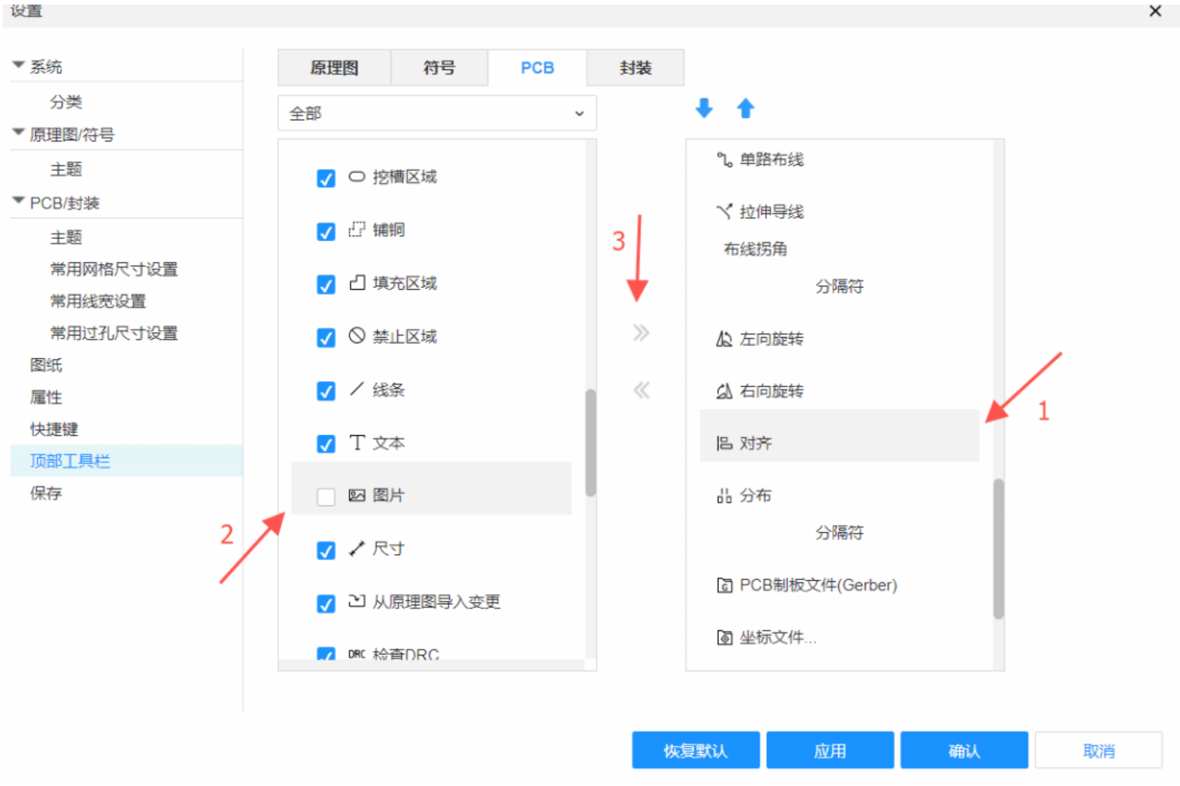
On the left side of the pop-up window are some shortcut tools that can be added to the top toolbar

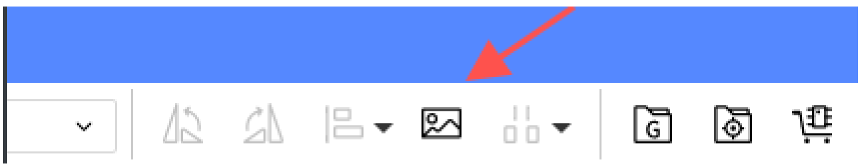


The right side of the pop-up window is the shortcut tool that has been set in the top toolbar



First select the location on the right to add the shortcut function, and then select the shortcut function to be added to the left on the left, click the arrow in the middle or tick the check box, and click OK to add it to the selected location. !

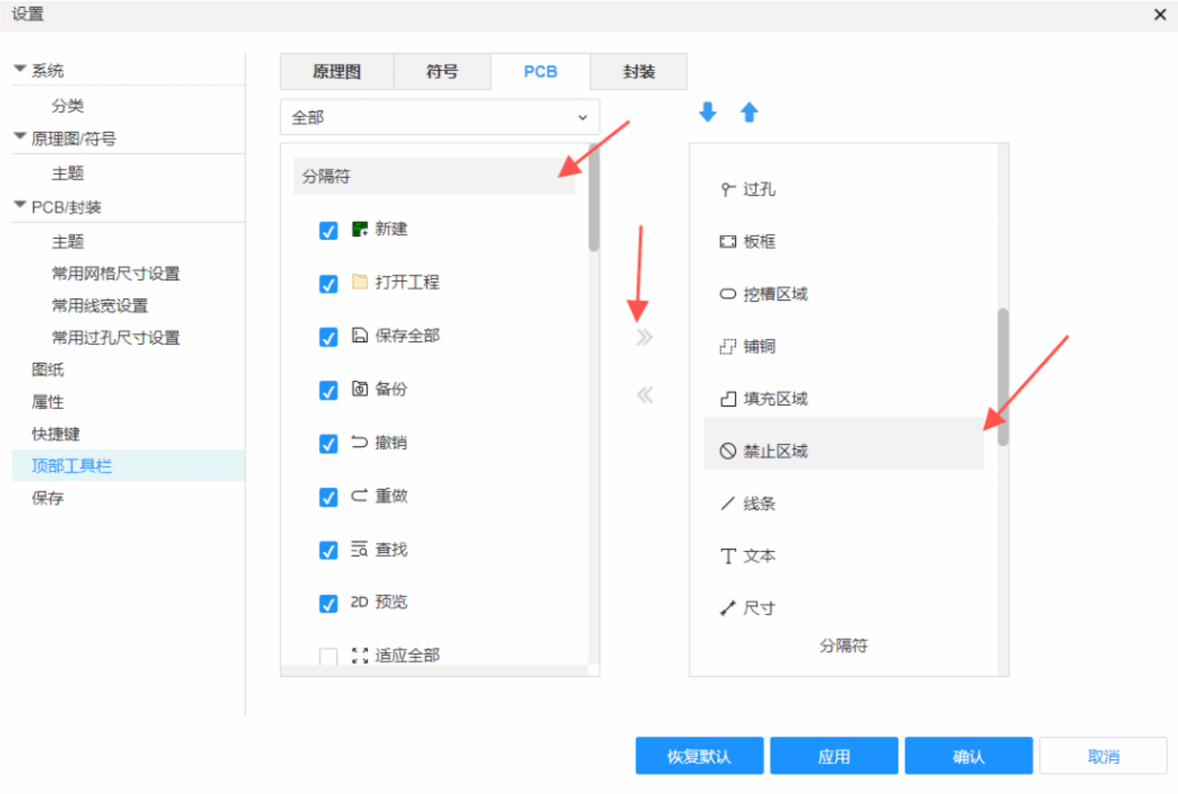




Classification

In the top toolbar, each function shortcut will have a separator as a classification plan and this separator can be modified in the settings.

The method of adding a separator is similar to the operation of adding a shortcut function. First, select the place to be separated on the right, select the separator from the left, and click the arrow in the middle to add the separator to the top toolbar, and click Save. Save and apply the current settings to the PCB interface.

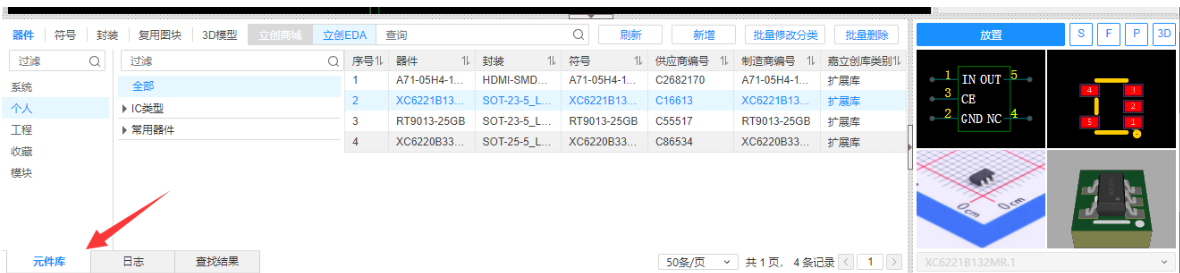


As you can see, the shortcut functions of the top toolbar are separated

Bottom Panel - Library

The component library is in the bottom panel of the PCB interface. The component library in the component library can directly place components in the PCB interface, but symbols, footprints and multiplexing blocks cannot be placed.

The component library includes system library, personal library, collection library and component library for joining the team. The component library also includes device library, symbol library, footprint library, and multiplexing blocks.



Device Search

The search at the top of the library is a global search that searches the entire library

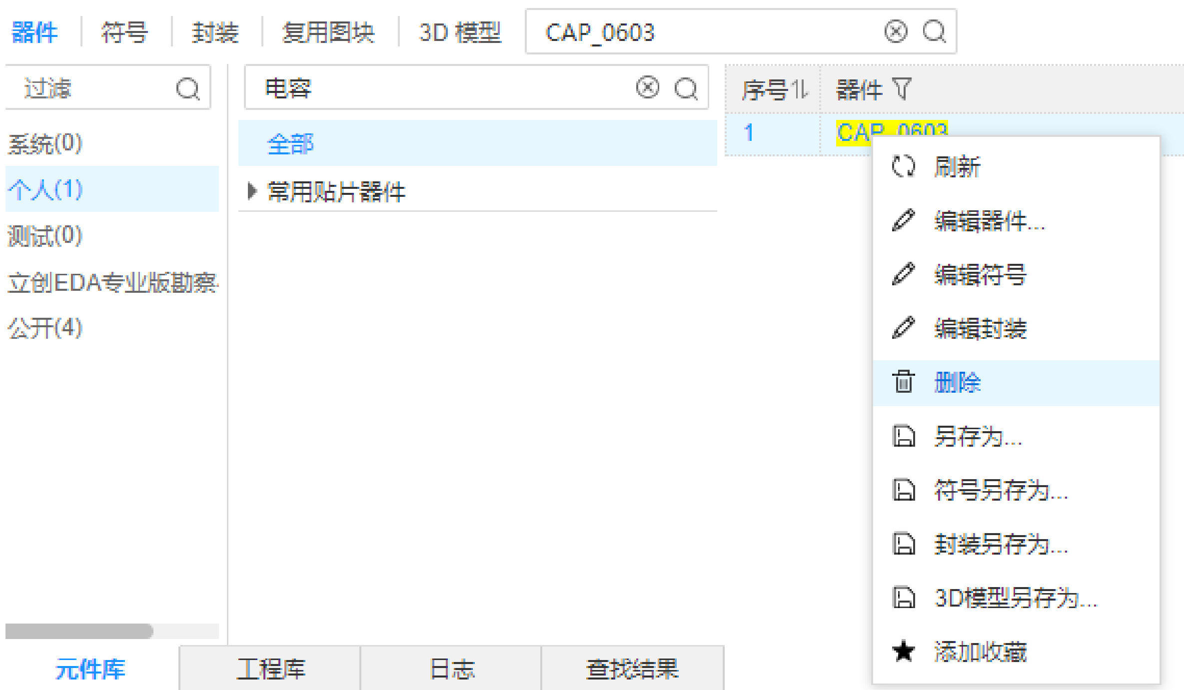
And the search box below is to search the system or personal device library by category

edit element

Select any device in the component library and click the right mouse button to modify the device.

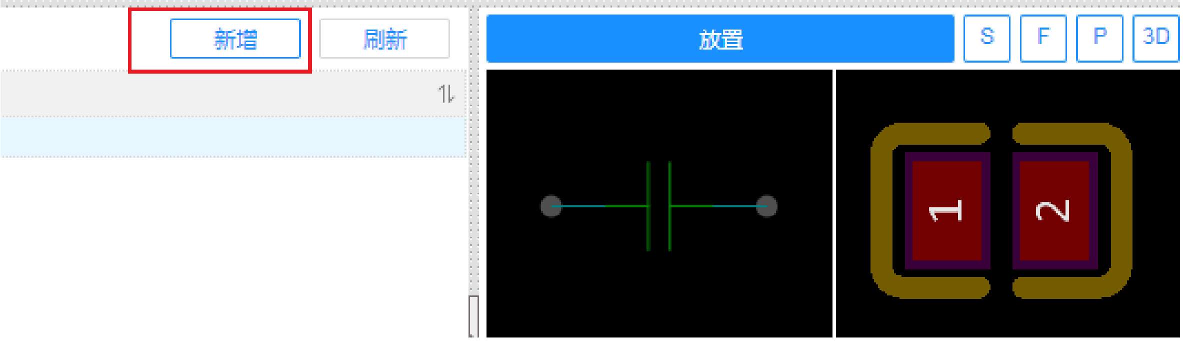
illustrate:

- System library does not support modification!



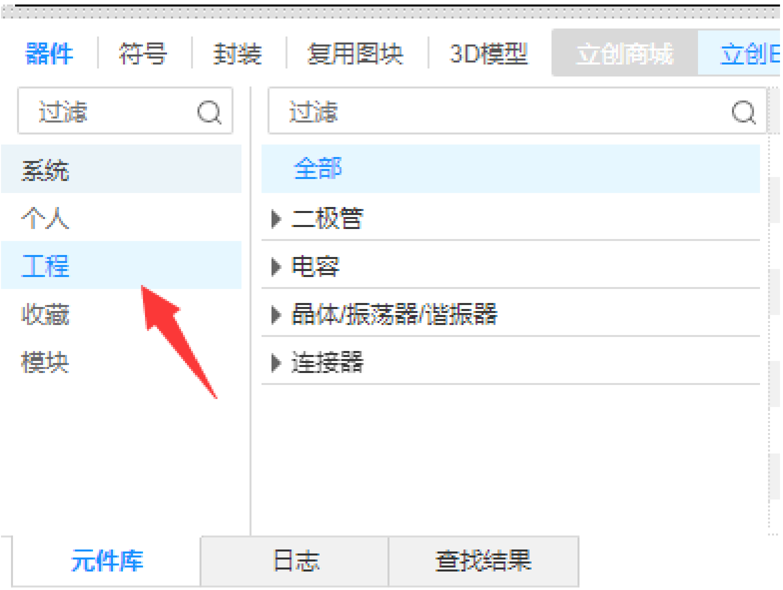
new device

Click the new button on the upper right in the component library to quickly create a device. The symbols and packaging operations are the same.



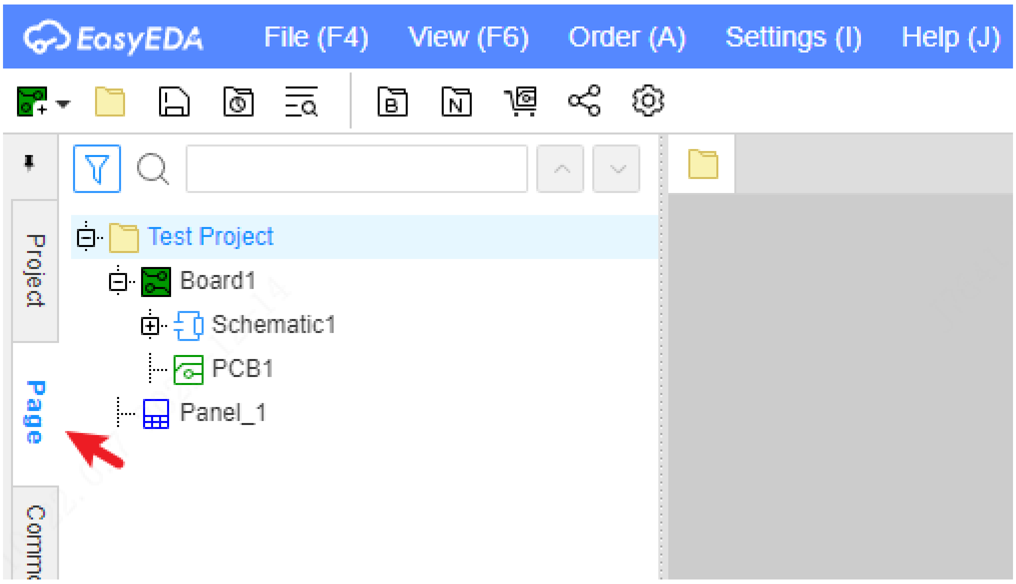
Bottom Panel - Project Library

The engineering library of the PCB is the same as the schematic engineering library. The engineering library is all the device libraries currently placed in the project, including symbols, footprints and multiplexing blocks. Adding devices and special symbols to the project can be displayed in the engineering library. , the device engineering library deleted in the project will not be deleted, it will be kept in the engineering library, make a historical record, record the devices placed in this project, and can also click to use or modify again, and modify it in the engineering library The device can only be used in the current project and has no effect on the component library, but the device that has been placed in the project cannot be deleted from the project library.

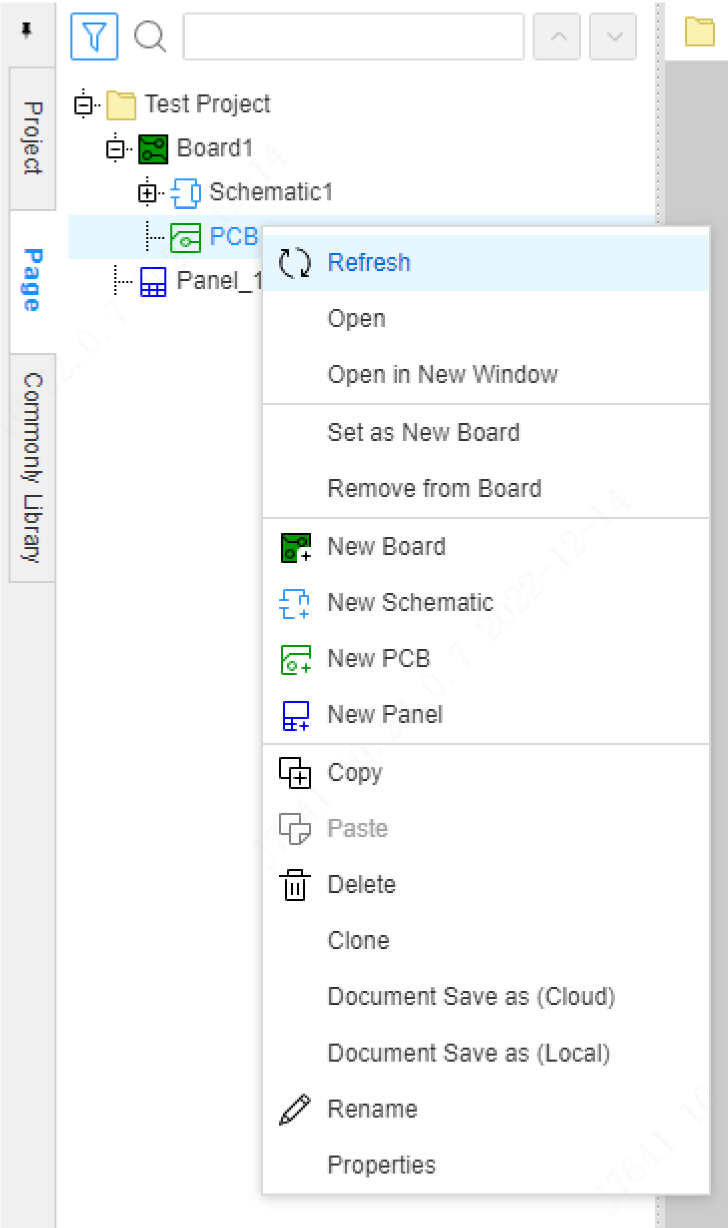


Left Panel - Page

The created project and drawing page can be viewed in the - drawing page in the left panel. Double-click to open the created file and edit it.

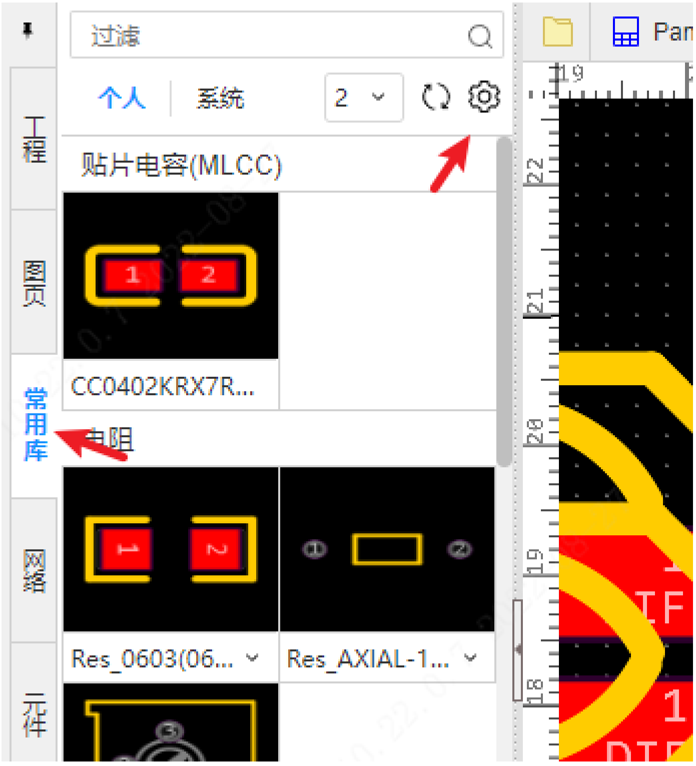


The diagram page Tab on the left is the same as the diagram page Tab of the previous schematic diagram, and various operations can be performed through the right-click menu.



Left Side - Commonly Library

After opening the PCB, you can see a common library Tab on the left panel, which supports some common libraries built into the display system, and can also support custom common libraries. Common libraries for PCBs can be easily used for designs that do not require schematic drawing.



The operation of setting the common library is the same as that of setting the common library in the previous schematic diagram, and it is shared. The set common library displays a preview according to the package bound by the device, and supports clicking and moving the mouse to the canvas for placement.

Fig.123

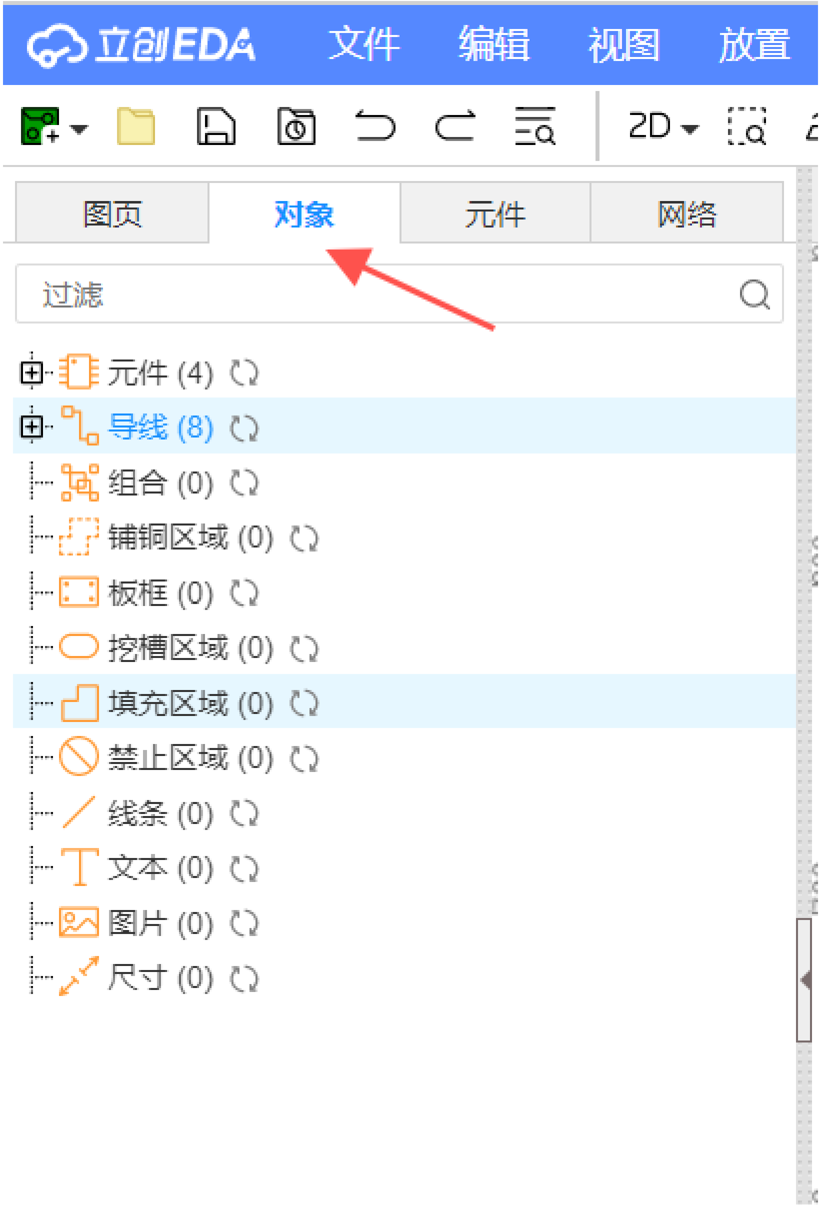
The Schematic and PCB tick box behind the device indicates whether the device is displayed in the Schematic or PCB common library.

Note: When the editor is opened for the first time, the thumbnails of the commonly used libraries will be automatically generated. When the computer performance is poor, the thumbnails of the commonly used libraries will be displayed slowly.

Left Panel - Object

In the "Object" panel on the left side of the PCB, you can view the elements and the number of elements placed under the current PCB interface.

Correspondingly, each branch can be single-clicked to highlight in the PCB, and double-clicked to zoom into the device and highlight it.



Left Panel - Components

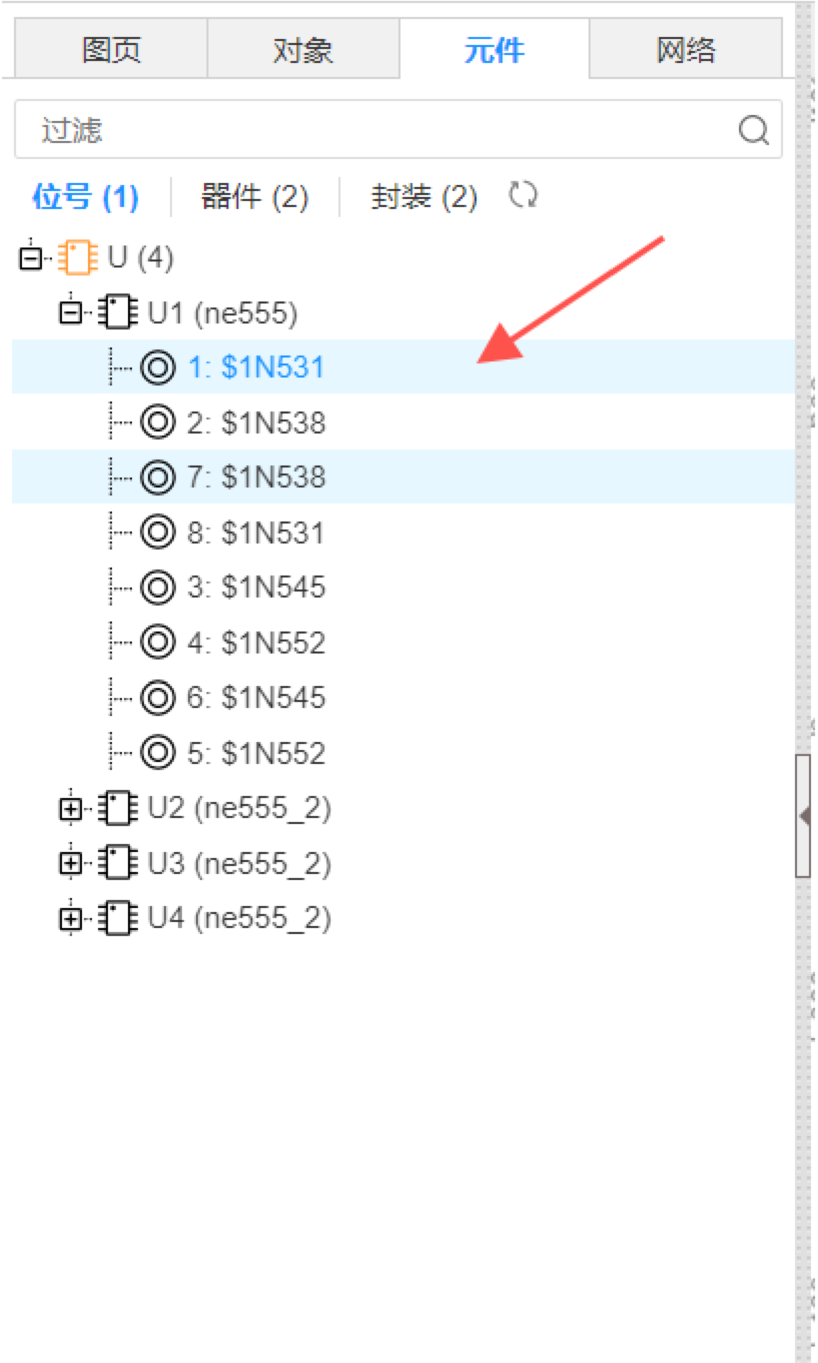
The component interface on the left panel displays information such as the number of components, pads, bit numbers, and footprints placed on the current PCB interface.

Click to highlight the selected element, double click to follow to the PCB interface and highlight it to zoom in.

Designator

Display the same type of tag device on the PCB interface

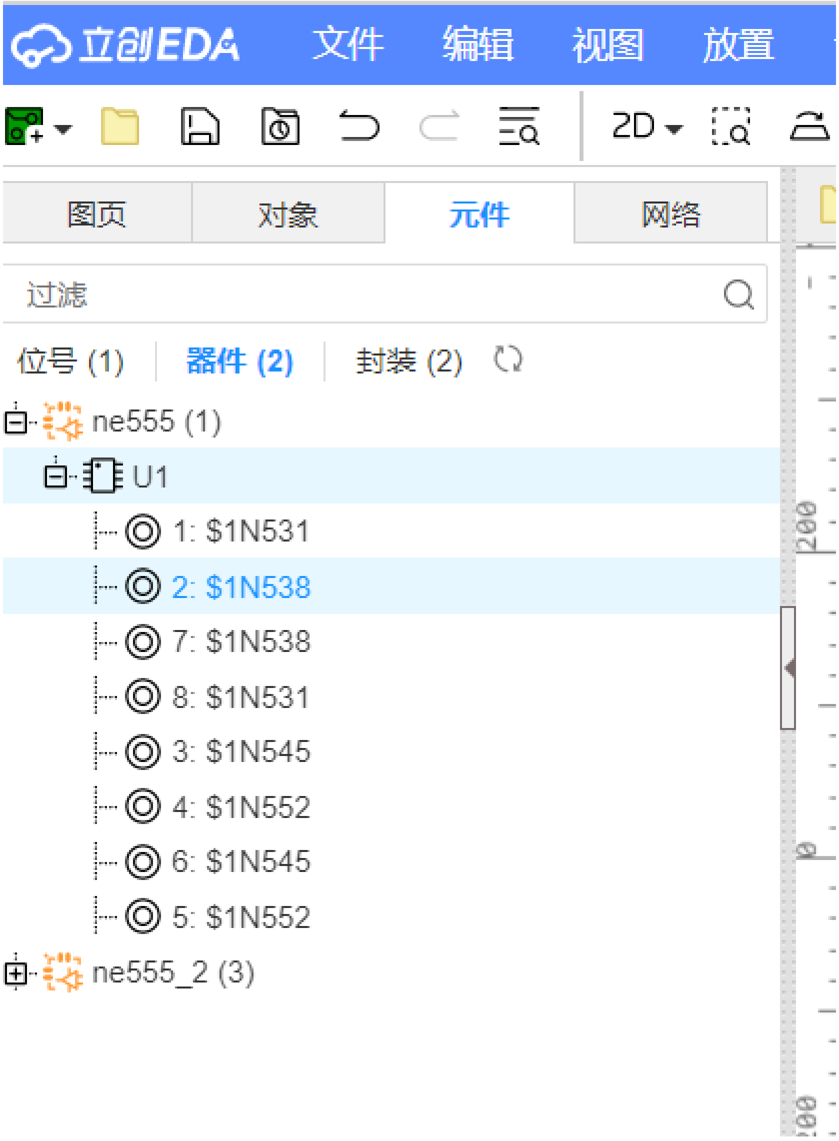
Click the designator or footprint to jump to the device under the corresponding designator on the PCB interface, and click the pad under the device to highlight the pad of the selected device.



Device

Display the components placed on the PCB interface, and the names of different components will be displayed separately.

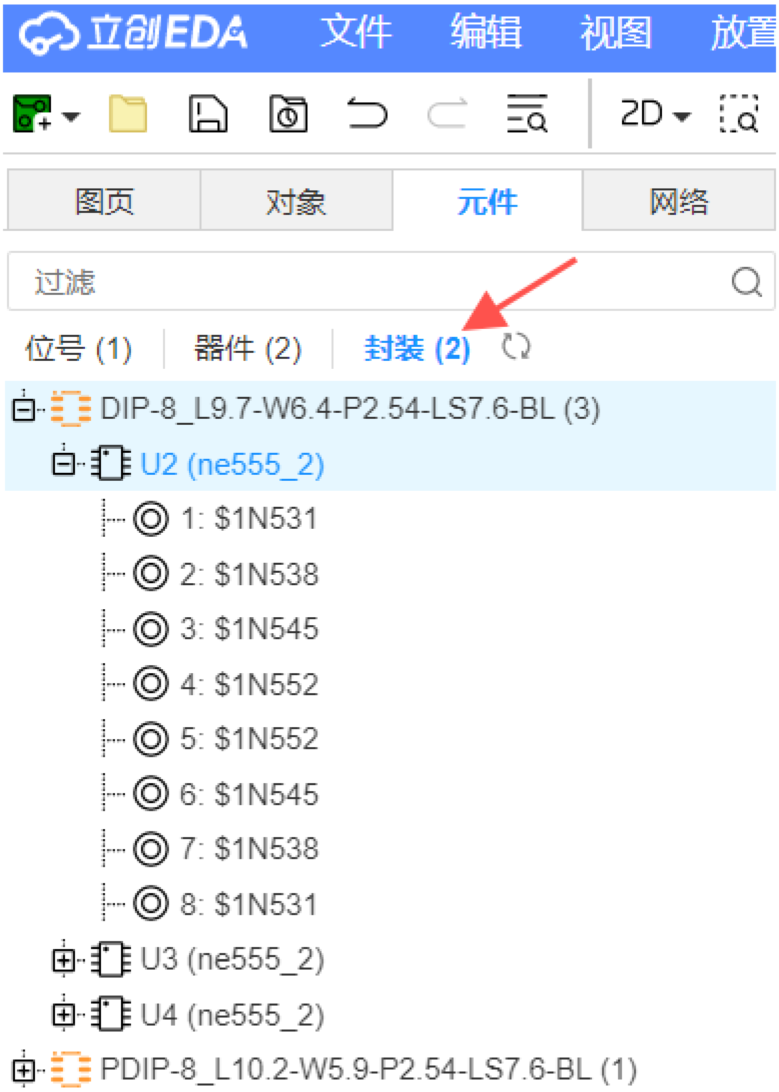
Click the device to jump to the corresponding device on the PCB interface, and click the pad under the device to highlight the pad of the selected device.



footprint

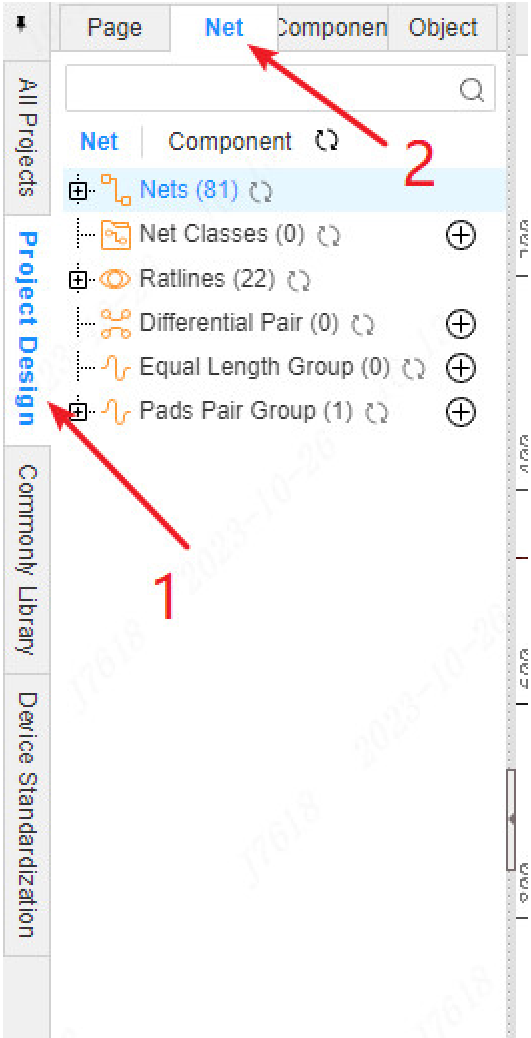
Displays the footprints that have been abandoned on the PCB interface, and different footprints will be displayed in categories.

In the same way, click the footprint to jump to the corresponding footprint on the PCB interface, and click the pad under the device to highlight the pad of the selected device.

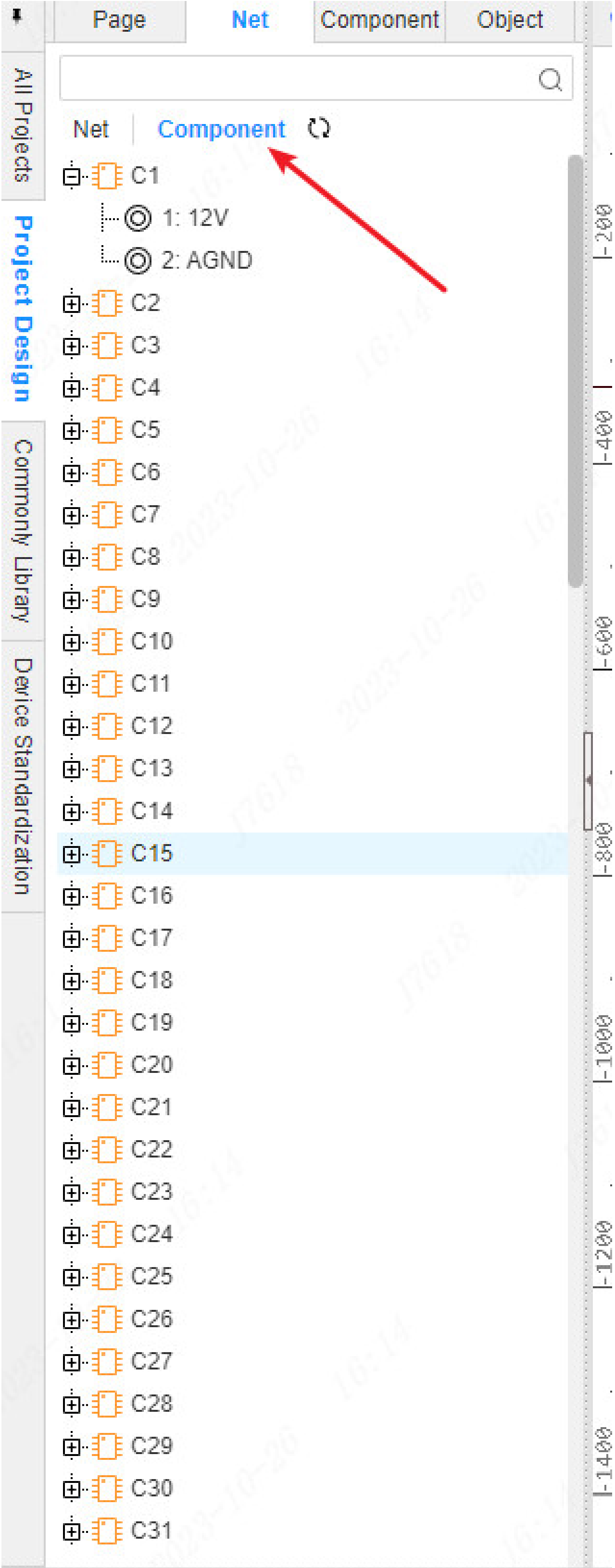


Left Panel - Nets

The left panel on the PCB interface-the network display the network of the current PCB interface, which is divided into nets, net class, ratlines, differential pair, equal length groups, pads pair group.



Switch the tag to the component above will display the network of pads in the current PCB



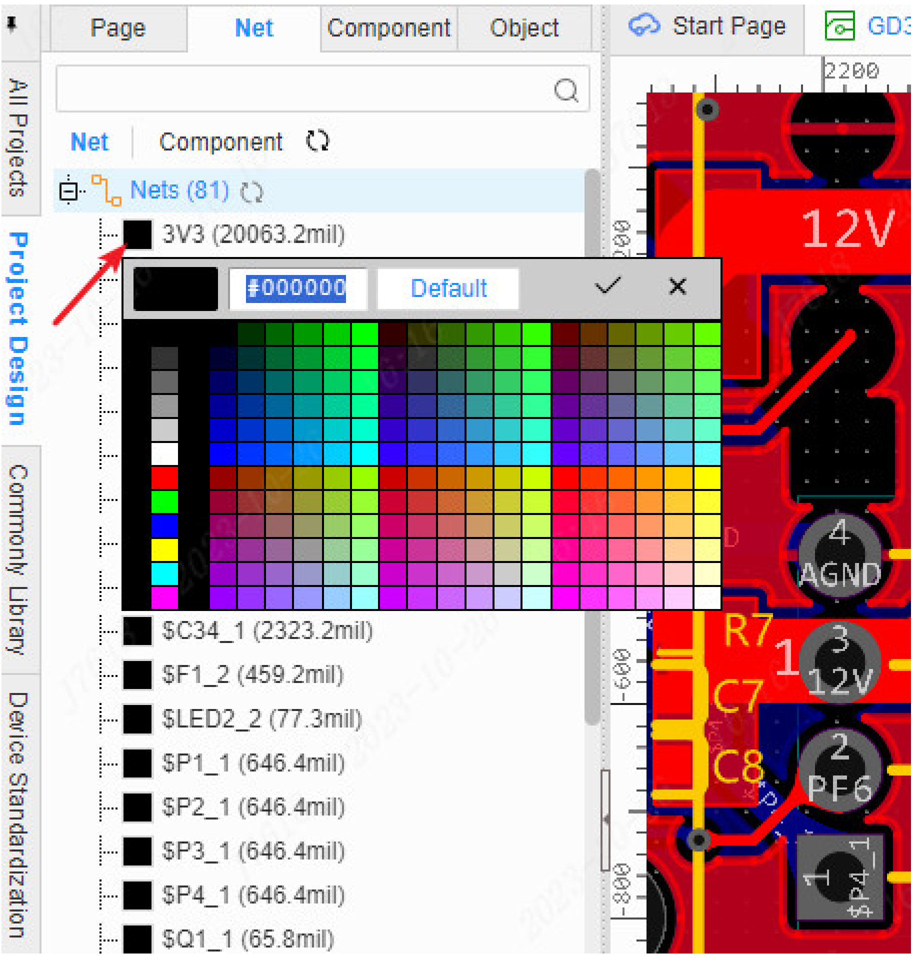
Net

Display the number of networks and specific networks in the current PCB interface,Among them, the empty network will also display, the name is None.

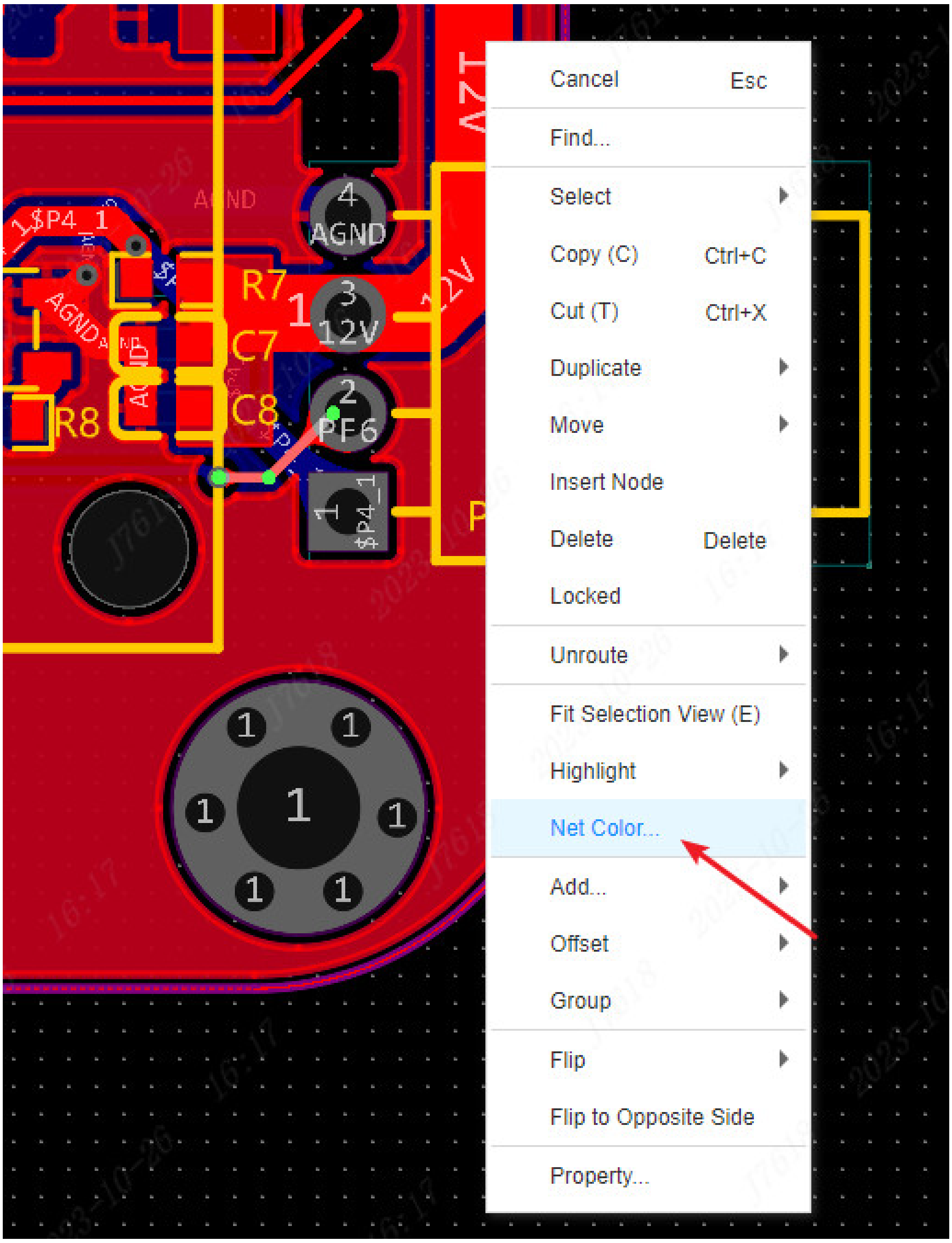
Network positioning: Click the network to light up in the PCB interface and translate to the center.

Double -click the network after the canvas and highlight it in PCB.

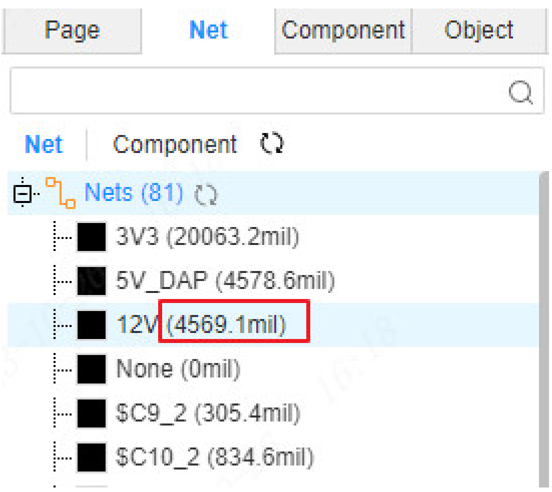
Network color: Set the network color: Click the color cubes in front of the network name to open the color setting panel and set the demonstration you need.



You can also right -click the color to set the color



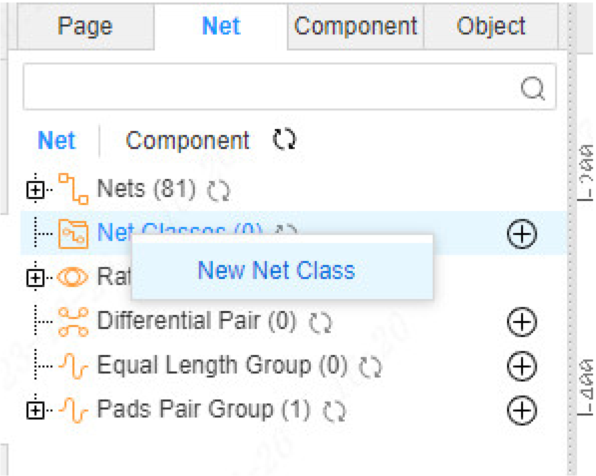
Network length: The parentheses behind the network name are the length of the current network. The unit follows the canvas unit. Note that the length of the network here is only the length of the wire. If the line or the filling area is used to connect the pad, the length of the network will not be included here.

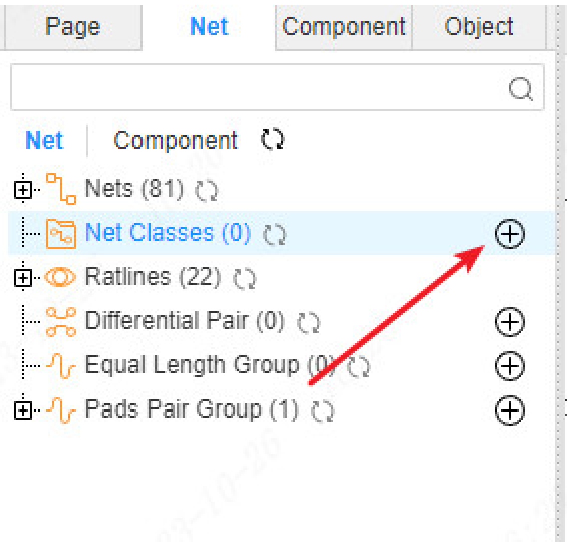


Net Class

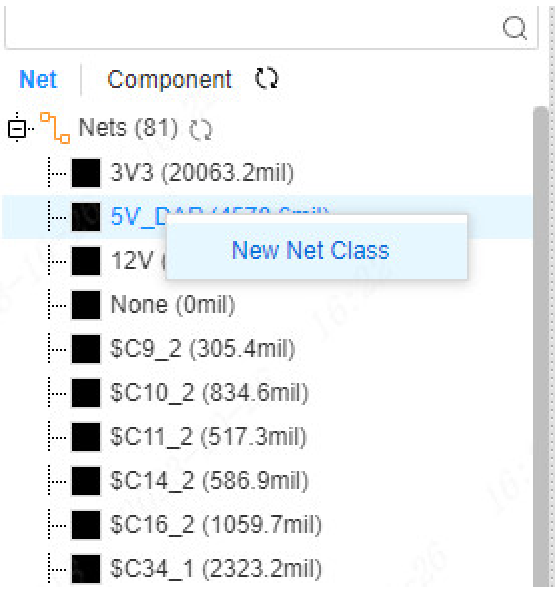
The network class is generally used when the same type of network needs to set the design rules together.

Right -click the mouse under the network branch -new classification or click a new small icon

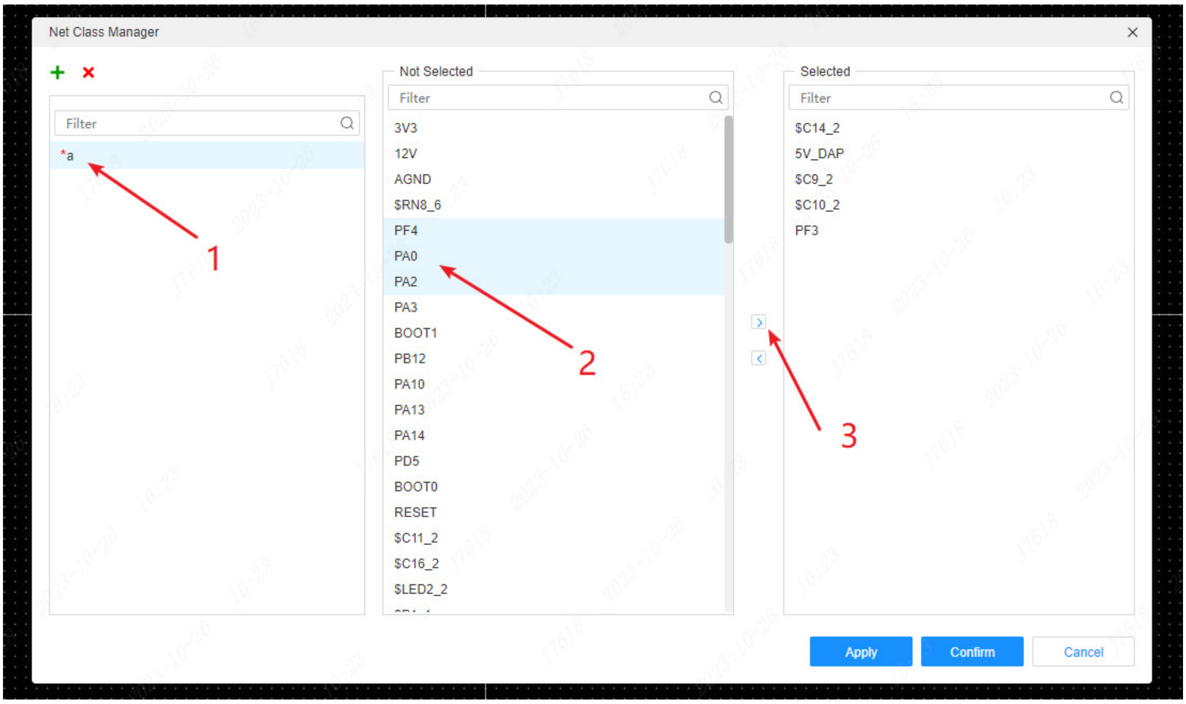




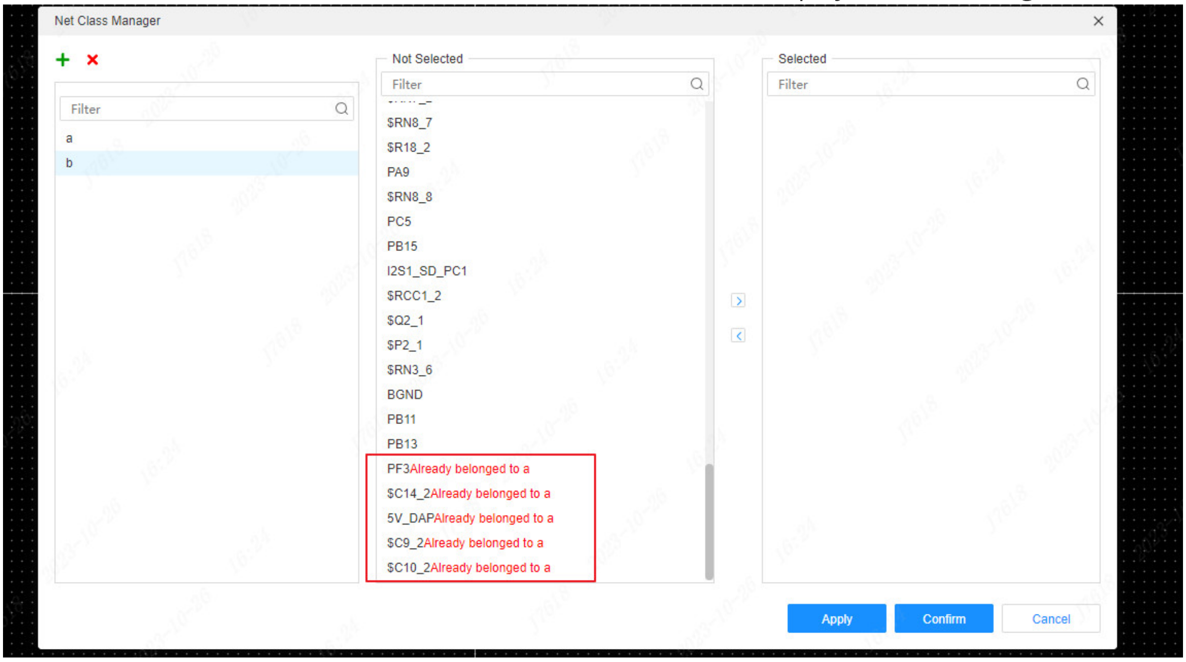
Or, right -click the "network" list on the "network" list above to create the "network".



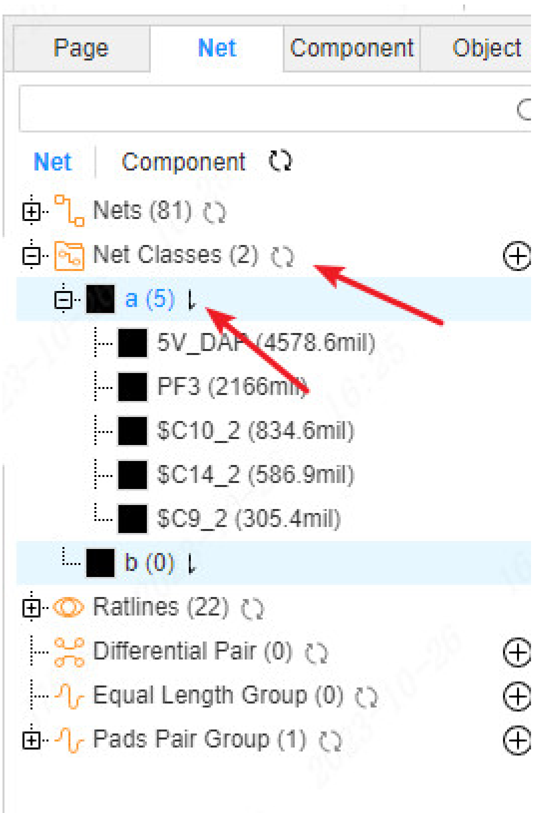
Set the name in the network pop -up window, and set the network.



The network name that has been added to the network will be displayed in the dialog box



After the creation is completed, refresh the left network class, and you can see the new network category.

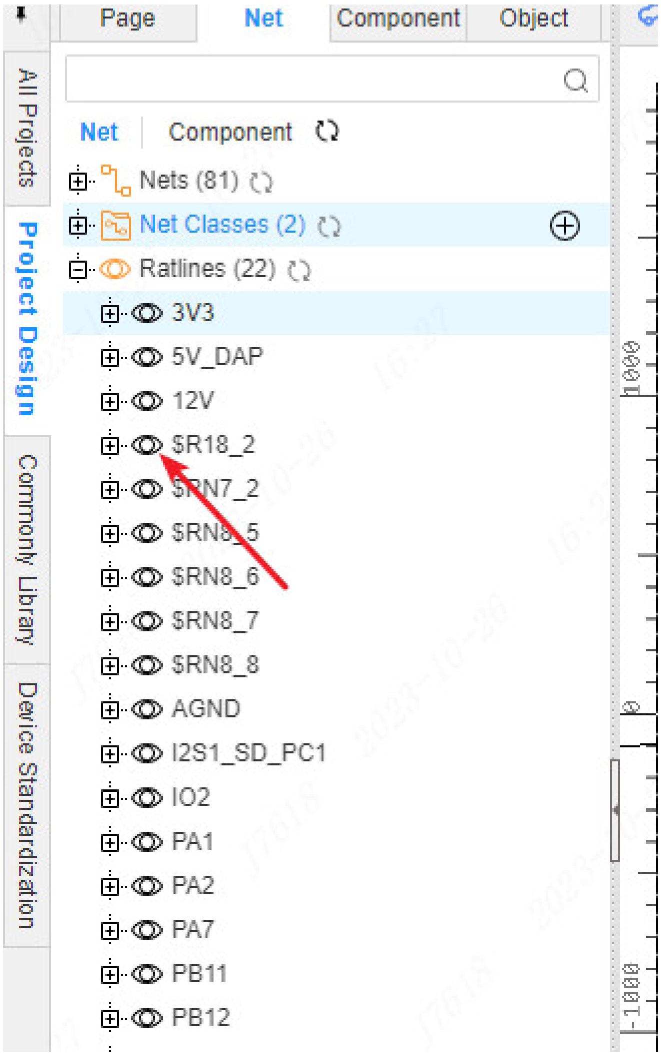


Click the sorting button to sort the network in the network class to facilitate viewing the network according to the network class.Click to sort according to the network name and network length.

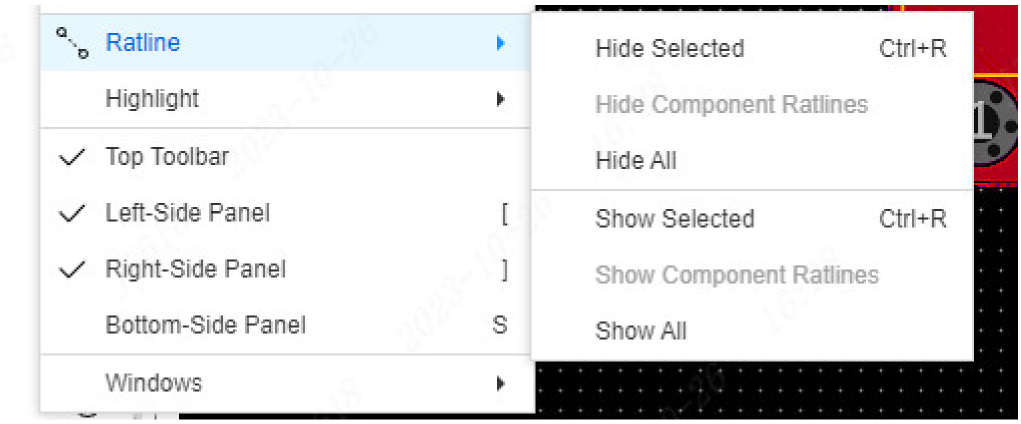
Ratlines

When wiring, hiding other flying wires that do not need wiring for the time being, which will help reduce interference and focus on the current network wiring.

You can hide the flying line on the left online tree



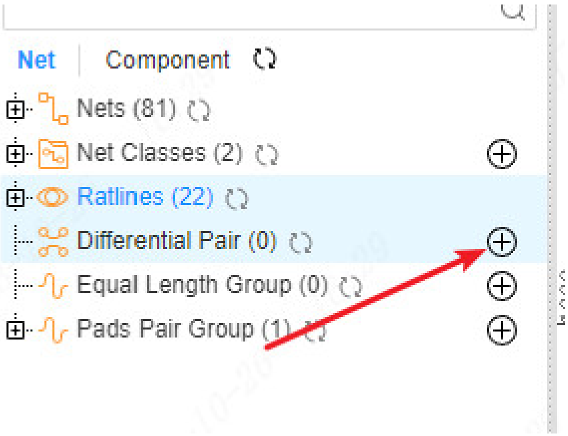
You can also select the wire after selecting the wire, and the hidden flying wire at the top view menu



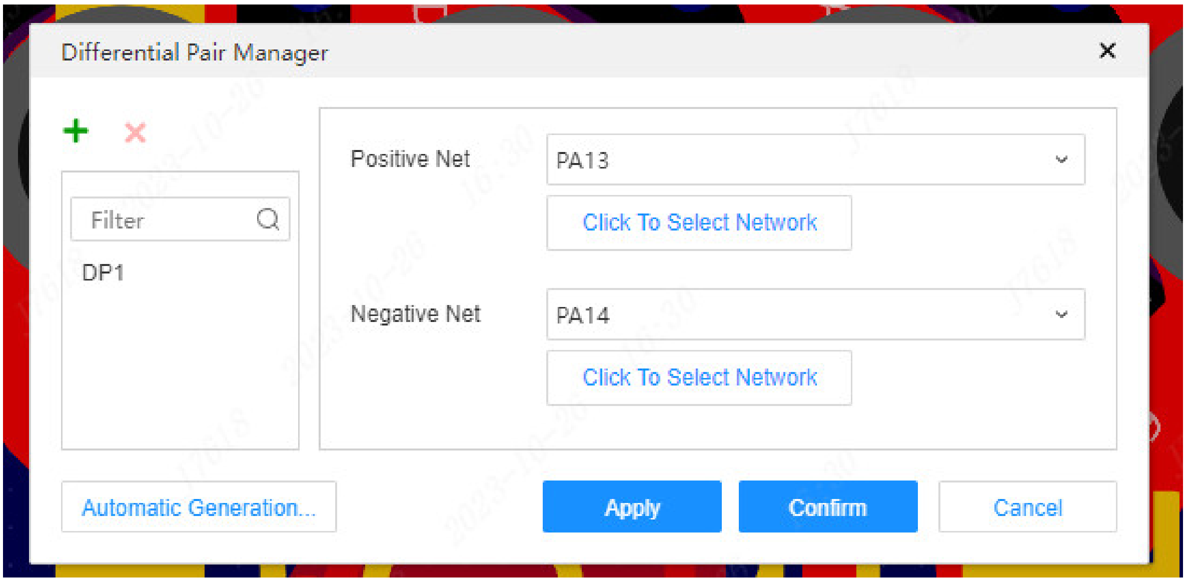
Differential pair

When the difference is required, the differential pair needs to be created first and the rules are set.

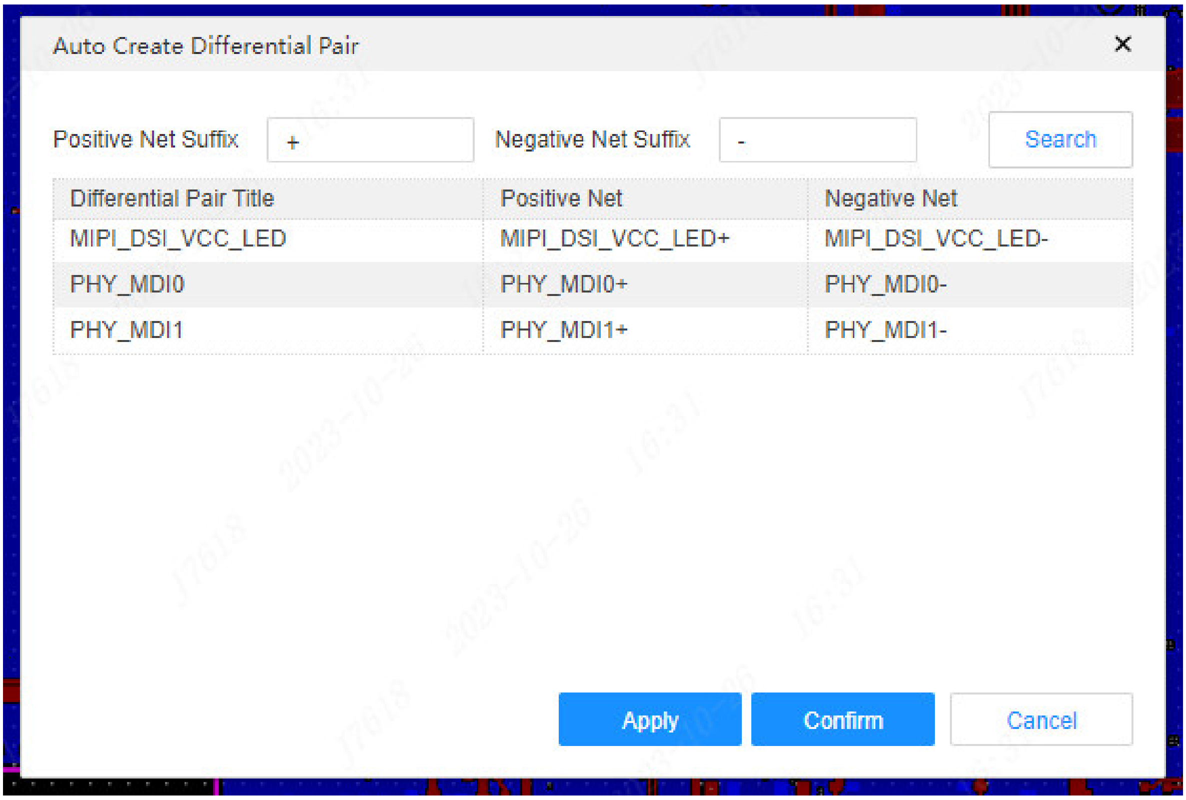
Click the new icon of the differential pair to create a new differential pair.



Set the name of the differential pair, add the positive and negative network after adding a positive and negative network

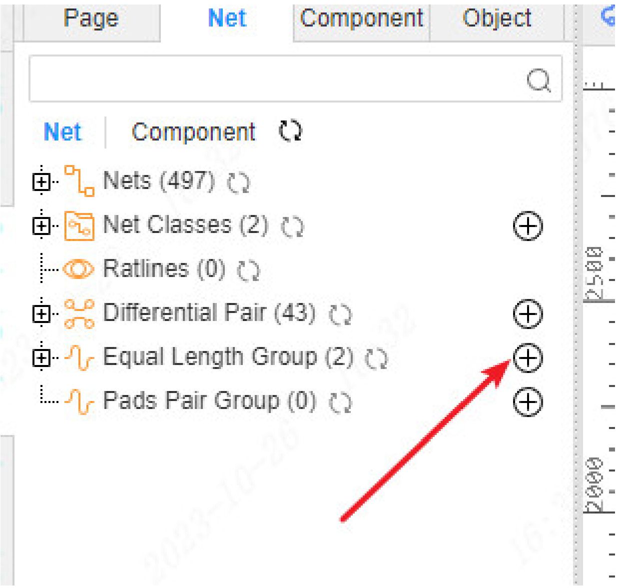


You can also use automatic generating differential pairs, which will match the same prefix according to the network name, but the suffix is different.

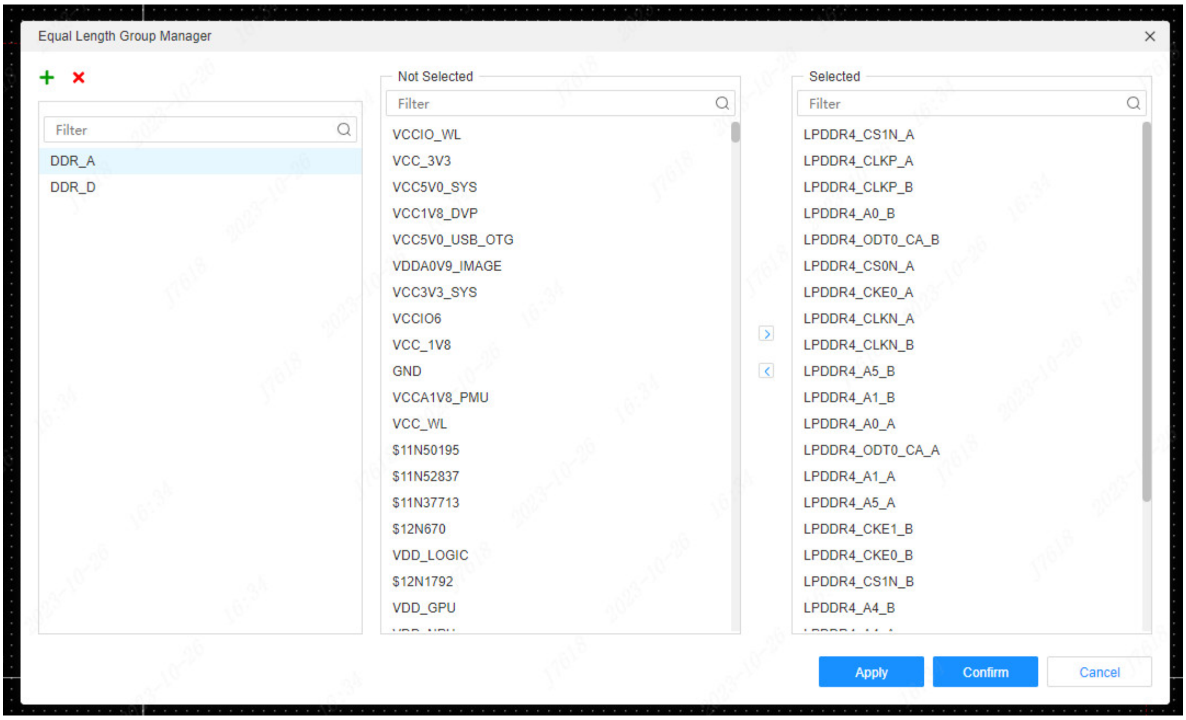


Equal length group

When the wiring of long -controlled control is required, the long network group can be set to compare the network length between the two or more networks, and it is convenient to set the network rules for the entire network group in the design rules.

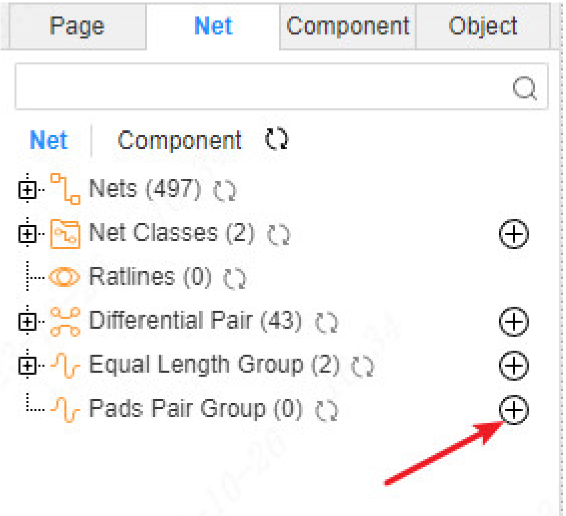


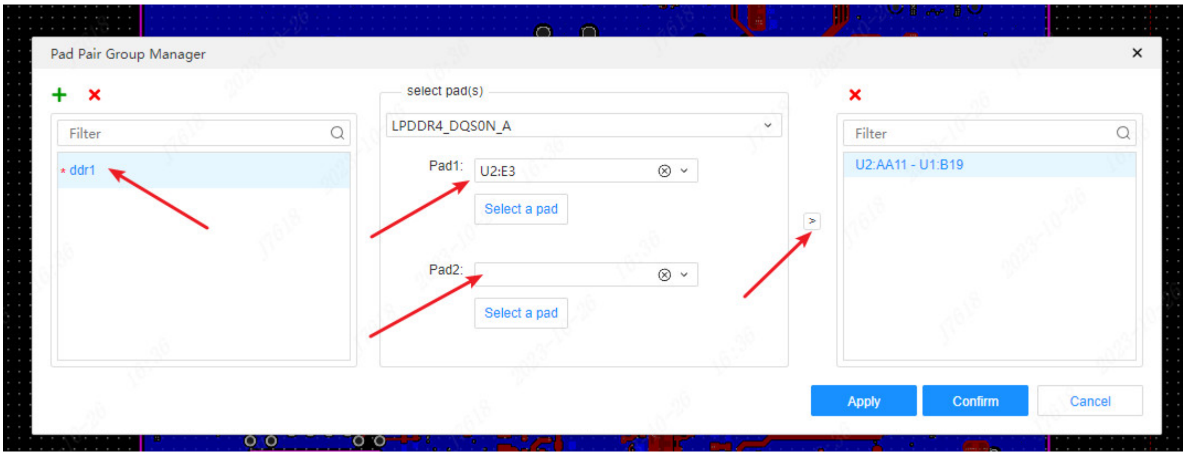
Similar to the new network class before, click the new button to create.



Pad pair group

Creating a pad to the group can be very convenient for differential control to the length. You can intuitively view the length of the wire network between the two pads, which is conducive to the equal adjustment.

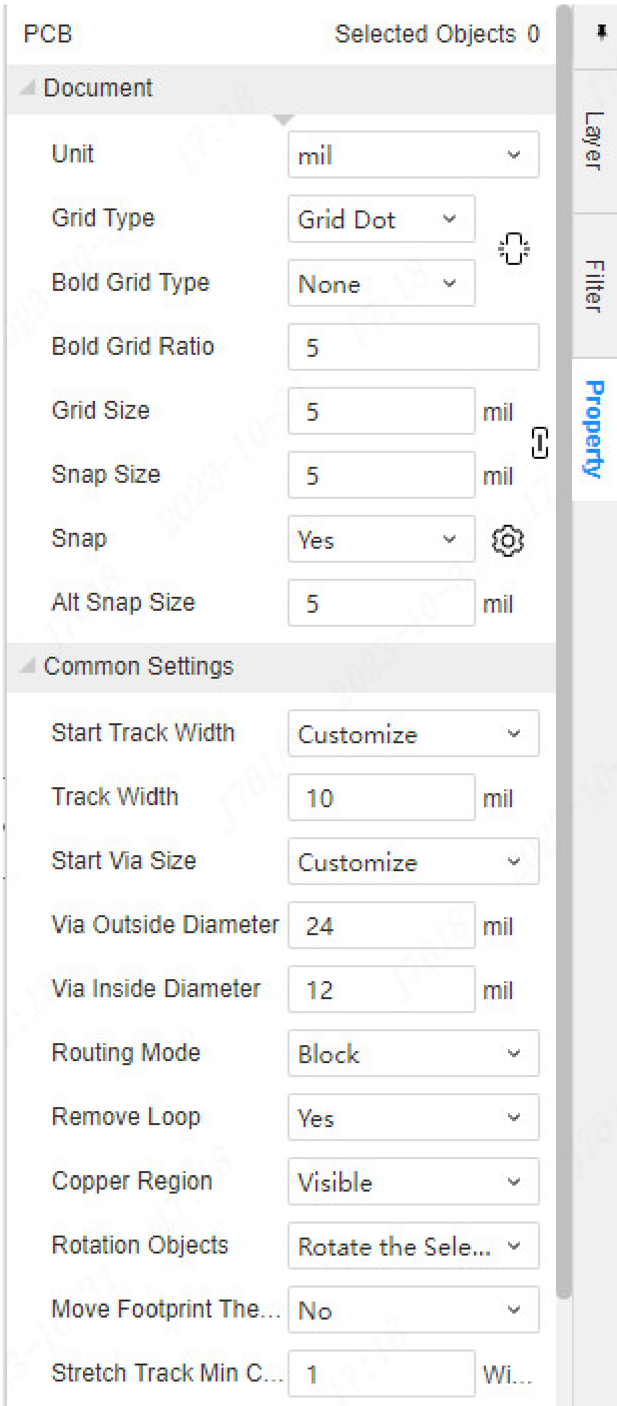




Right Panel - Property

Canvas attribute

When you click on the blank space of the canvas, you can click to switch to the canvas attribute settings.Jialin EDA provides many common settings items for subsequent drawing operations.



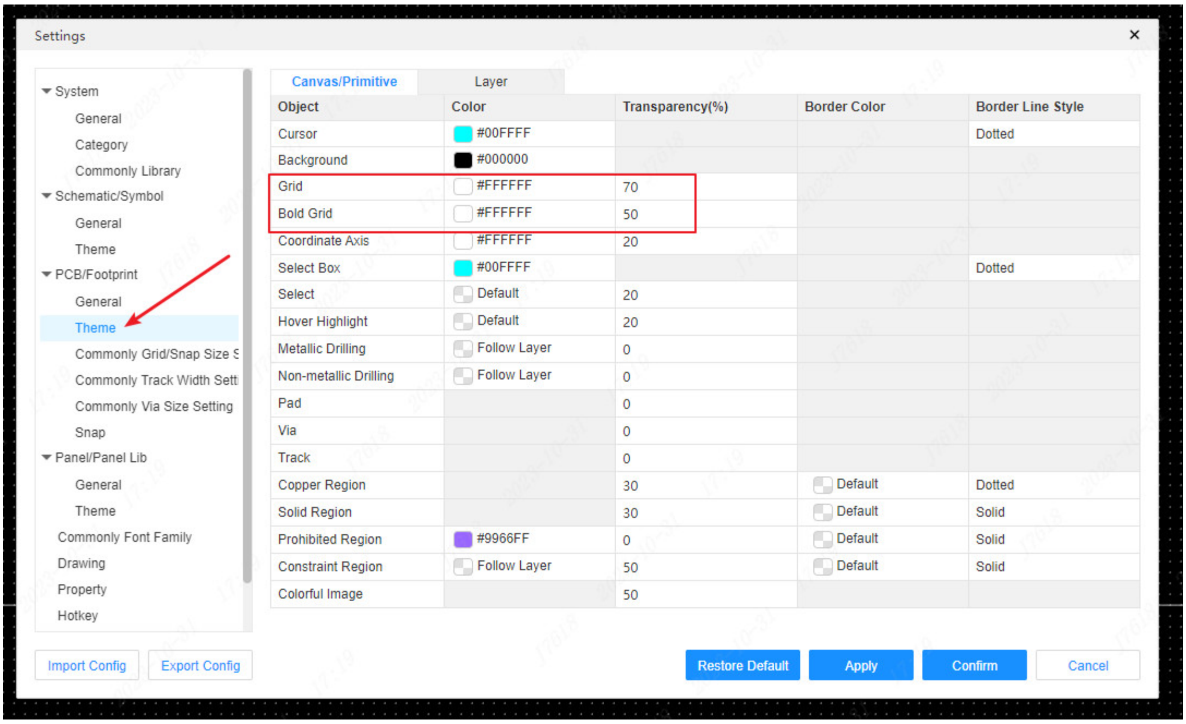
Document

Unit: Switching of the canvas.Switch as the unit of the top toolbar, the unit of the top view menu.

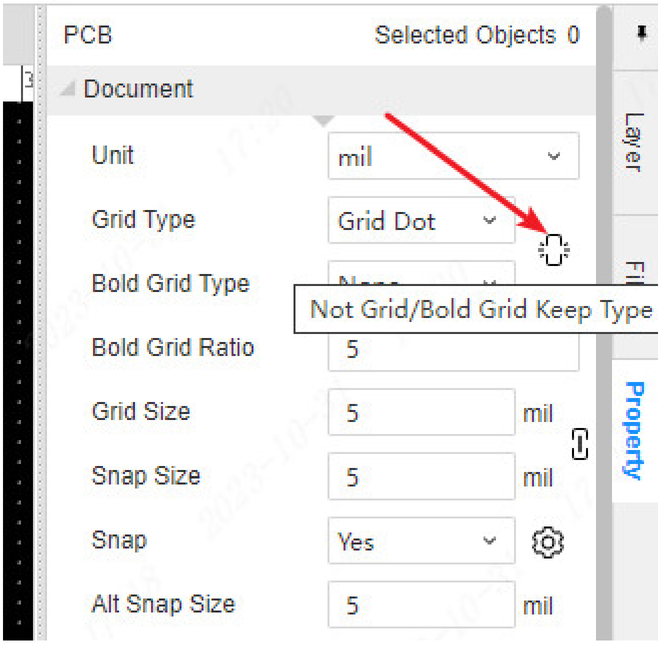
Network type: grid settings of canvas.

Black grid type: grid settings deeper than ordinary grid.

Tips: The color and transparency of the grid and bold mesh can be modified in the PCB theme settings



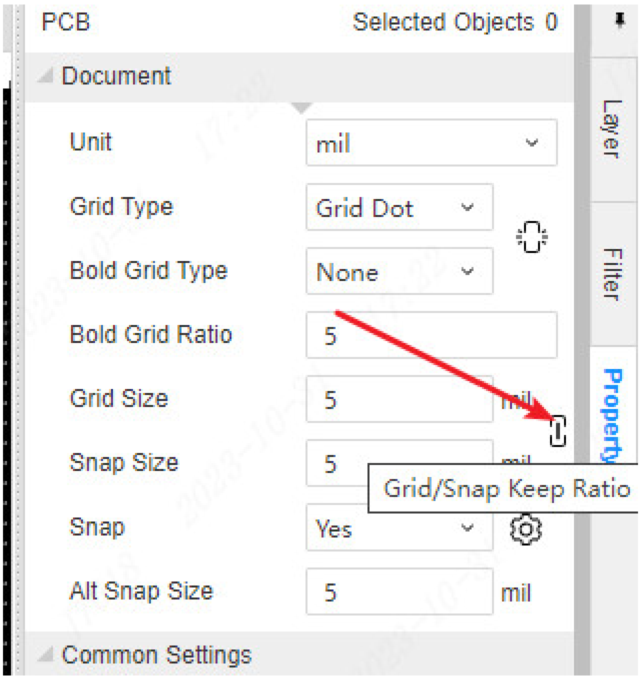
Tips: Click the synchronous icon on the right to keep the network type and thick grid type always switch together.



Grid size: grid size of canvas.You can see the grid.

Snap size: Mobilization point size during adsorption.

Tips:Click the equal ratio icon on the right to keep the grid size and grid size. After modifying, you can always modify it according to the current proportion.



Snap: Whether it is automatically adsorbed.You can set the adsorption object option.

Alt Snap Size: When pressing the ALT key, the adsorption grid size.

Common Settings

Start track width:

- Follow rule: Follow the wire width of the network set in the design rules.
- Custom: The width of the custom starting wire wire will ignore the wire width settings in the design rules.

Start via size:

- Follow rule:Follow the setting size set in the design rules.
- Custom:Customizing the punching size will ignore the perforated size settings inside the design rules.

Routing Mode: corner mode of wire wiring.

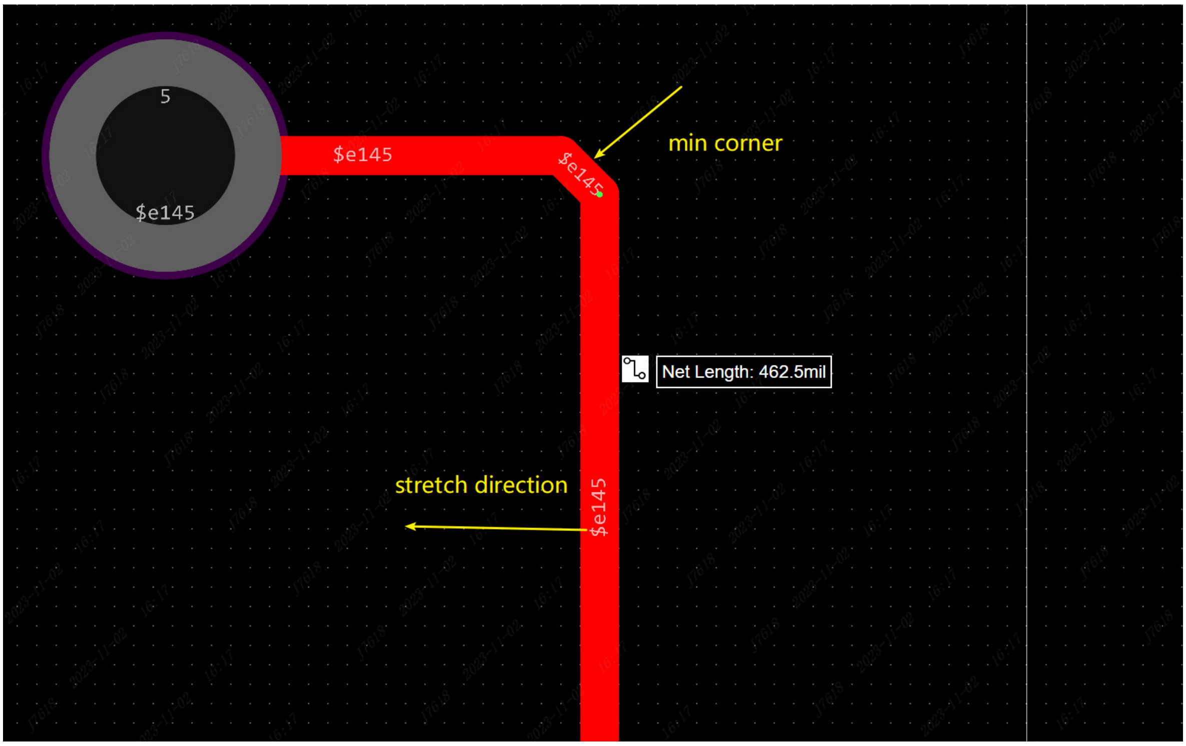
Remove Loop: Whether the wire is automatically removed when the wire is drawn.

Copper Region: Whether to display copper filling in the copper area.

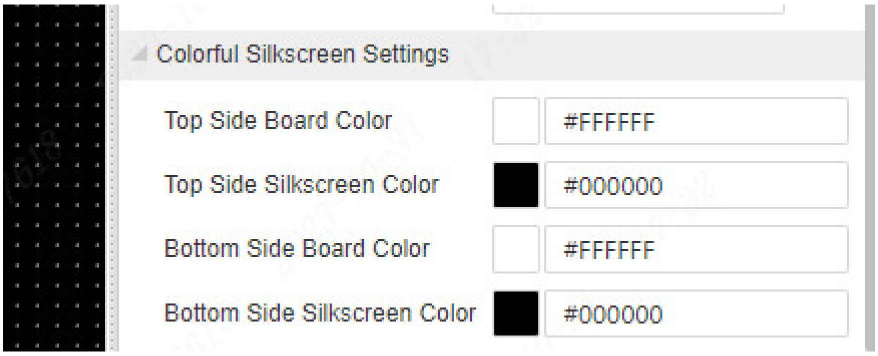
Rotating Objects: Rotate the overall selection; rotate the selected object separately.

Move Footprint The Wire Follows: When the mobile packaging is moved, whether the connected wire automatically extends follows.

Stretch Track Min Corner: When dragging the stretching wire, the adjacent wire reaches the smallest corner, and it will not continue to be stretched.



Colorful Silkscreen Settings



Top side board color: Modify the background color of the top surface, which will take effect on the PCB of the color wire printing process.

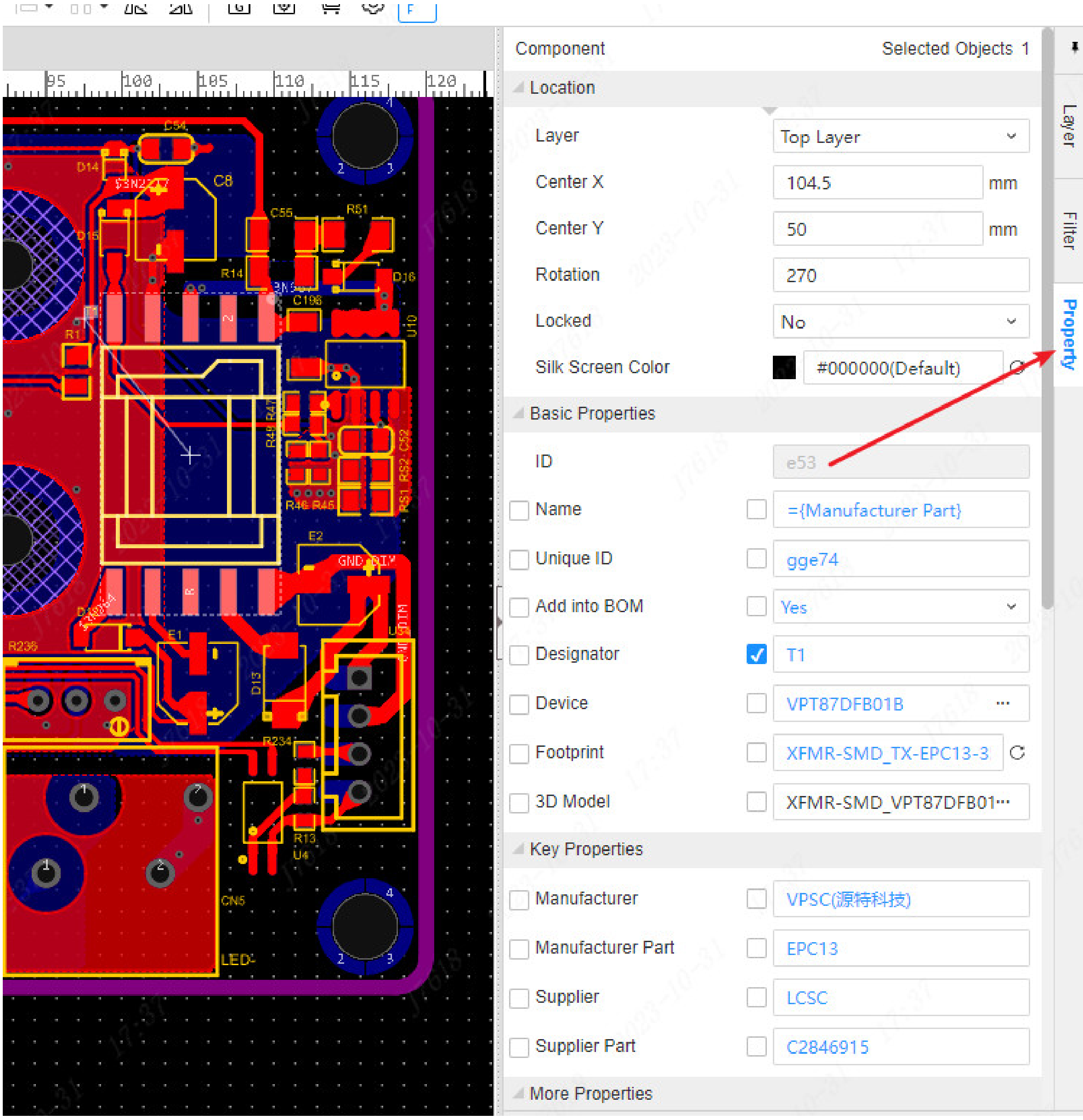
Top side silkscreen color: modify all the silk printing colors that are not set alone on the top surface, which will take effect on the PCB of color silk printing

Bottom side board color: modify the background color of the bottom surface, the PCB of the color silk printing process will take effect

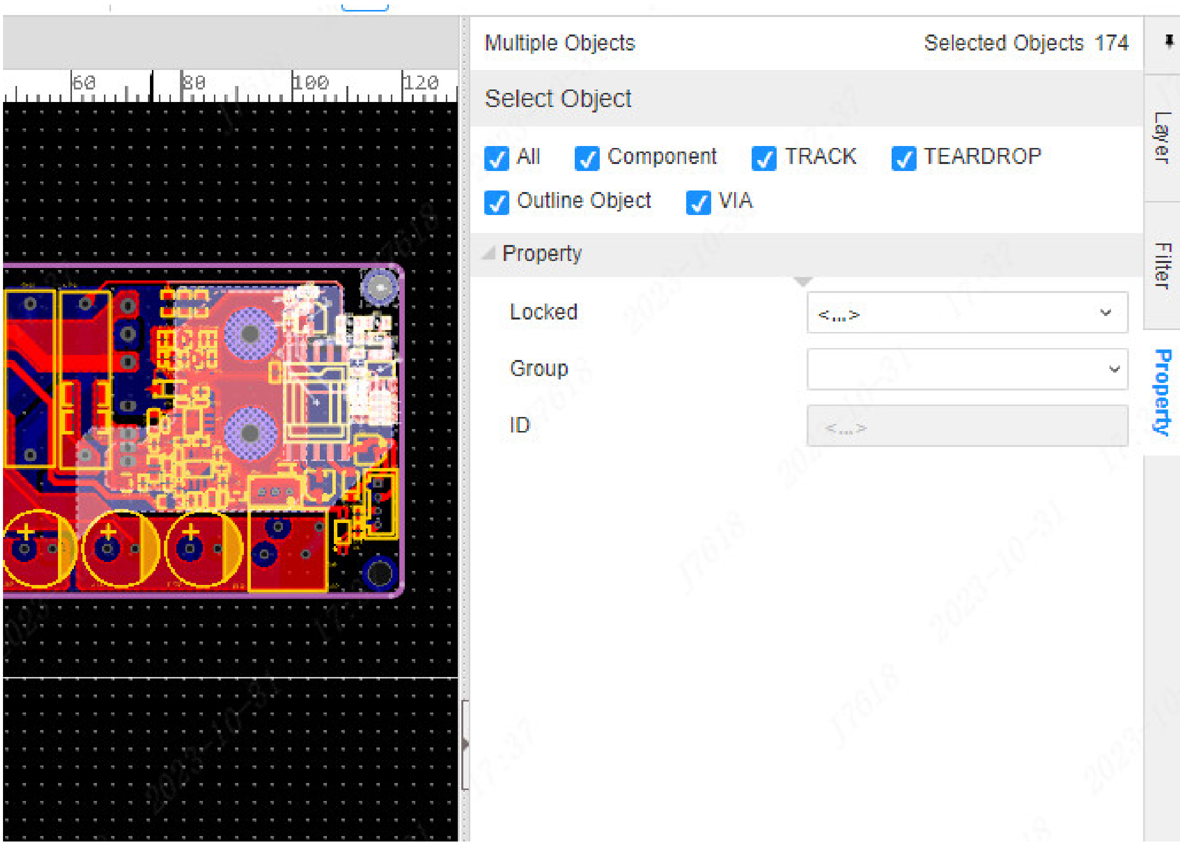
Bottom side silkscreen color: modify all the silk print color of all the color that has not been set separately, which will take effect on the PCB of color silk printing

Object attribute

When you click on one or more objects in the canvas, you can modify the attributes of the device or other elements.

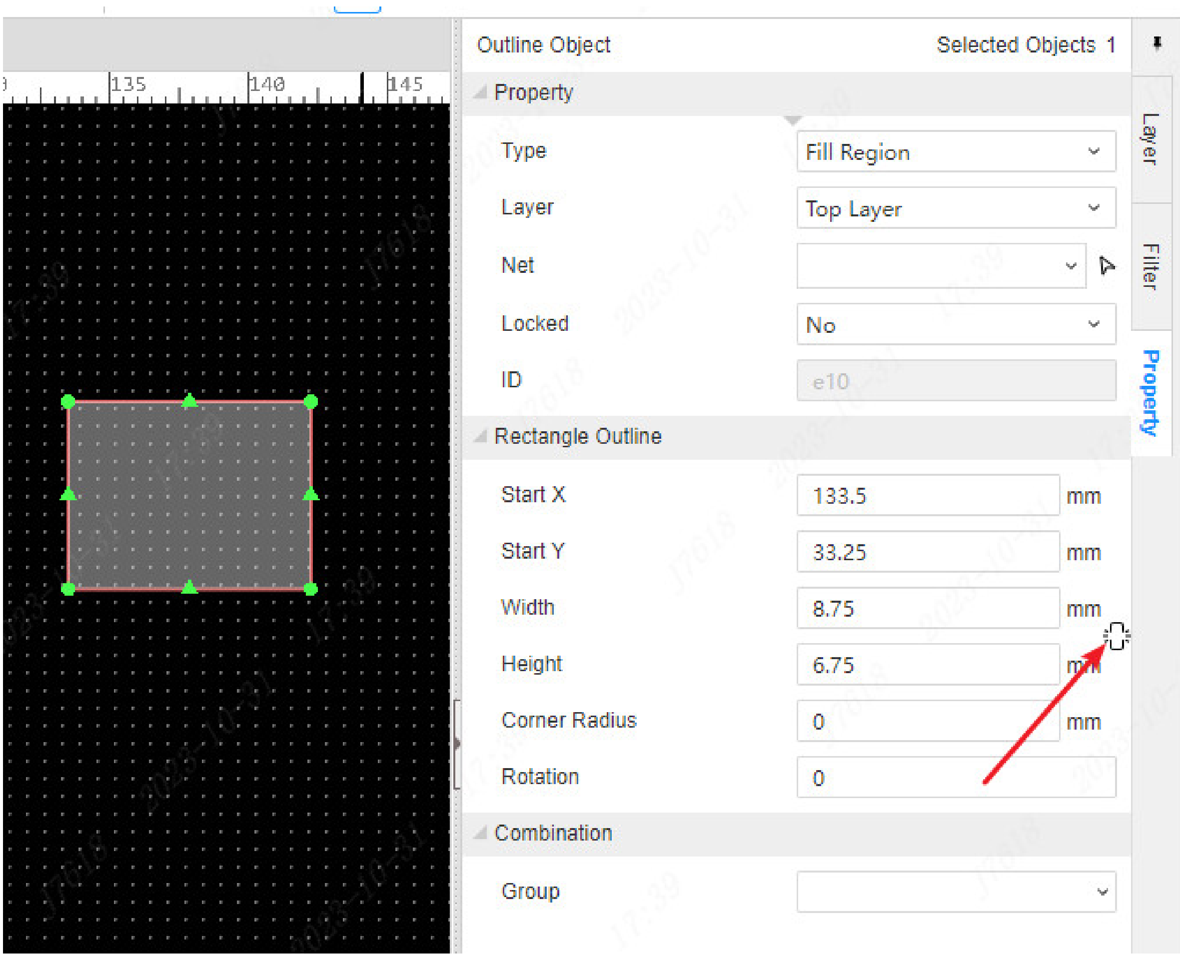


When selecting multiple objects, the attribute panel supports the same attributes of the aggregation of different objects by filtering, which is convenient to quickly modify the same attributes.

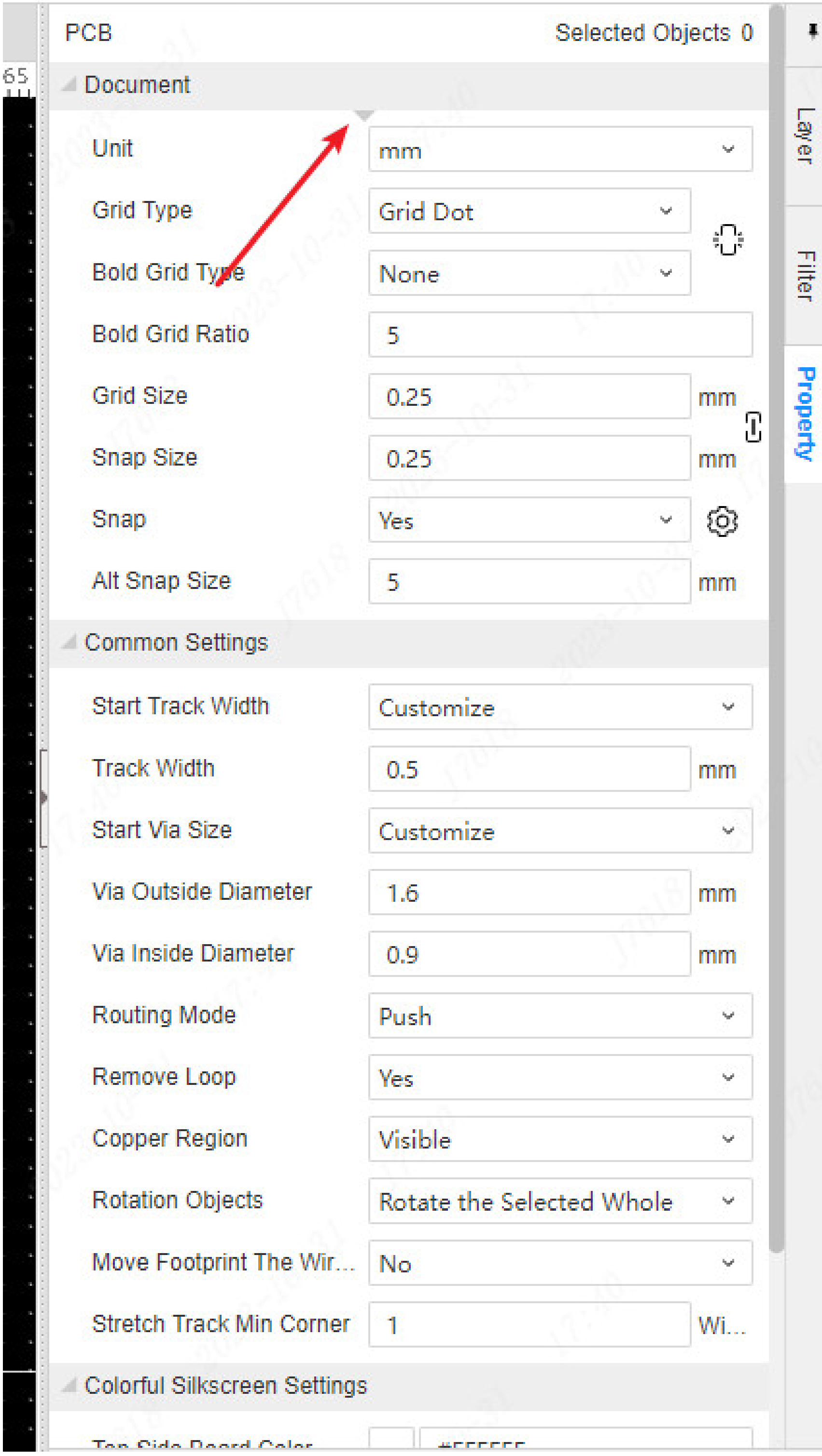


Tips:

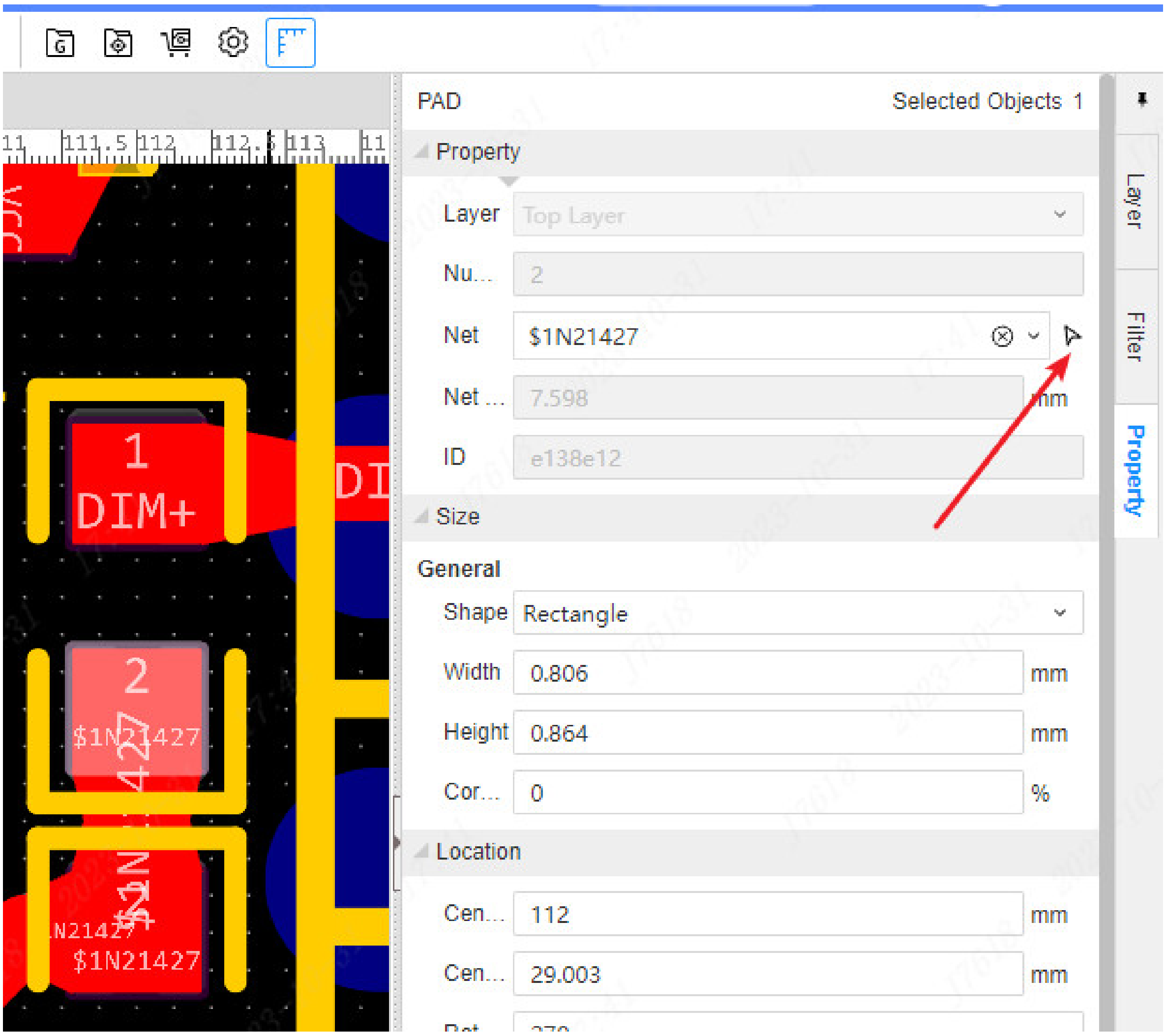
1. With a wide height, when the attribute panel is modified, it can be adjusted whether it is adjusted.After clicking the equal proportion icon to lock the height, when the width is adjusted, the height will change together.



2. Drag attribute width adjustment button, which can modify the width ratio of the attribute name and attribute value of the attribute panel, and it will be recorded in personal preference



3. When setting the map of the network, click the "Click to select the Network" button to quickly click the cloth chart element to set the network name

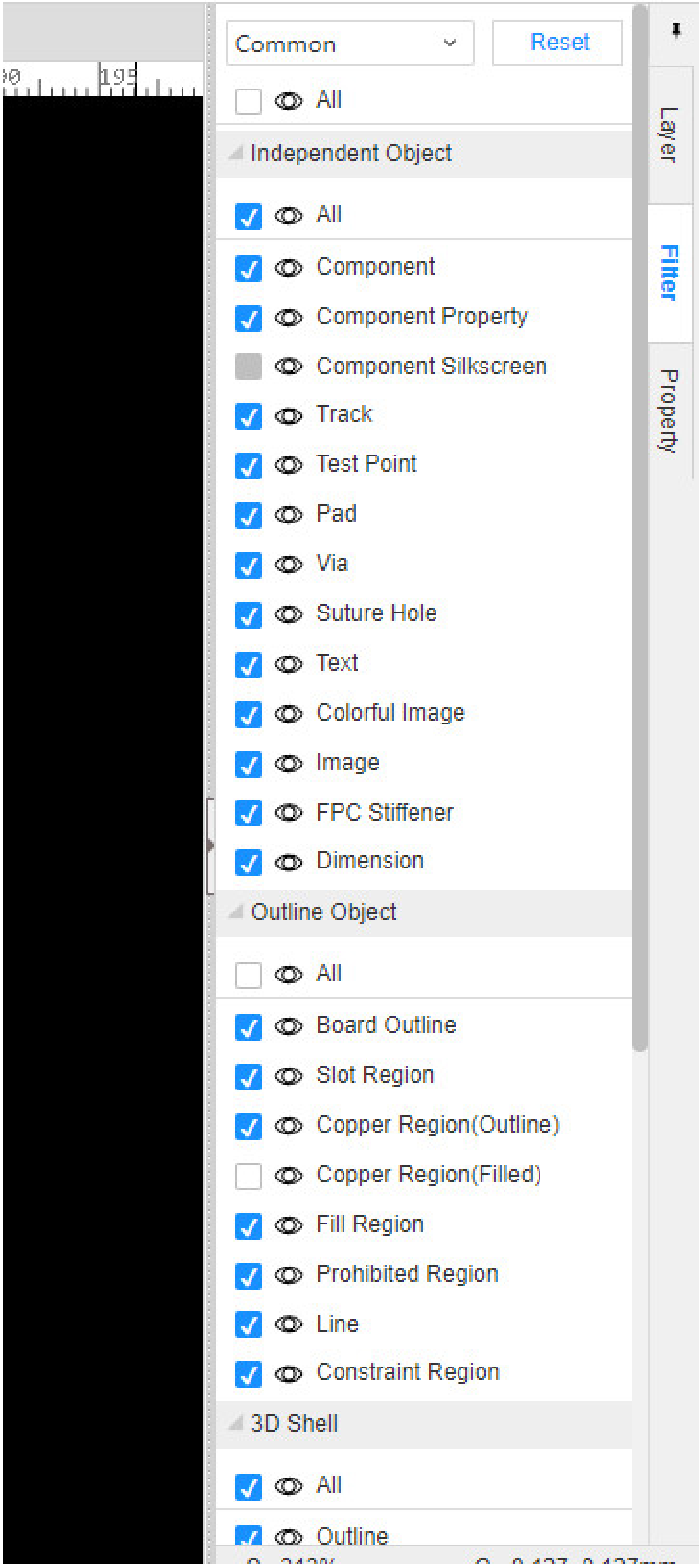


Right Panel - Filter

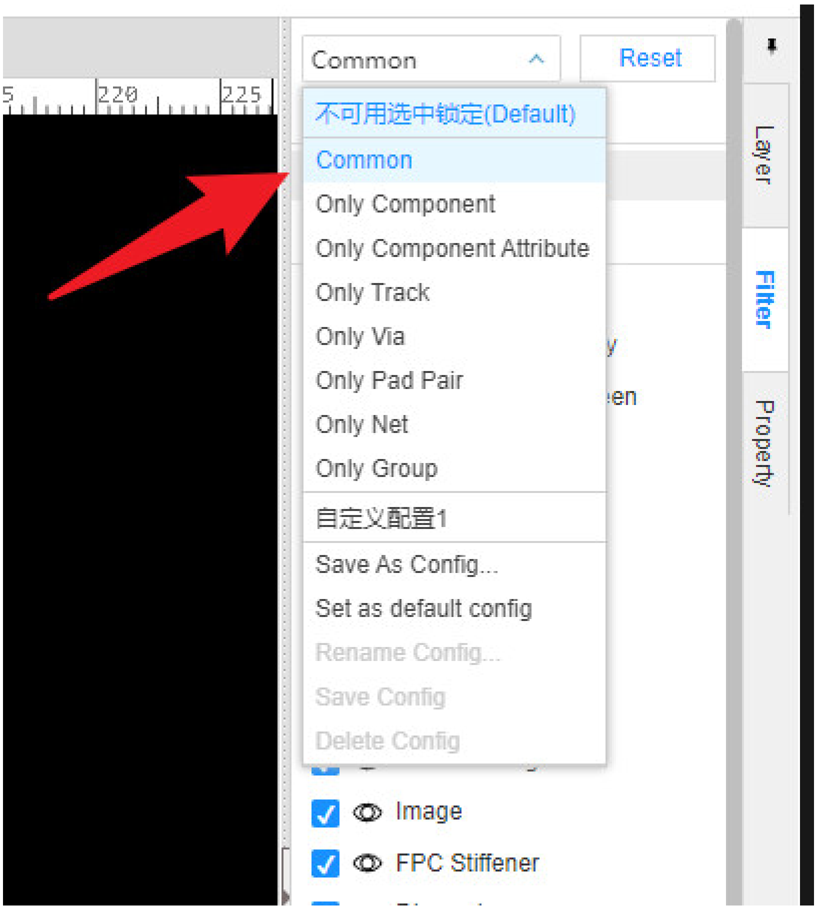
Filtering

The element that needs to be selected on the right panel filter on the PCB interface

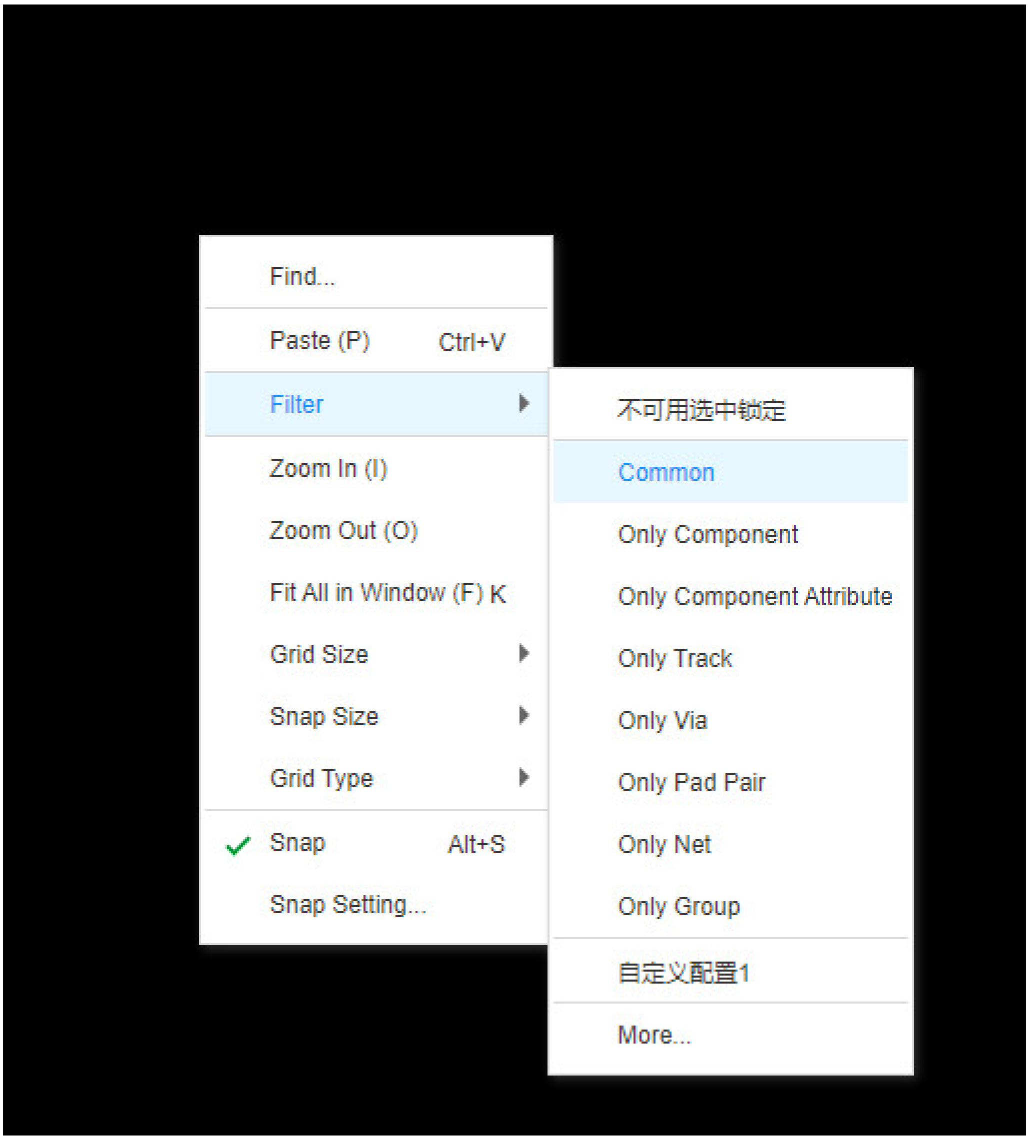
After canceling one of the elements, the corresponding element will not be picked up by the mouse in the canvas, but can be selected in batches through shortcut keys.



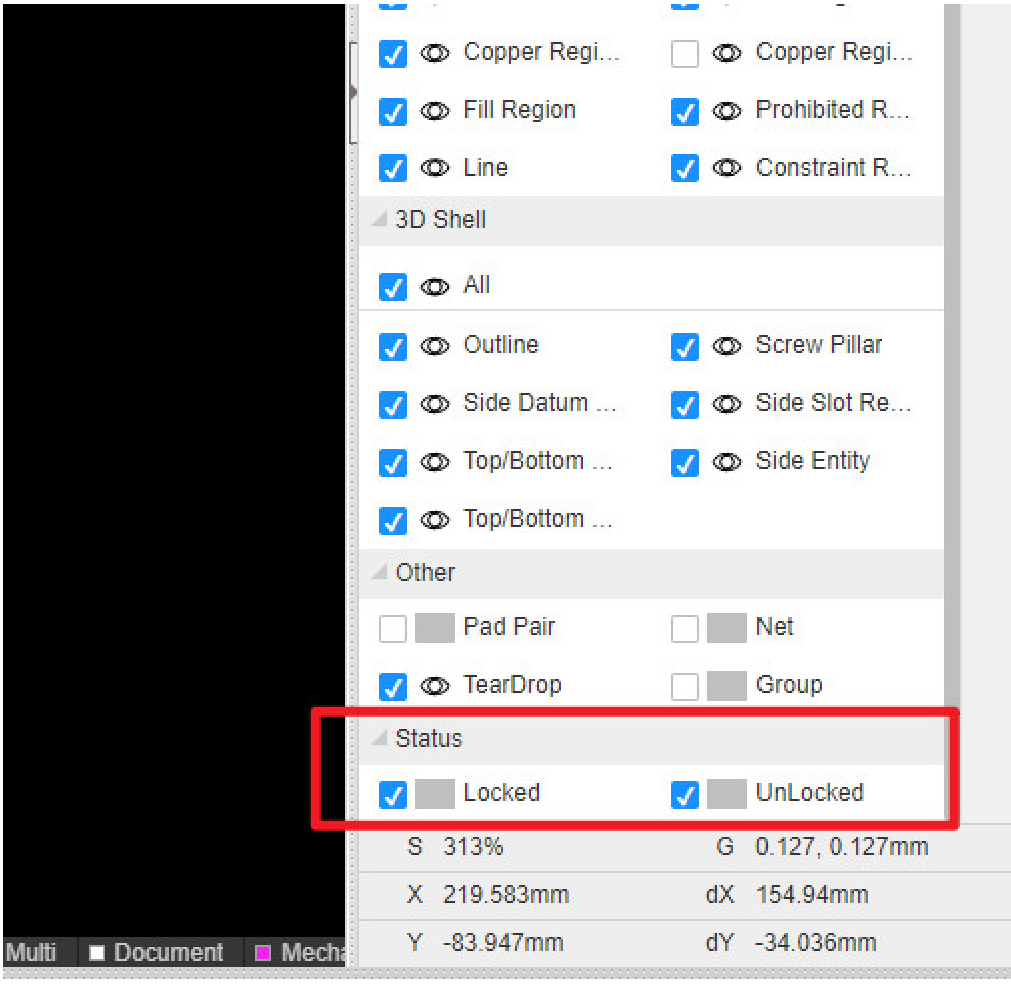
You can also quickly switch the built -in filtration type, and you can save your own filtration preference



The filtering function in the right -click menu is the same as the configuration of the filter directly in the filter panel, which can quickly switch the filtering configuration



It also supports filtering according to the lock state.

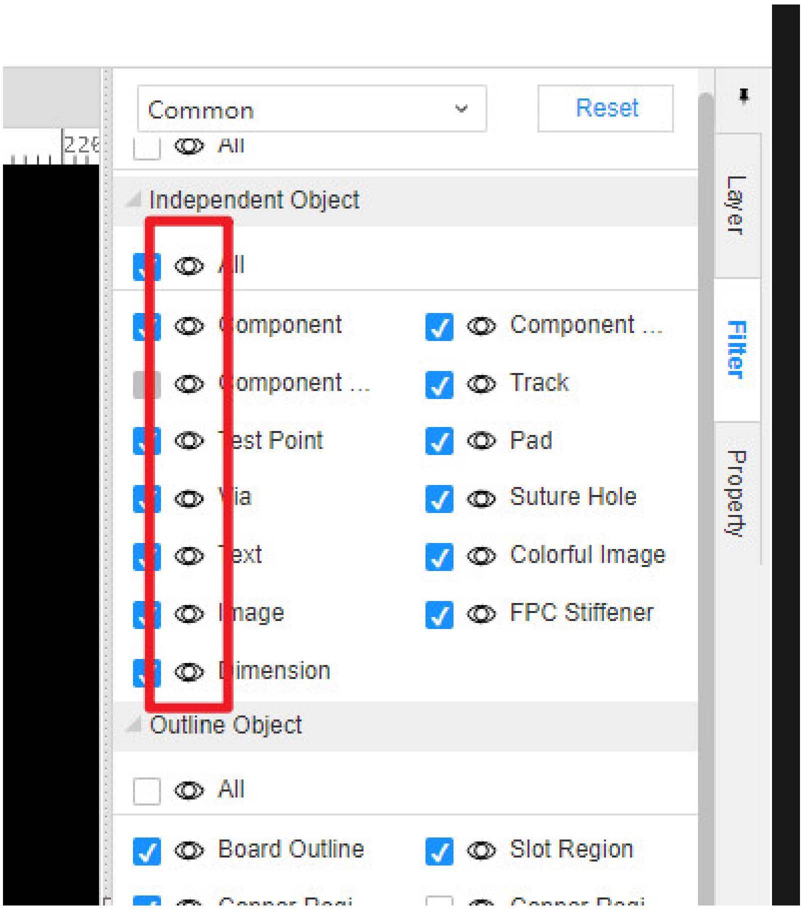


Other types of filtering:

- Pad Pair: After checking, click a pad or wire of the picture cloth to select the two pads connected to the wire.
- Net: After checking, click a network of dollars with a network, all the maps of this network will be selected. After checking, you can also click an empty network.
- TearDrop: tears can be selected in the mouse
- Group: After checking, you can select the entire combination for easy movement and pick up the combination.

Show hide

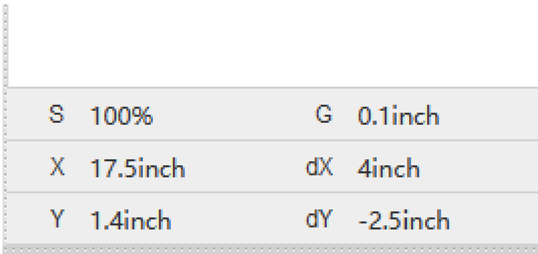
Click to modify the display and hiding of the corresponding elements in batches. Click the eye icon in front of the object name to achieve batch hidden objects.



Note: The hidden display of the layer shows that the hidden display of element screening will not enter the historical record of revocation.

Right Panel - Status

After opening the document, the area at the lower right corner of the right panel displays the current canvas and cursor status.



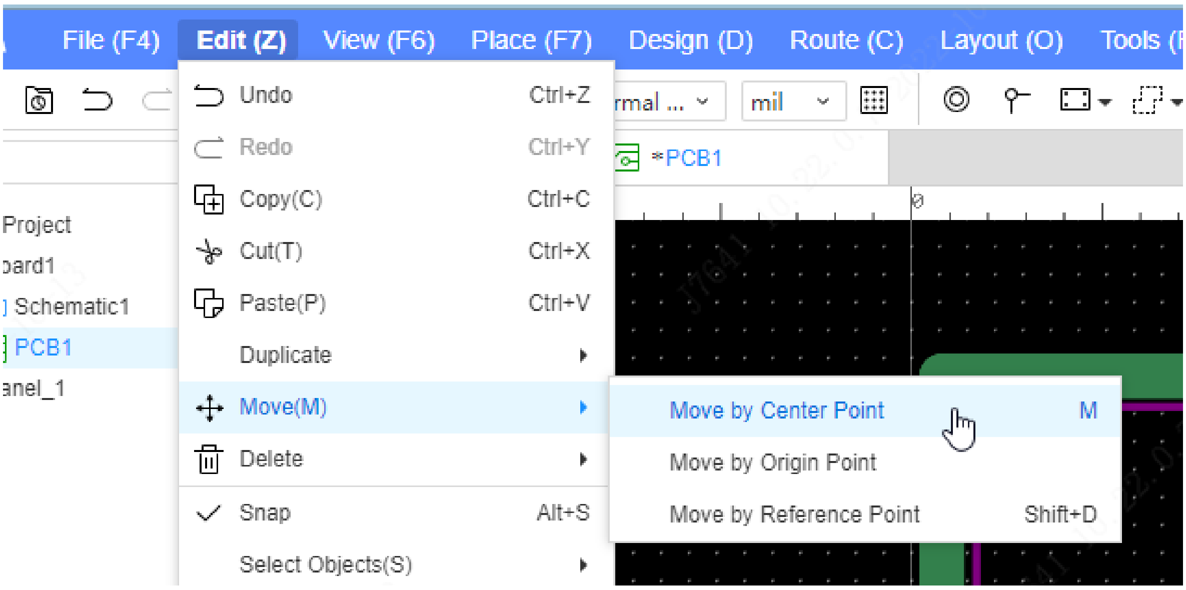
Parameter Description:
S (Scale): Canvas zoom factor;
G (Grid): Canvas grid point X, Y size, the schematic has only one value;
X (X): Cursor current coordinate X;
Y (Y): Cursor current coordinate Y;
dX (Distance X): the distance that the current cursor moves on the X axis;
dY (Distance Y): The distance that the current cursor moves on the Y axis.

Move

Move the selected element.

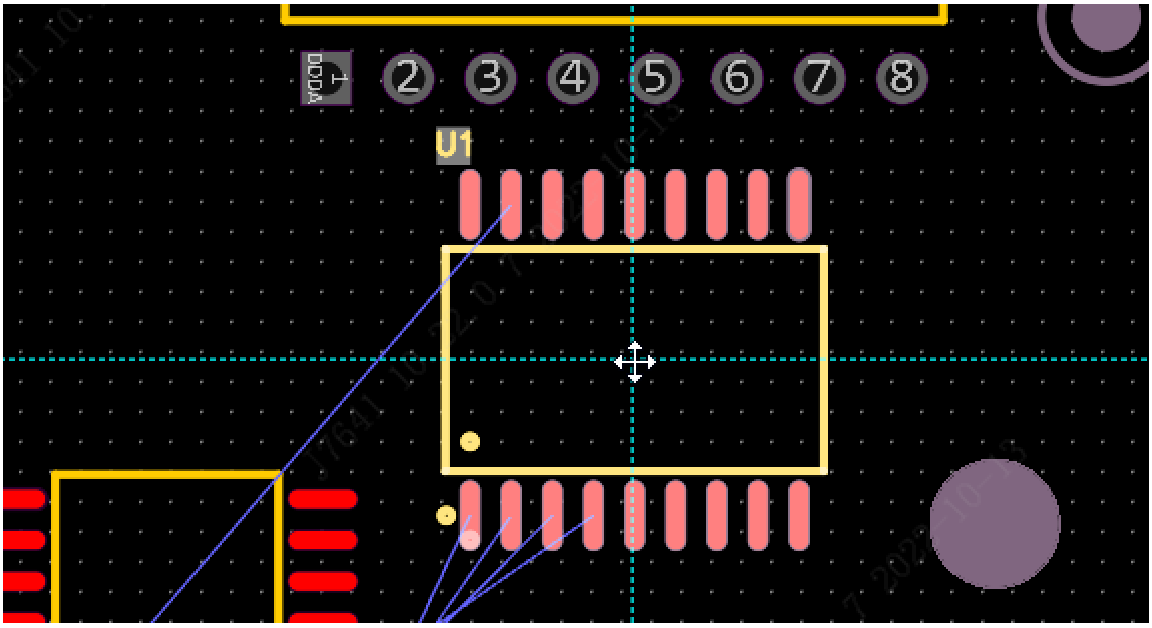
Instructions:

- Select object Top Menu - Edit - Move or select object right click



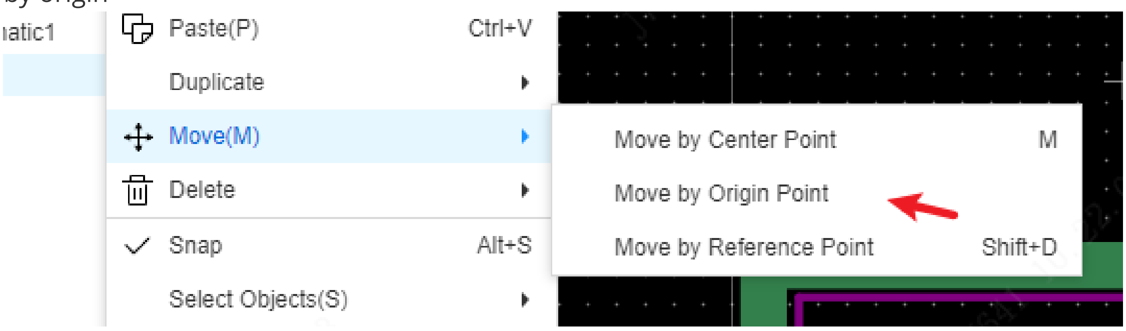
Move by Center Point

1. Select object - Top Menu - Edit - Move - Move by Center Point
2. Select the object - click the right mouse button - move - move according to the center
3. Shortcut key "M"

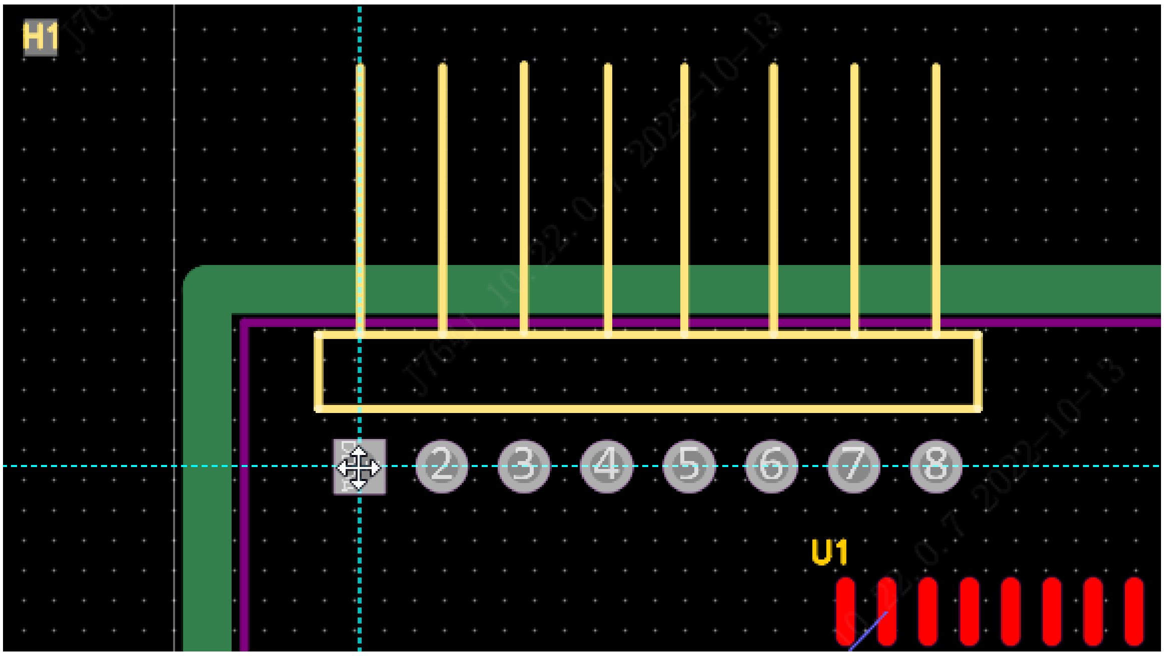


Move by Origin Point

- select object - Top Menu - edit - move - move by origin or right mouse button - move - move by origin



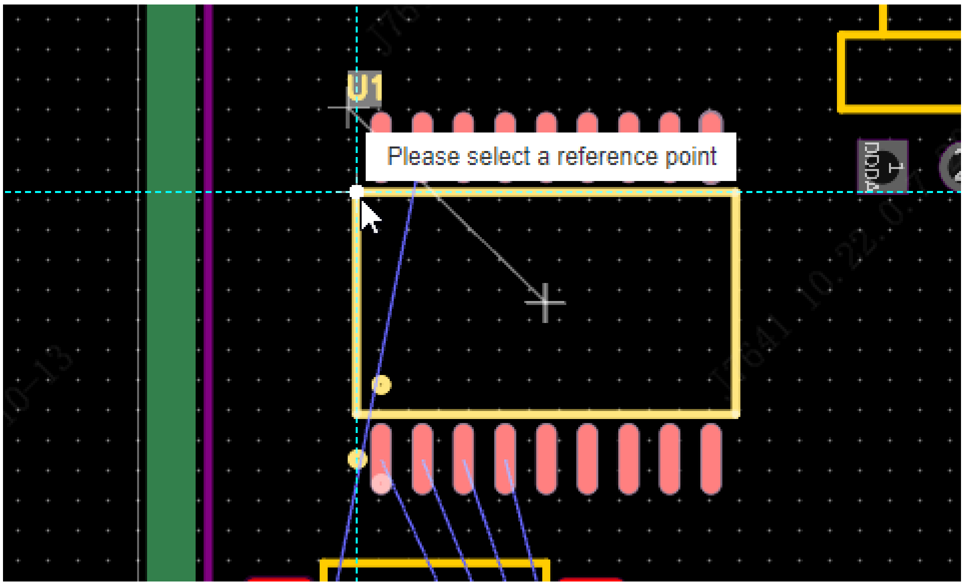
After selection, the object can be dragged according to the object origin, such as footprint origin

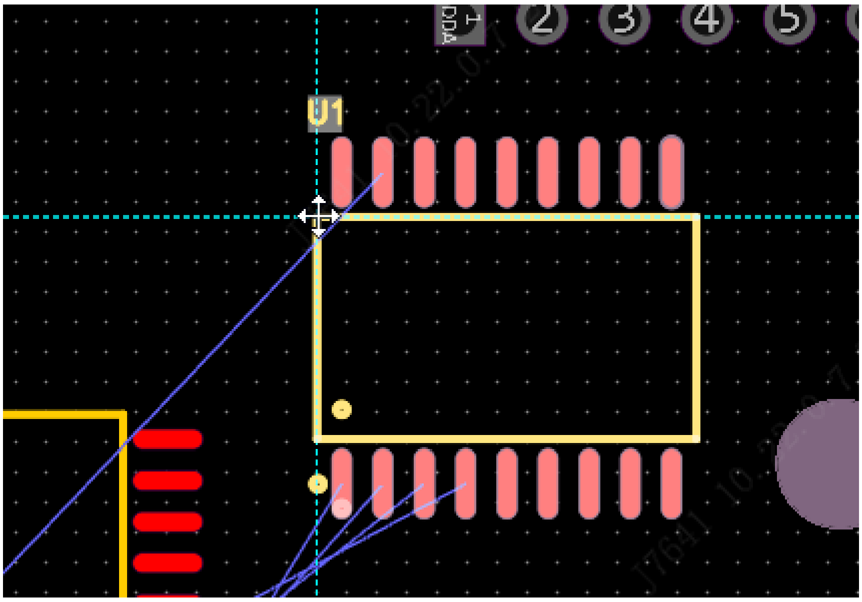


Move by Reference Point

Move the selected object according to the reference point selected by the user, click and select a reference point as the moving reference point to drag.

- select object - Top Menu - move - move by reference point or right mouse button - move - move by reference point



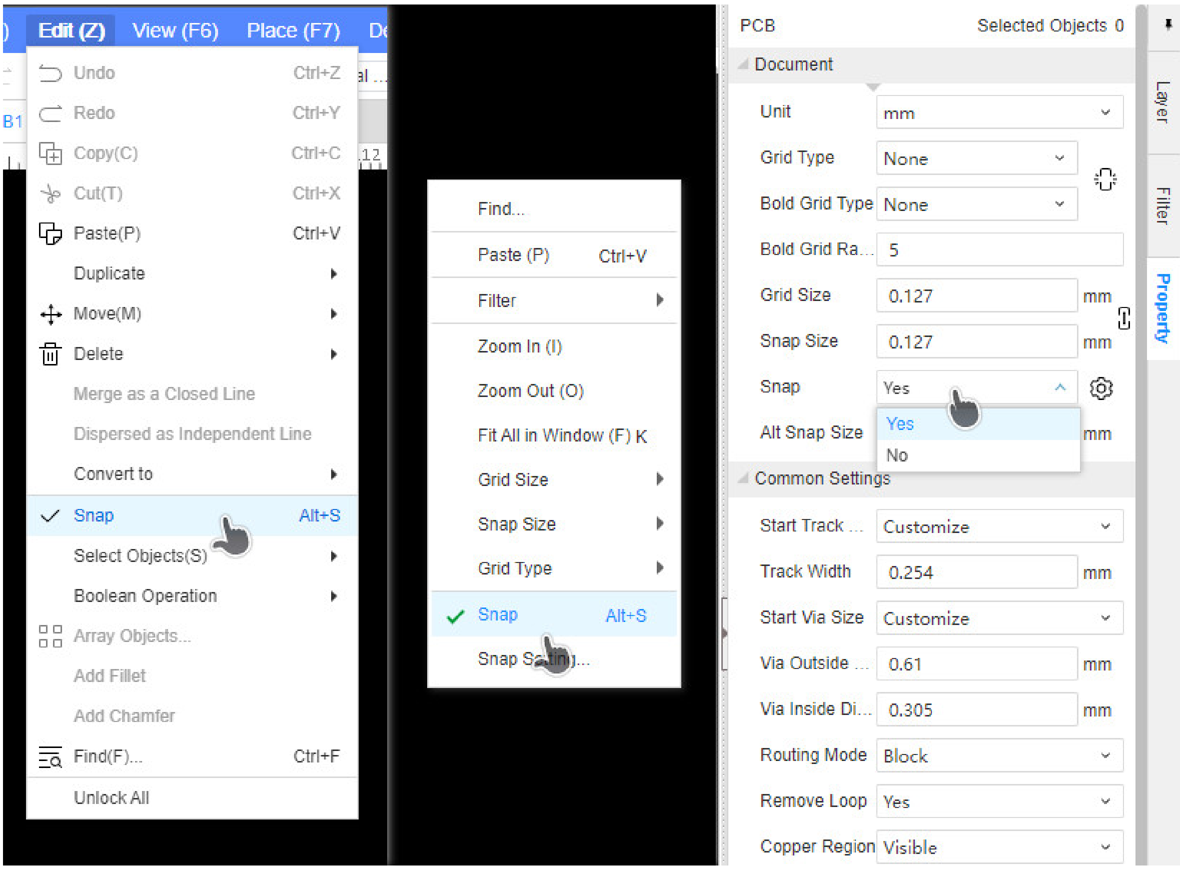


Snap

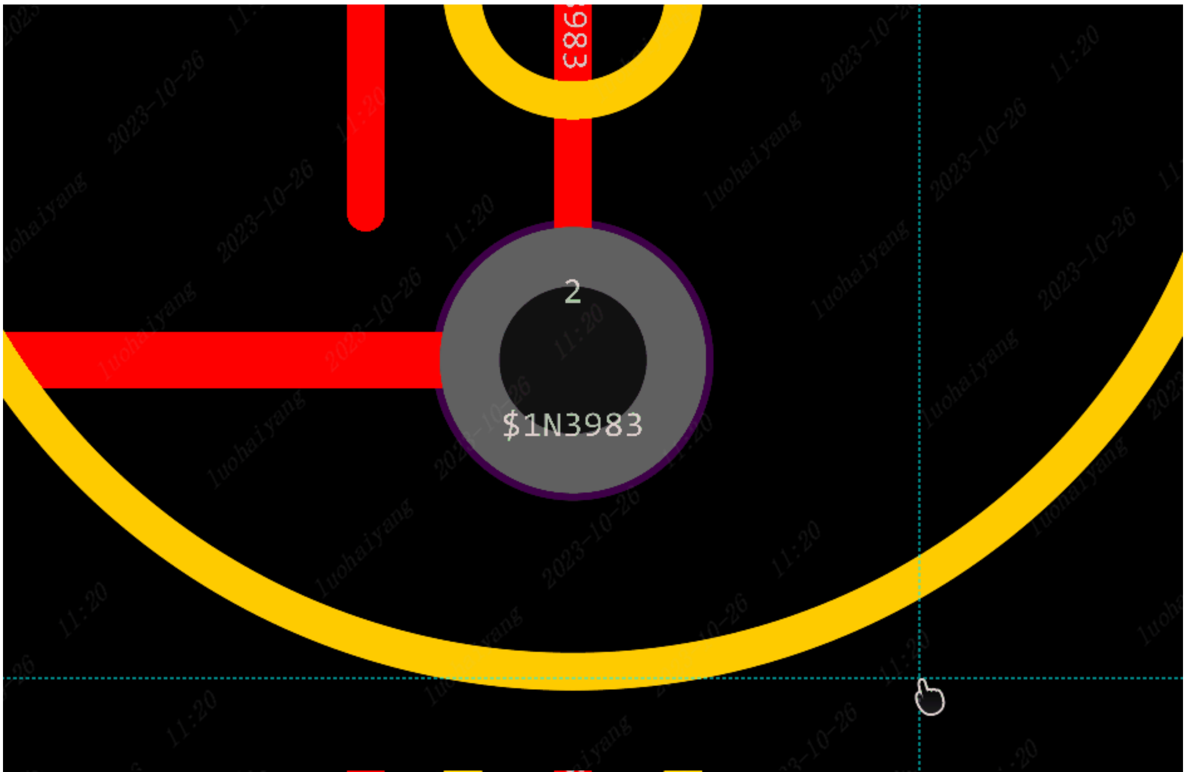
EasyEDA Pro provides users with powerful adsorption function, which supports adsorption to certain specific positions of existing elements on the canvas during wiring, measurement, drawing and other operations, making it convenient for users to quickly align and accelerate design efficiency. At the same time, the range and position of adsorption support can be customized by users, which can adapt to different user habits

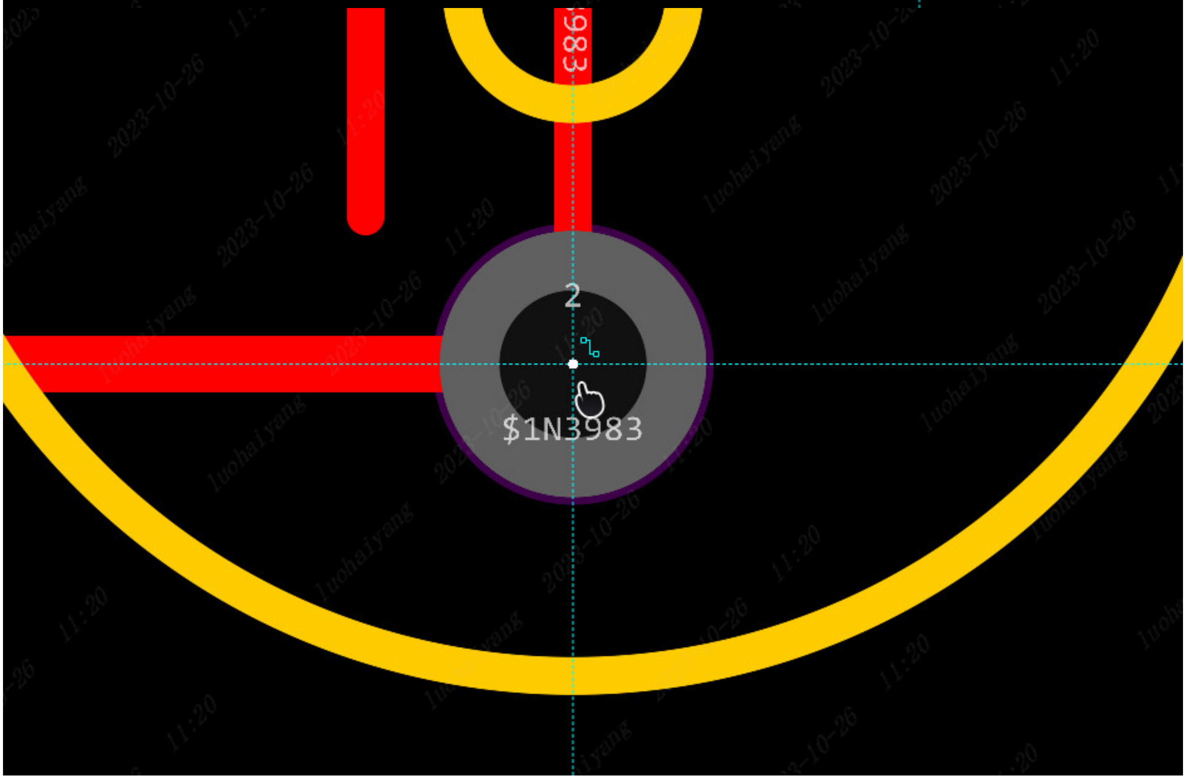
Instructions:

- Top menu - Edit - Snap
- Right-click - Snap
- PCB right-side panel - Snap
- Default shortcut key `Alt` + `S`

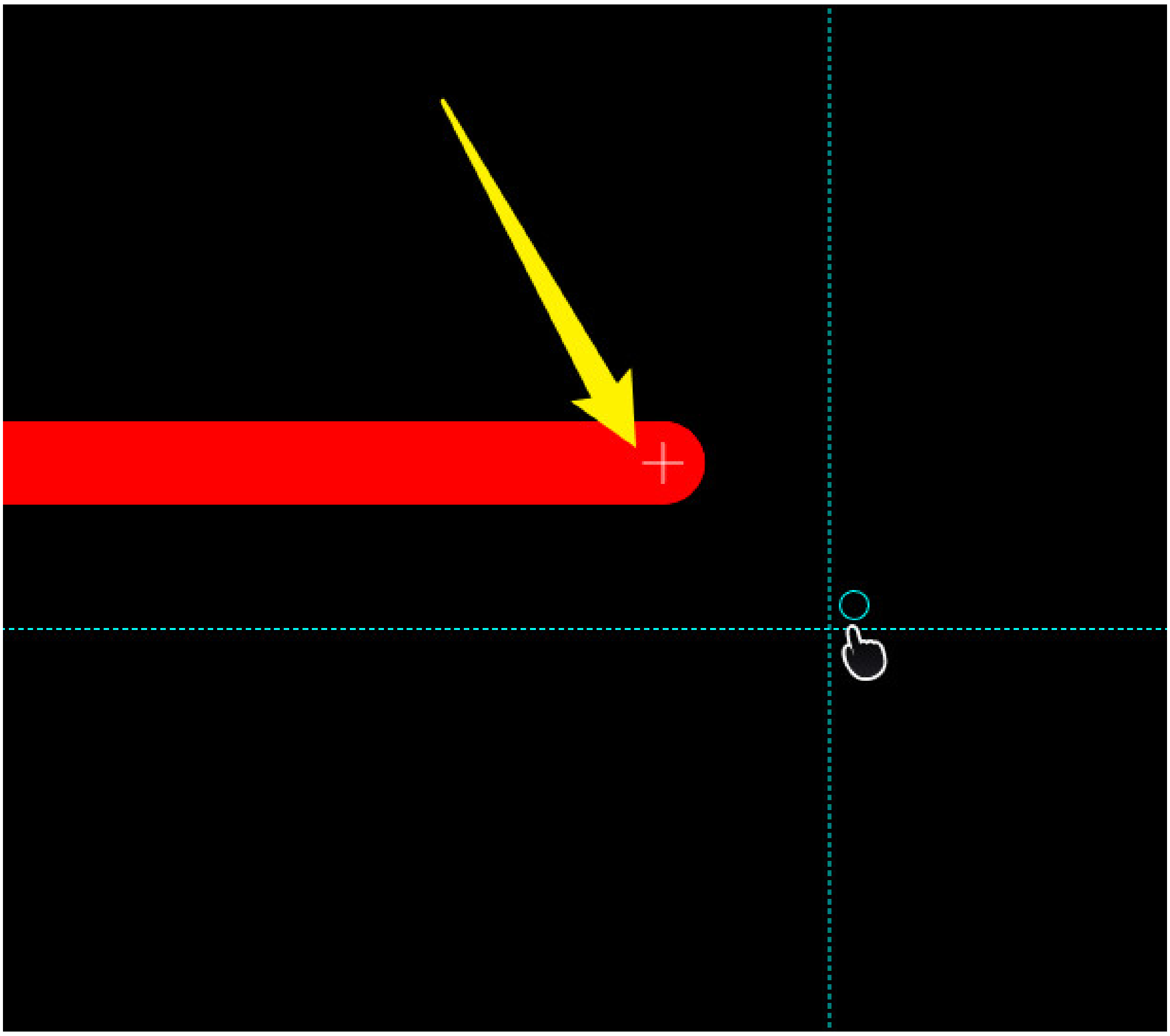


After opening the adsorption, move the mouse to the adsorption range, and it will automatically adsorb, and a white dot will be displayed at the adsorption position



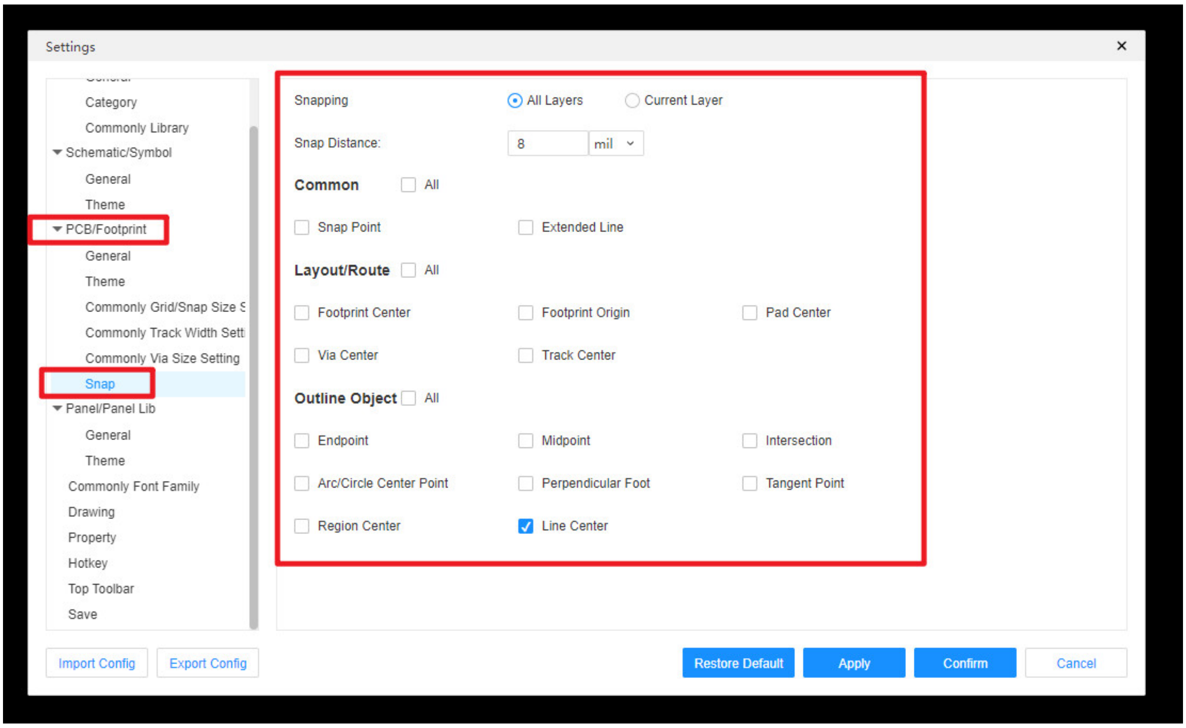


When performing some operations other than drawing wires, if some positions on the canvas are adsorbed, a white cross adsorption mark will also be generated, up to six of which can exist on the canvas at the same time. This mark can be used for operations such as extending the line adsorption. Scaling the canvas or exiting the current operation will clear all marks



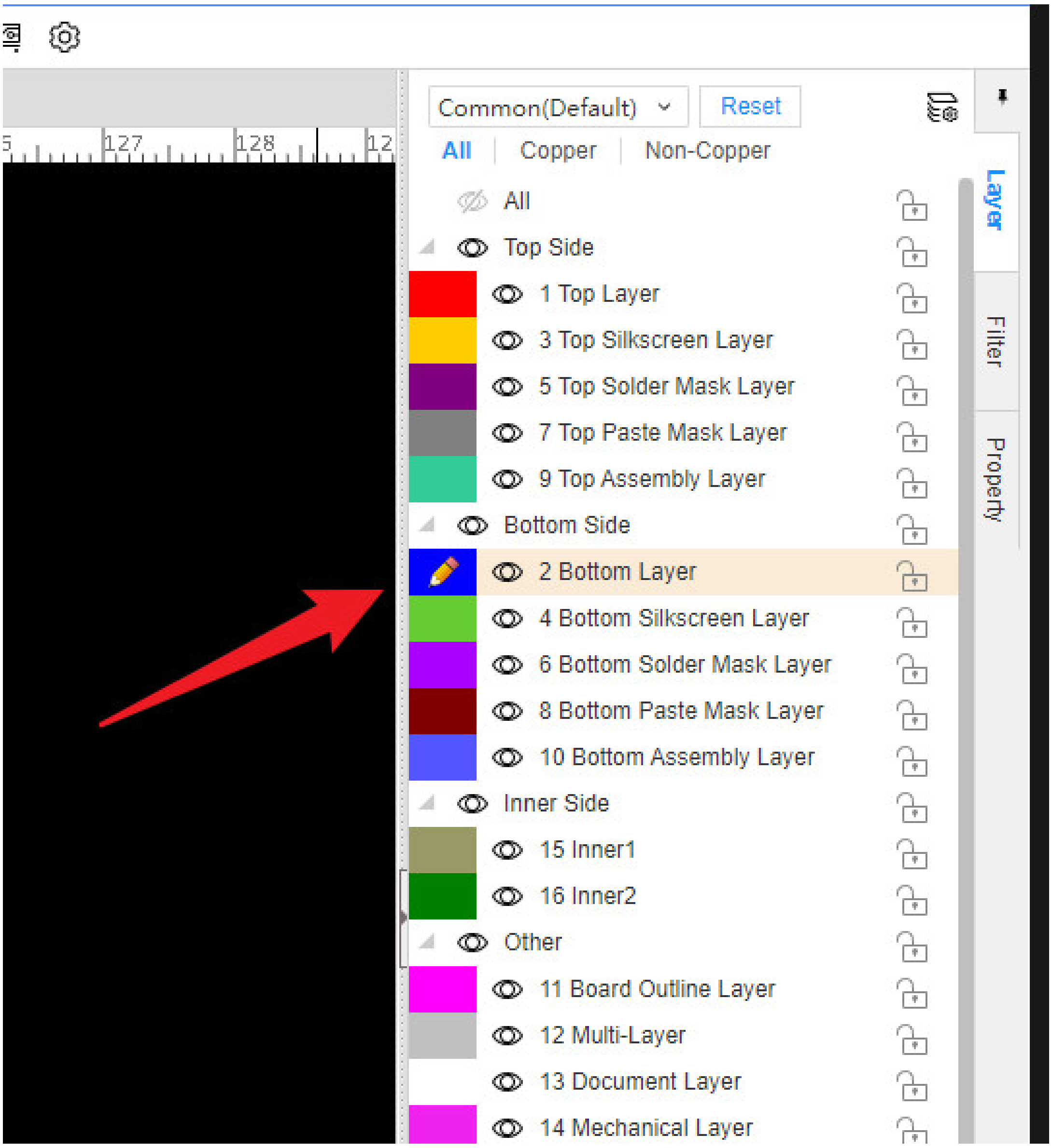
Adsorption setting

We can configure the desired adsorption distance and adsorption position in the adsorption settings.



Adsorption layer configuration

It can be set to **adsorb all layers** or **only the current active layer** . When it is set to only the current active layer (where the pencil icon is is the current active layer), it will not adsorb the positions on the elements of other layers. Selecting all layers is the opposite. **Wiring is not affected by this configuration** , only the current active layer will be adsorbed



Adsorption distance

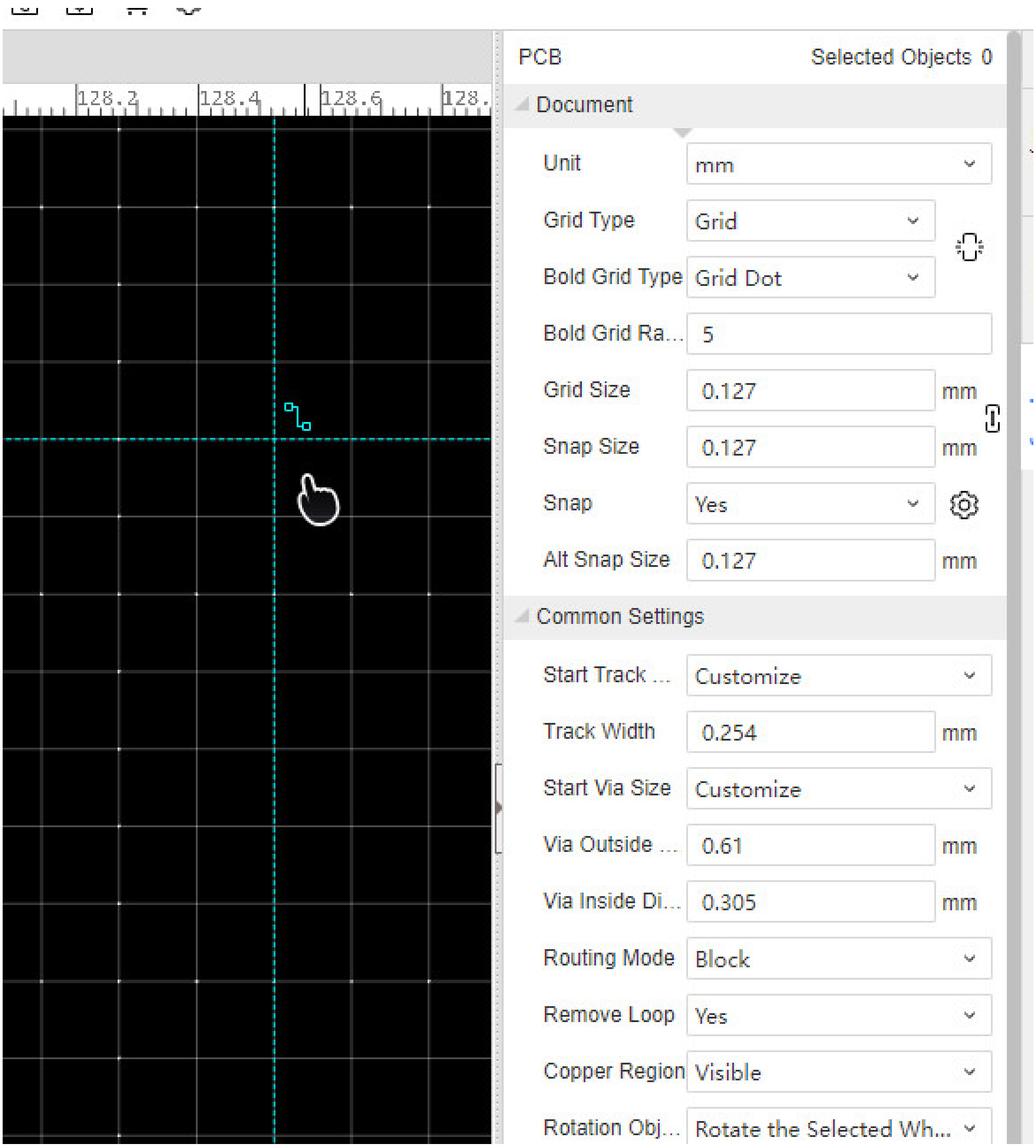
The adsorption distance can be set at this position. Note: During wiring, adsorption will only begin in the pad or via. Therefore, even if the mouse is already within the adsorption range during wiring, if it is not inside the above graphic element, it will not produce adsorption effect.

General purpose

In **all** the check boxes on the right side, you can check or uncheck the grid points and extension lines at the same time, the same below

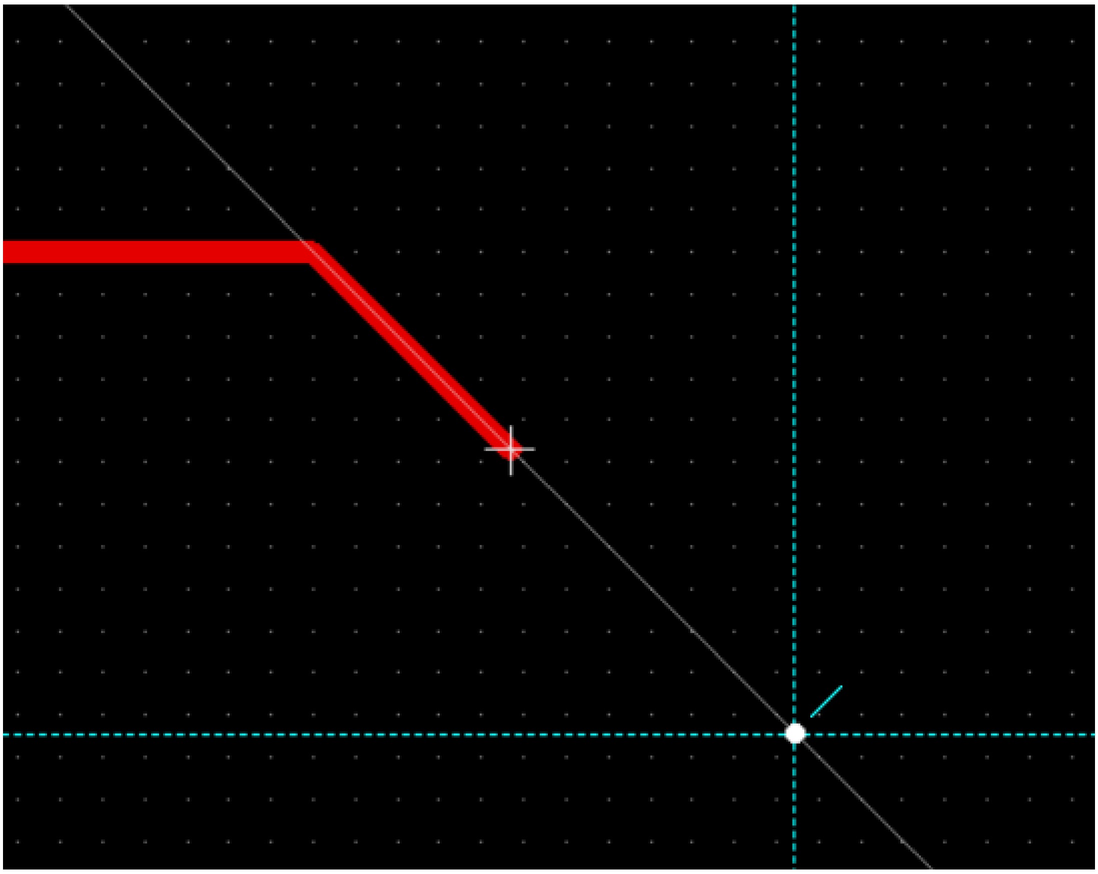
Grid points

After checking, the cursor will be attached to the grid points on the canvas, and the white points will not be displayed when attached to the grid



Extension cord

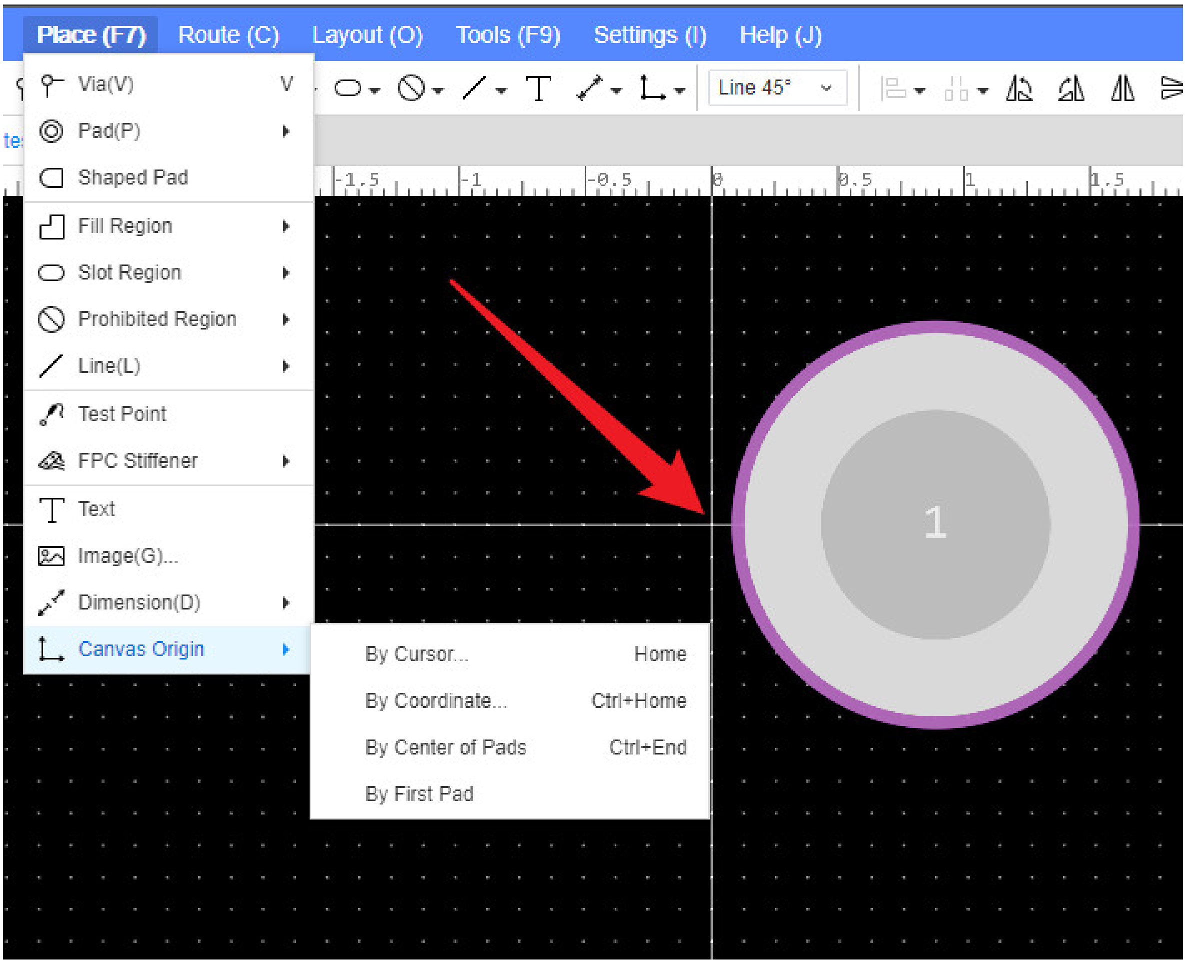
When drawing lines, using the movement function to move elements, adjusting the outline object shape and other operations. Some cross adsorption marks will be generated. When moving the mouse to the direction of the line extension or orthogonal position, it can be adsorbed on the extension line for easy alignment



Layout wiring

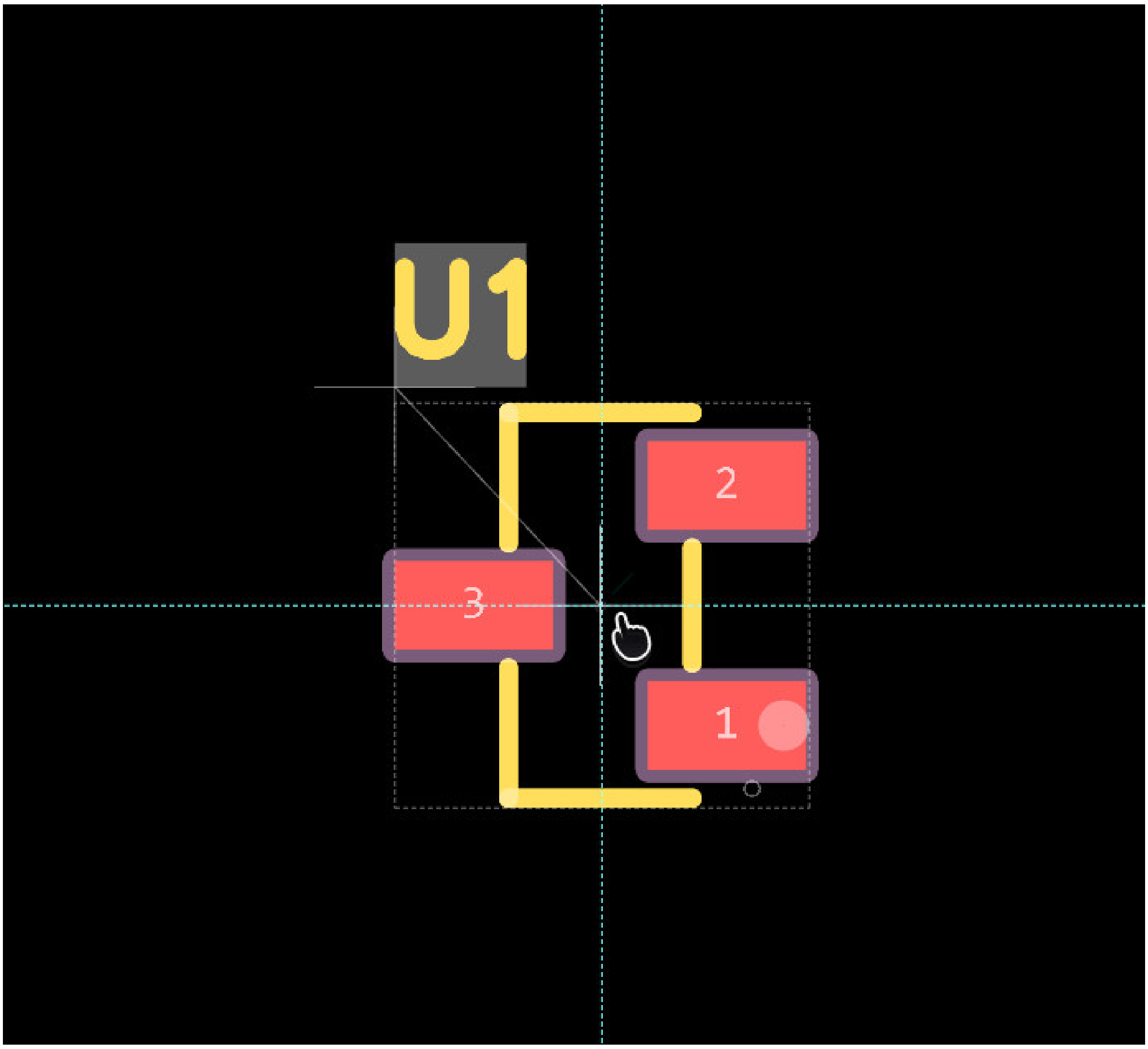
Encapsulation origin

After checking, it can be adsorbed to the origin of the package. The origin of the package is the center position of the cross in the package, which can be modified by placing the menu when drawing the package



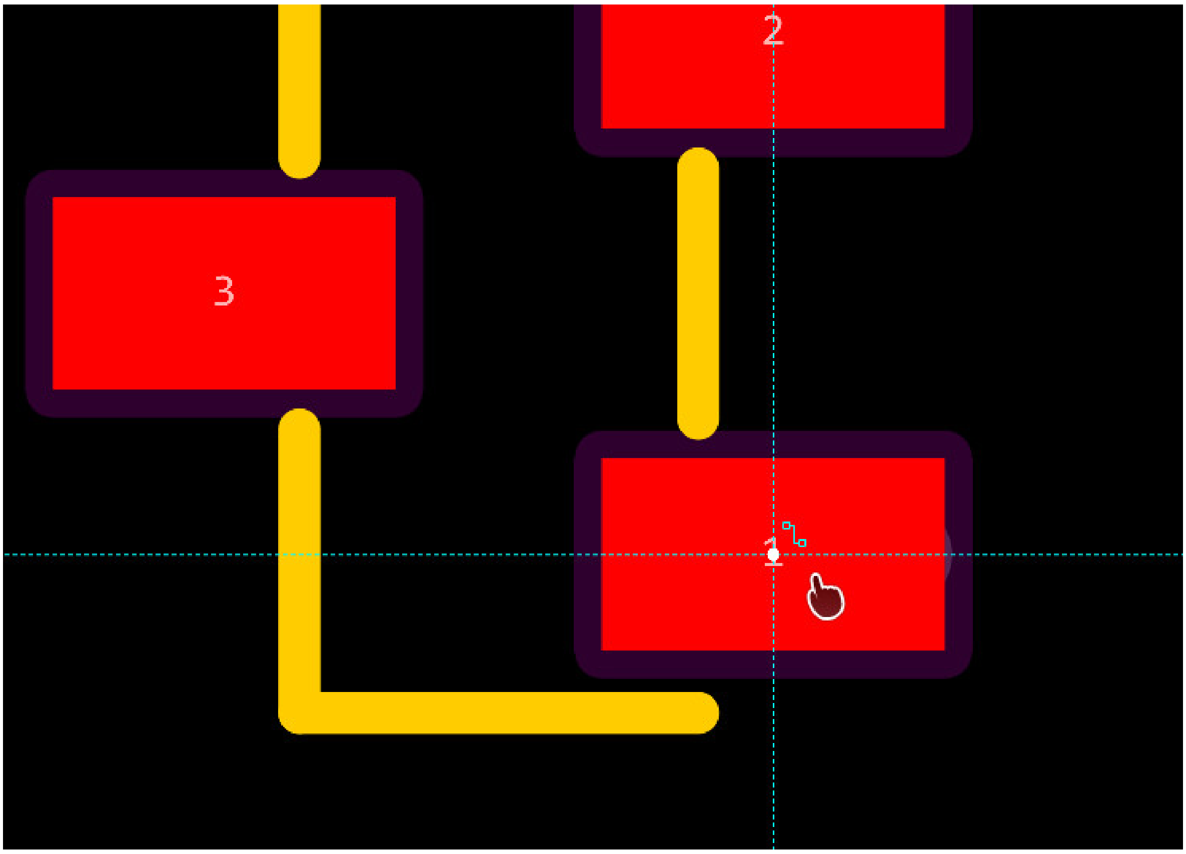
Packaging center

After checking, it can be attached to the center point of the entire encapsulation graphic. Only one of the encapsulation center and encapsulation origin options can be selected at the same time



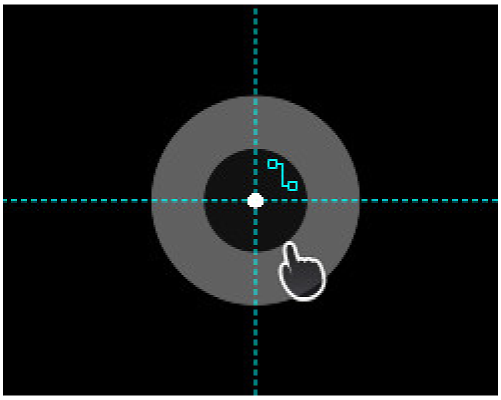
Center of solder pad

Check the center point of the pad pattern that can be adsorbed



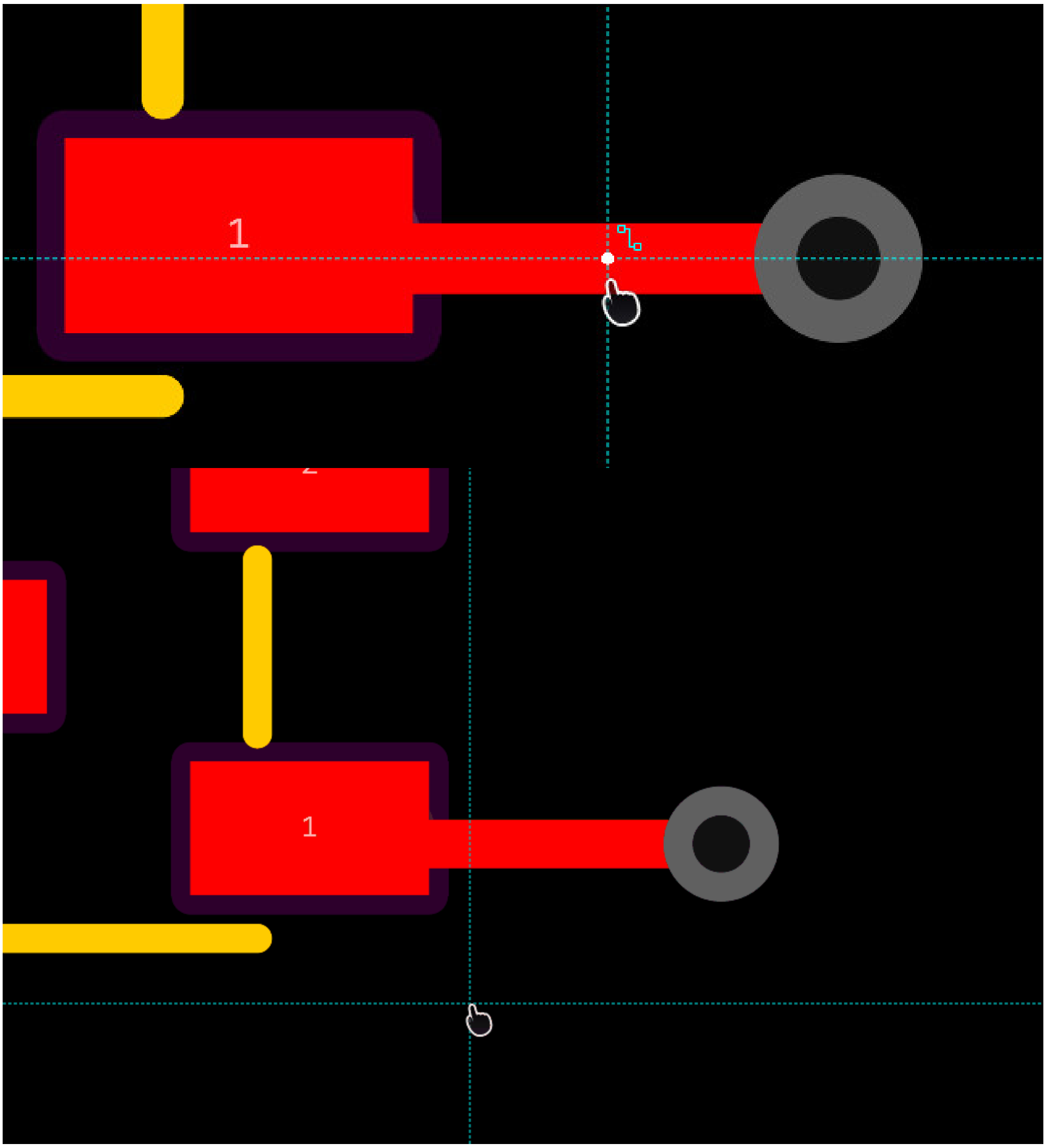
Center of via

Check the center point that can adsorb through the hole

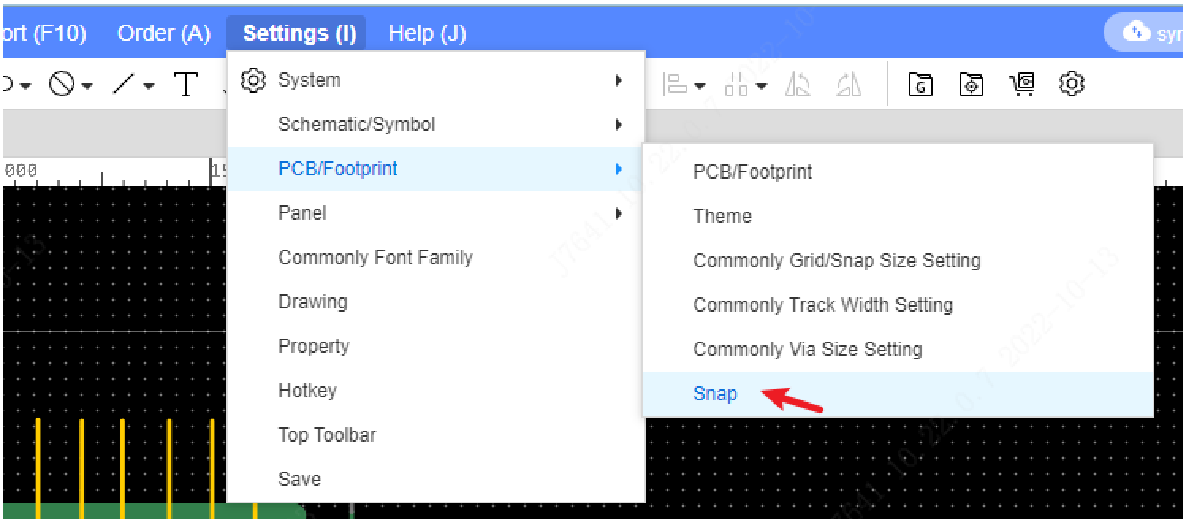


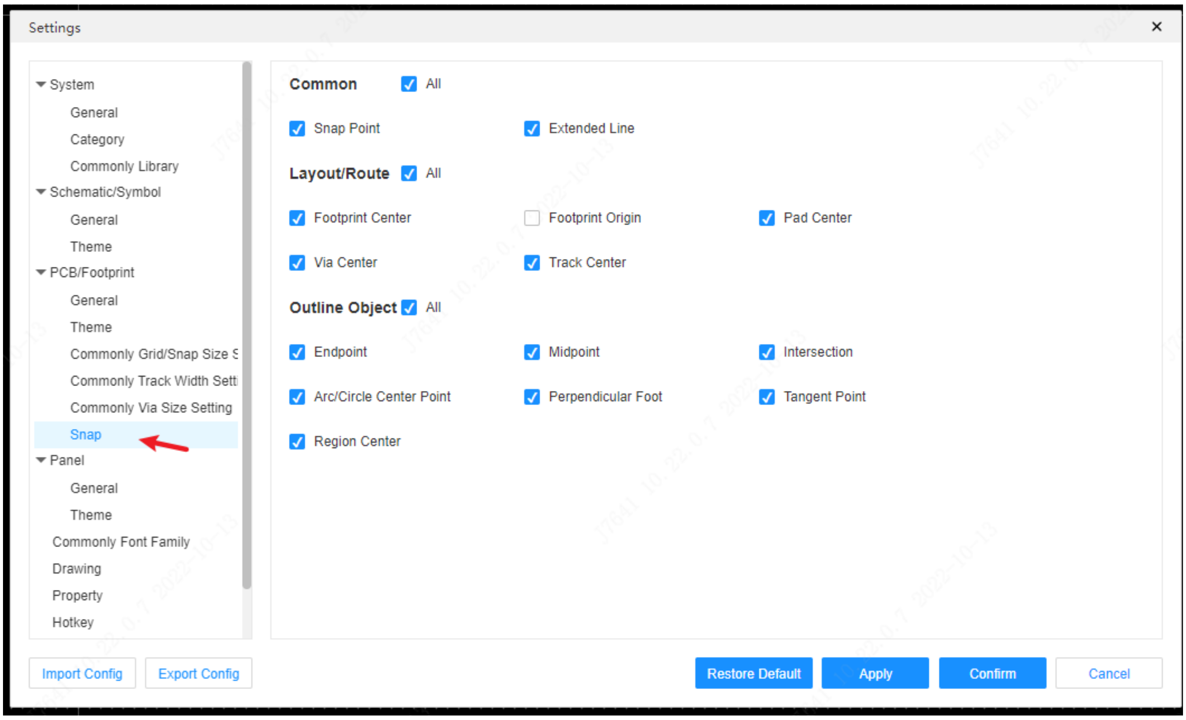
Center of conductor

After checking, it can be adsorbed to the wire



Entrance: Top Menu - Settings - System Settings - Snap

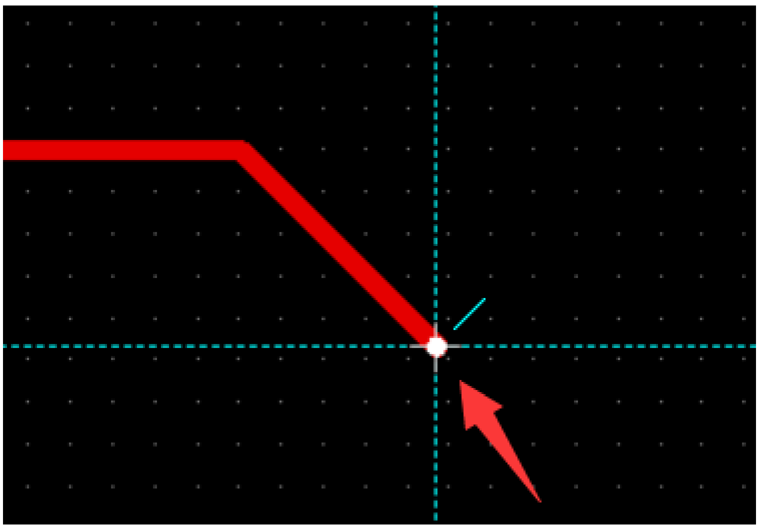




Outline object

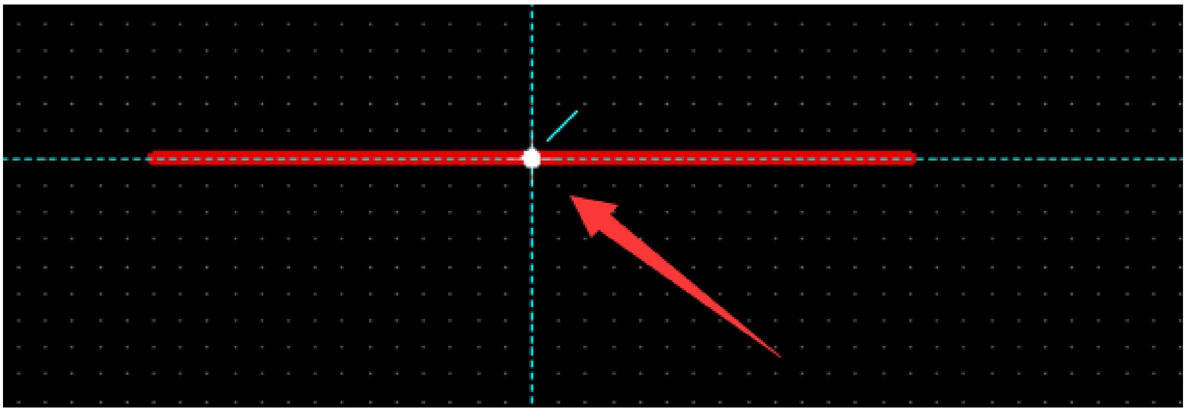
Endpoint

The endpoint at the end of the polyline, this setting is only valid for polylines.



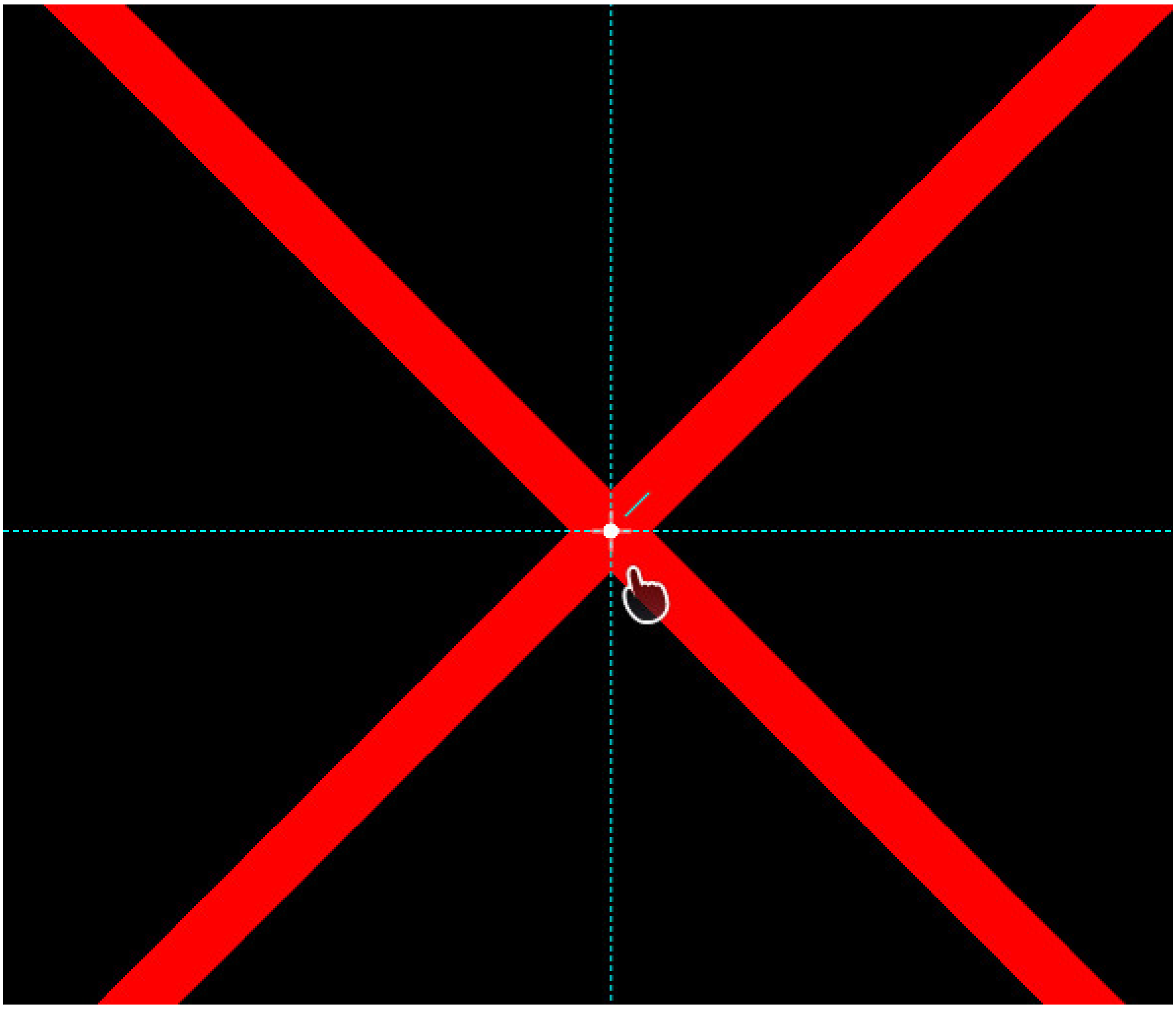
Midpoint

After checking, it can be attached to the center point of the line segment of the line or contour object.



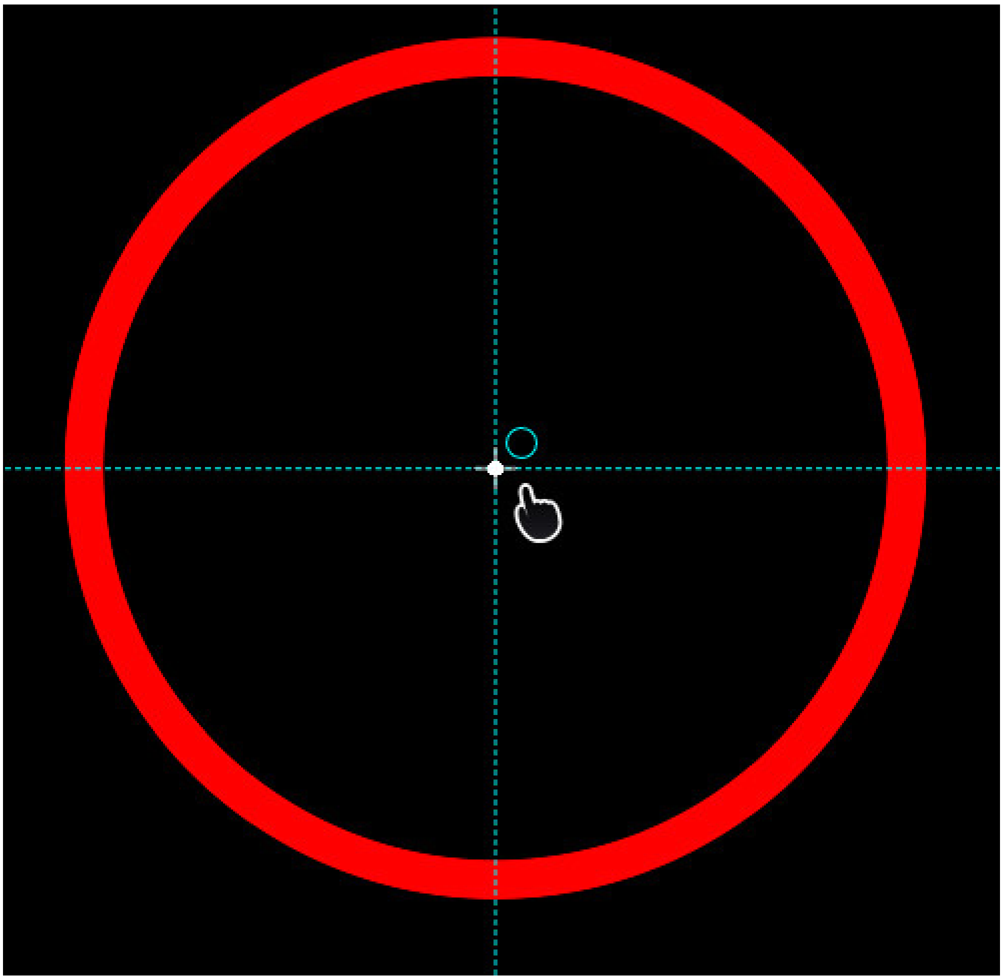
Intersection point

After checking, you can adsorb the intersection points between two or more lines in the same layer.



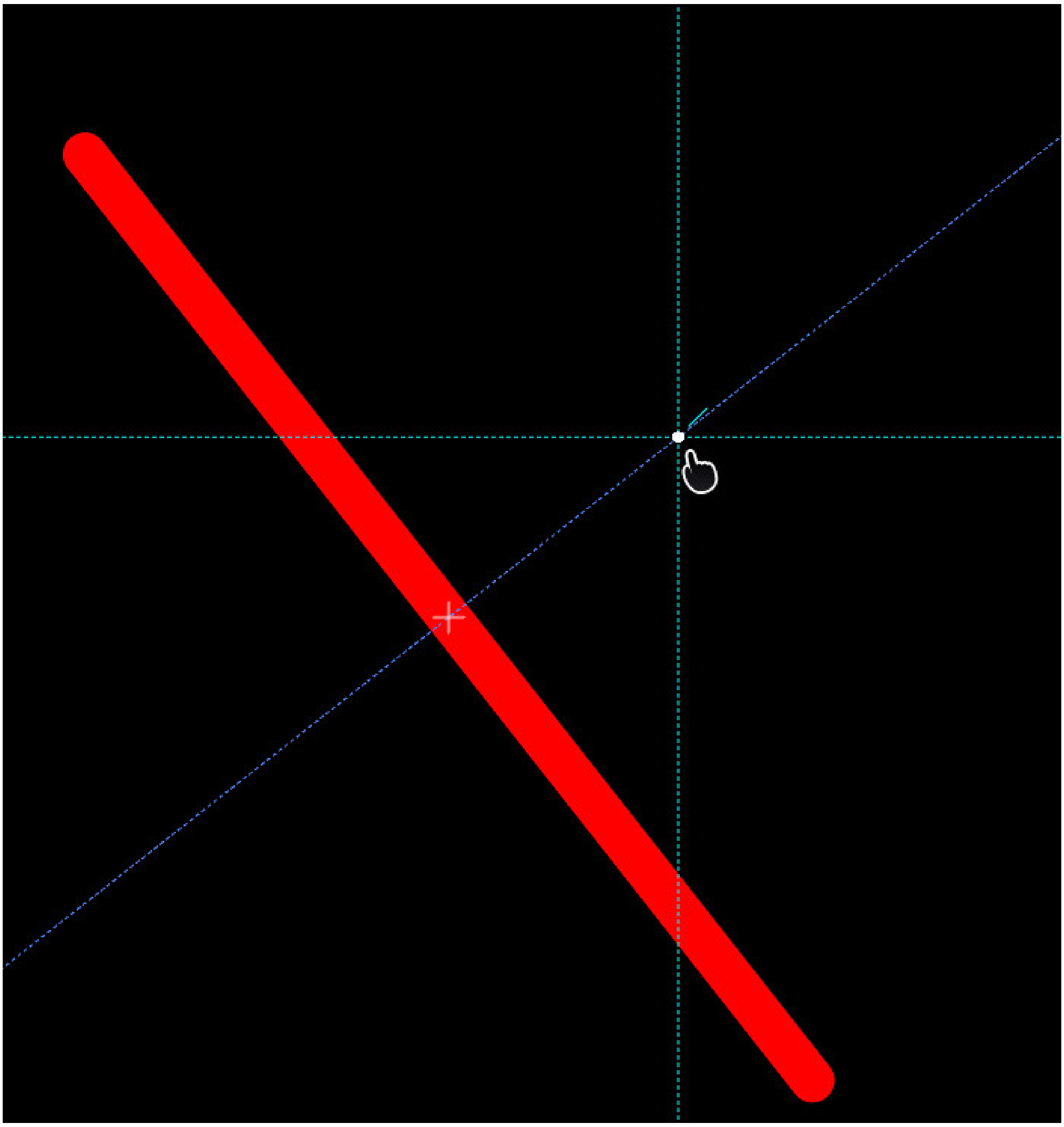
Center of circle

After checking, the center points of circles and arcs can be adsorbed. You need to move the mouse to the prototype's ring before the middle adsorption mark will be generated



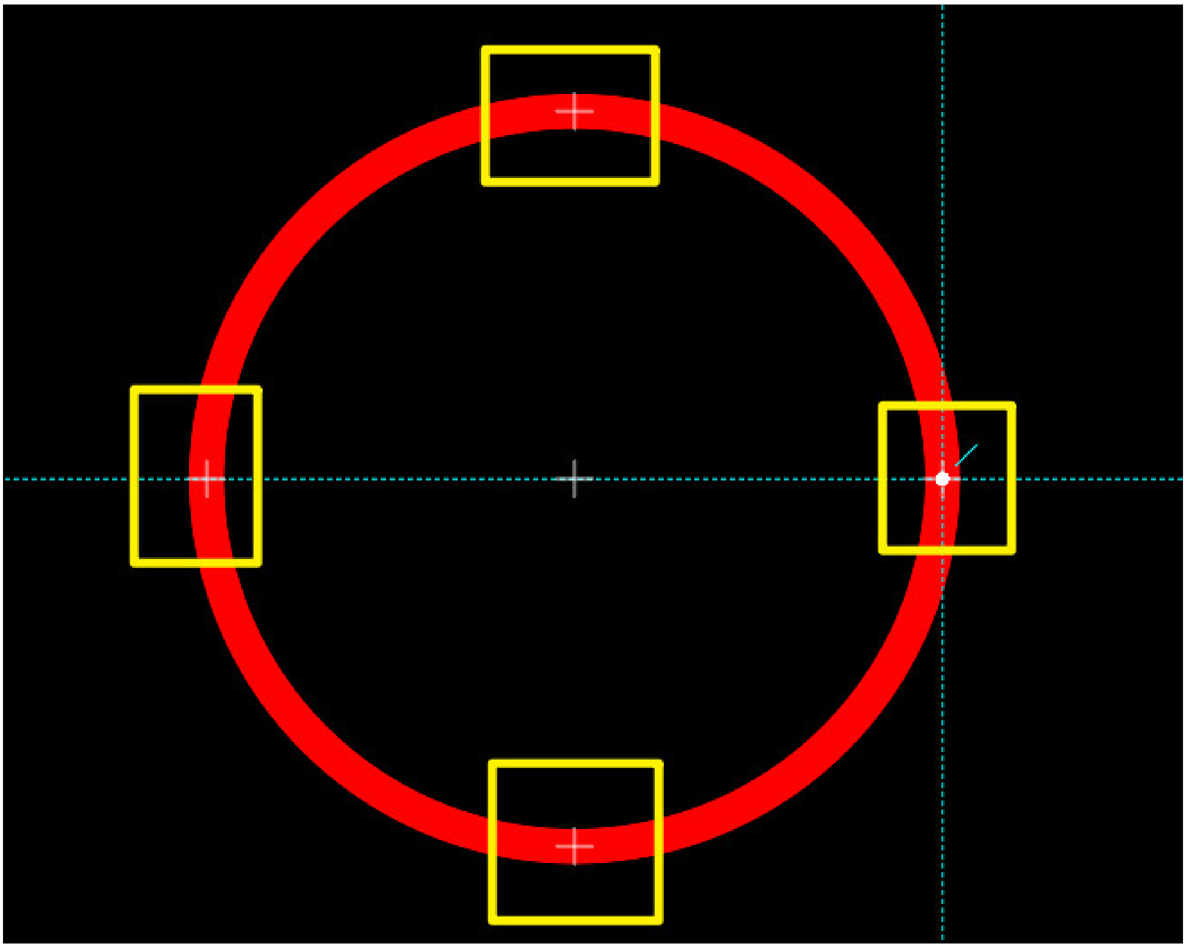
Vertical foot

After opening this item, you can adsorb to the vertical line based on the line segment that produces the adsorption mark



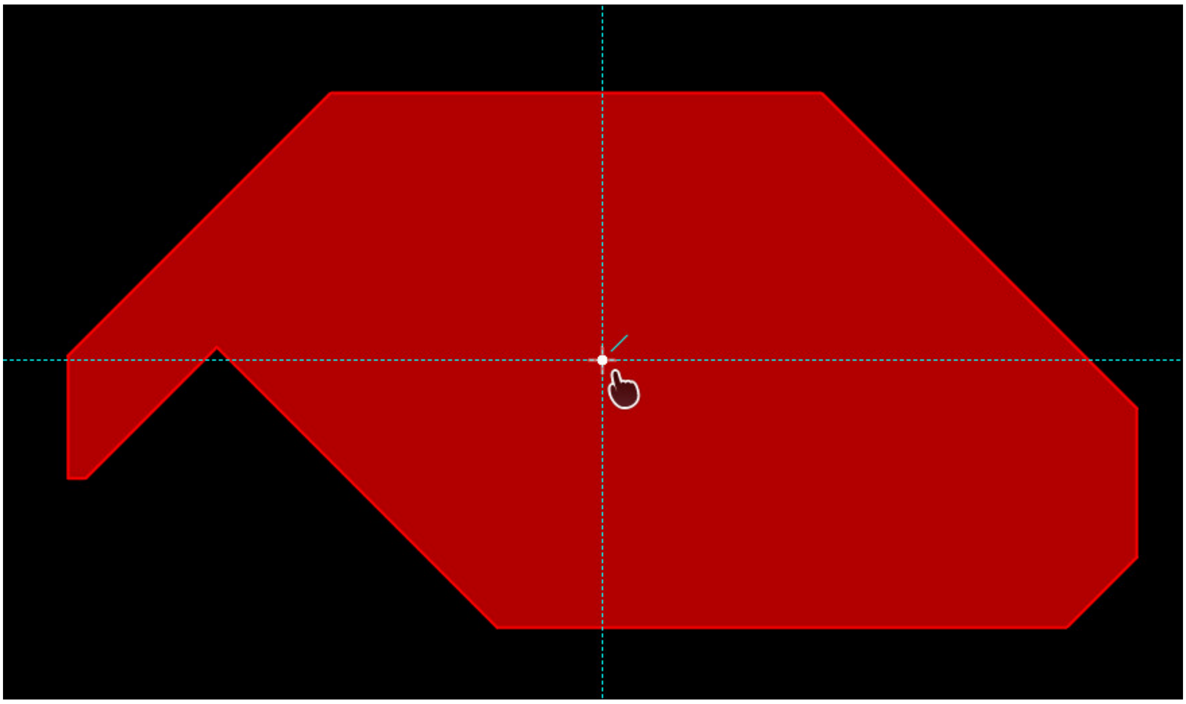
Tangent point

After this option is selected, it can be attached to four tangent points of the prototype orthogonal tangent



Regional centres

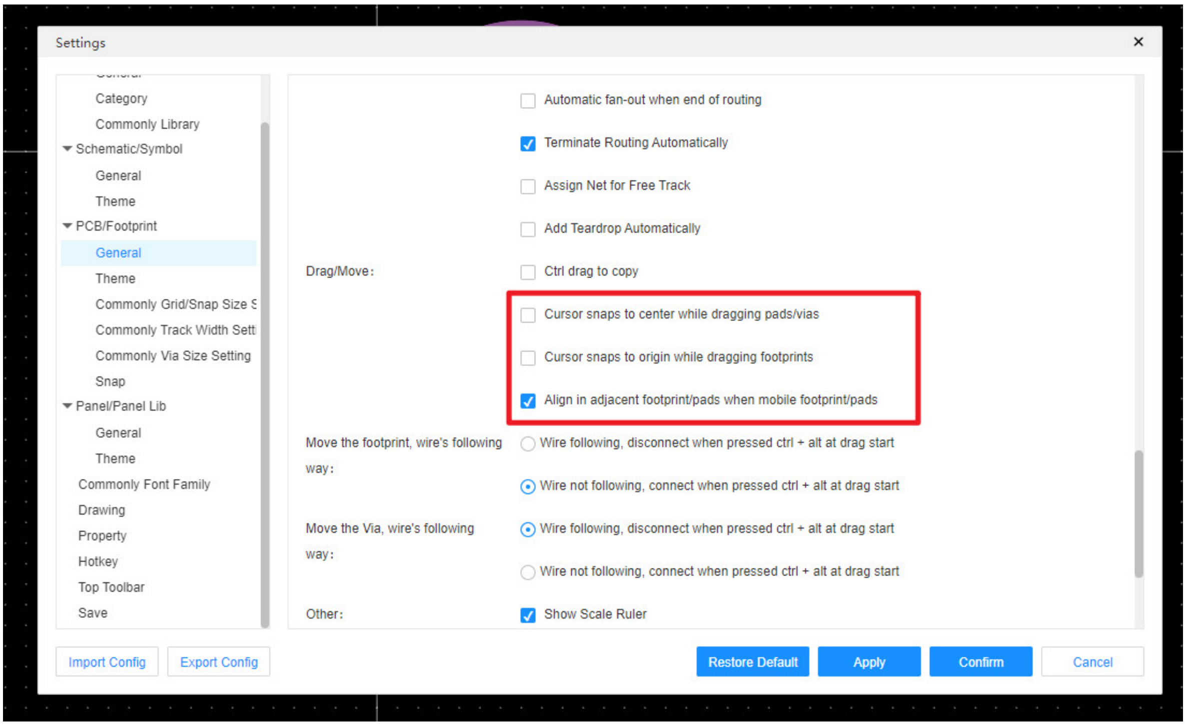
After checking, it can be adsorbed to the graphic center point of contour objects such as copper paving area and filling area.



Center of line segment

After this option is selected, it can be adsorbed to the center of the line segment, similar to the adsorption of the wire center

Other adsorption



In PCB conventional settings, some adsorption effects can be opened when dragging packages or solder pads

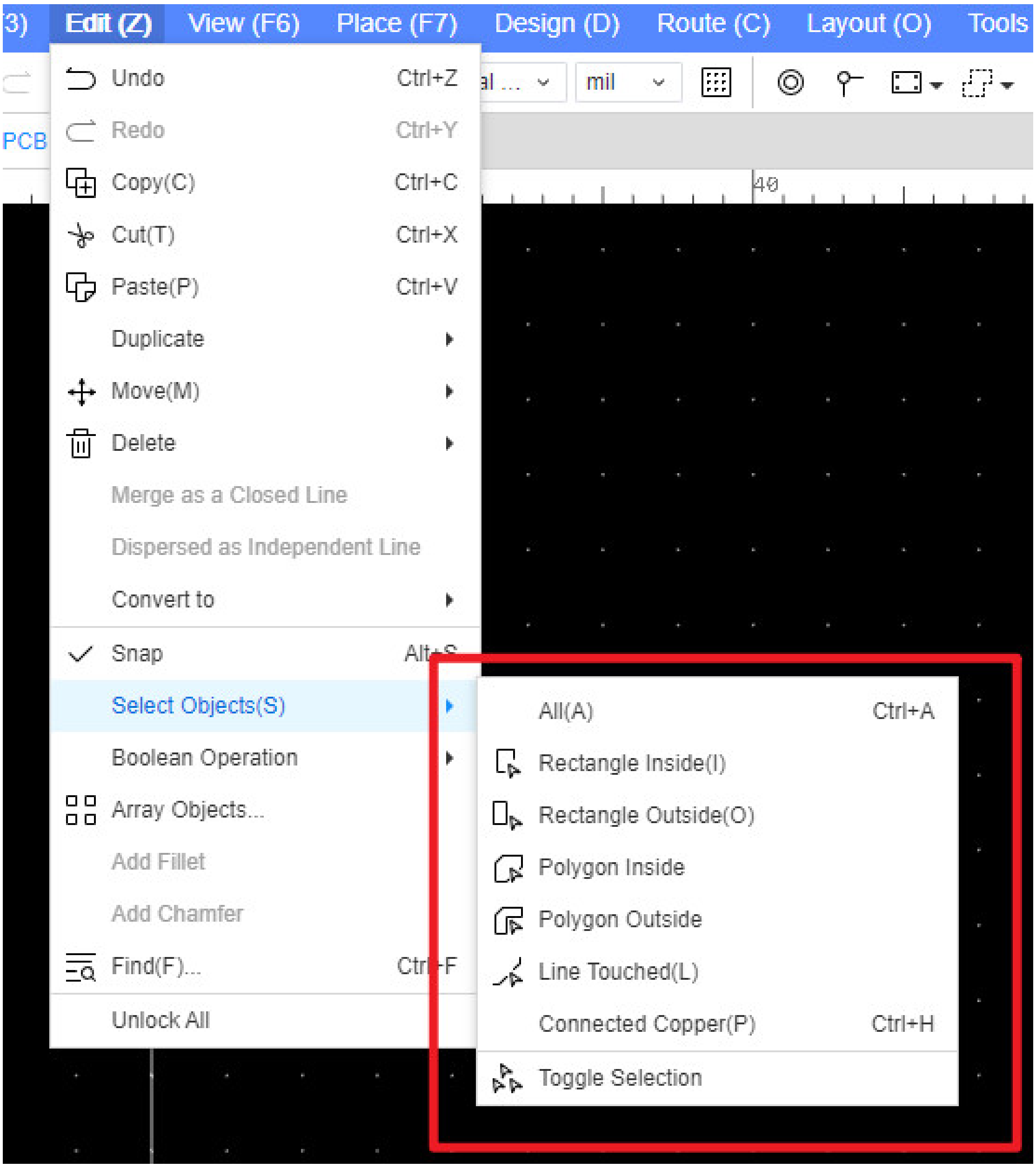
Select Object

all

Select all the devices in the PCB editor, and batch modification can be performed after selection.

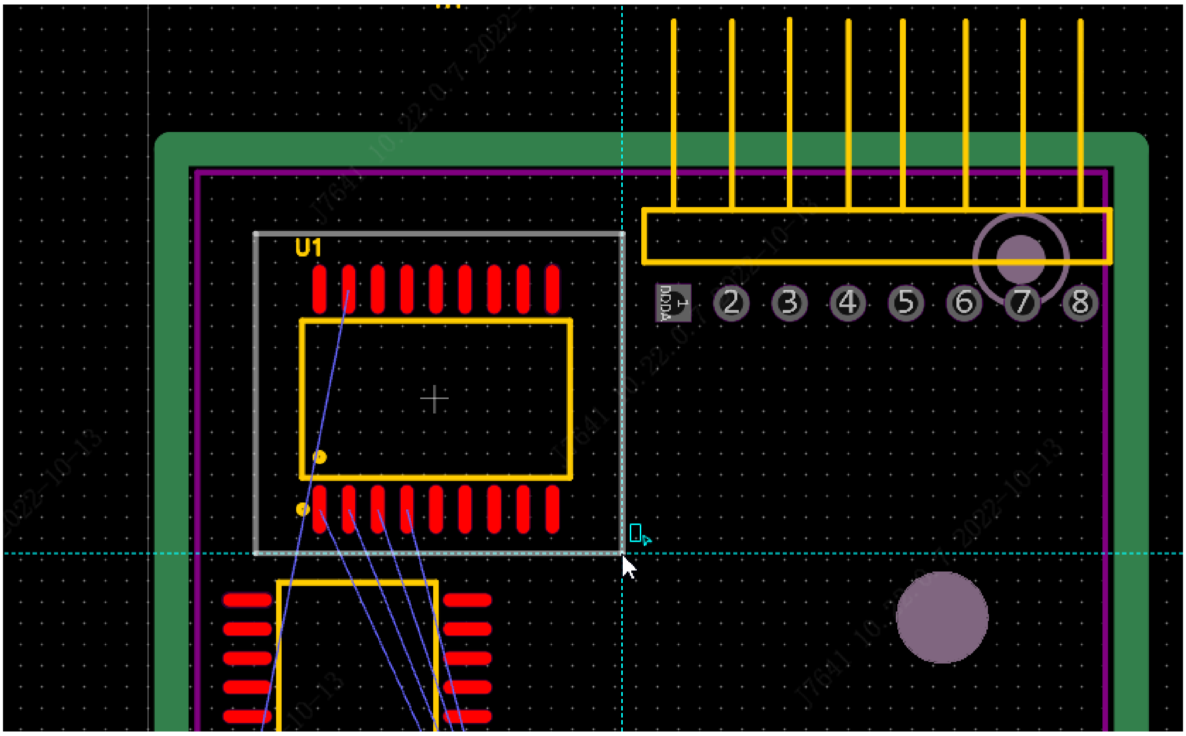
Instructions:

- Top Menu - Edit - Select Object - All
- Use shortcut keys `Ctrl` + `A`



outside the rectangle

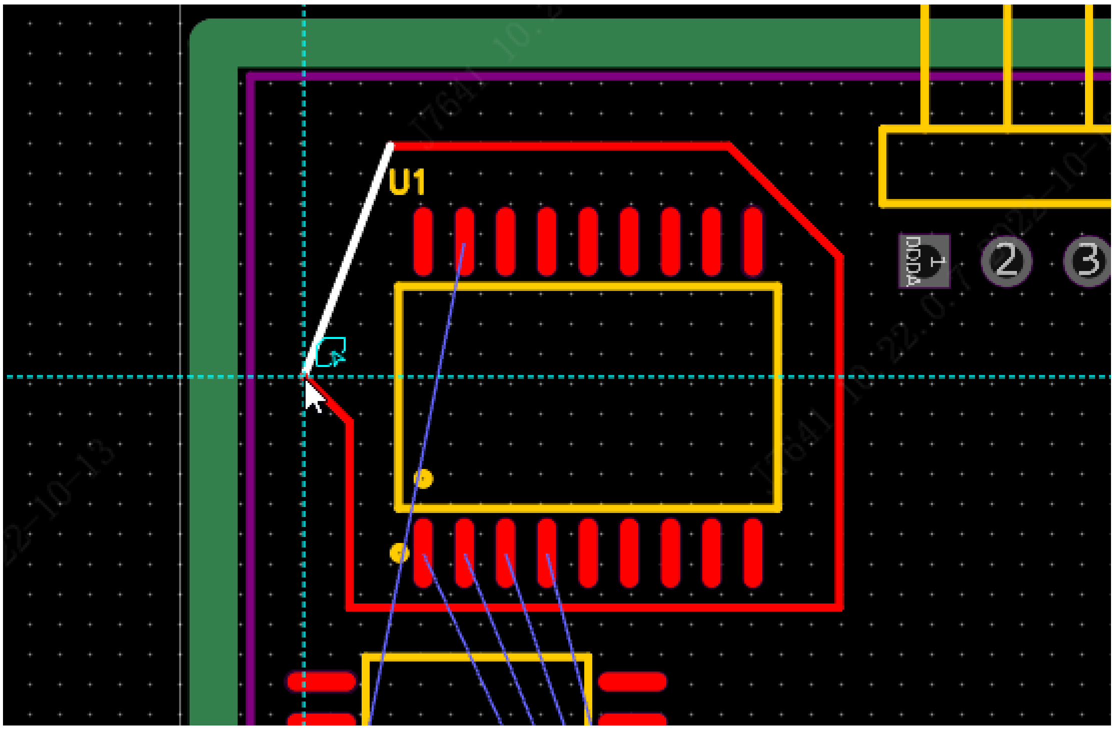
- Top Menu - Edit - Select Object - Outside Rectangle



All elements outside the rectangle will be selected, and elements inside the rectangle will be unselected.

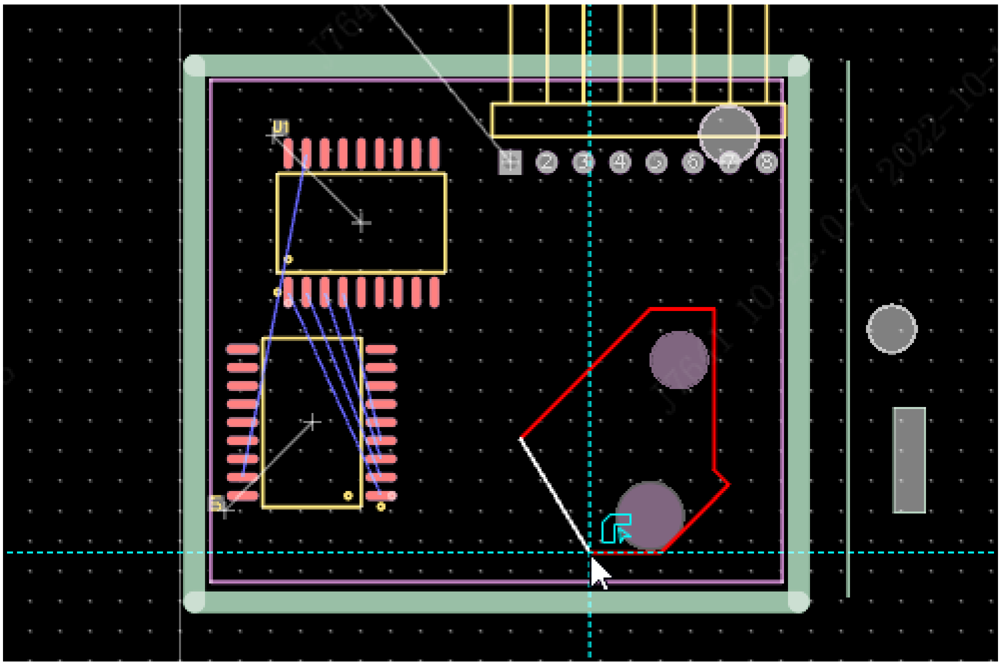
inside the polygon

- Top Menu - Edit - Select Object - Inside Polygon



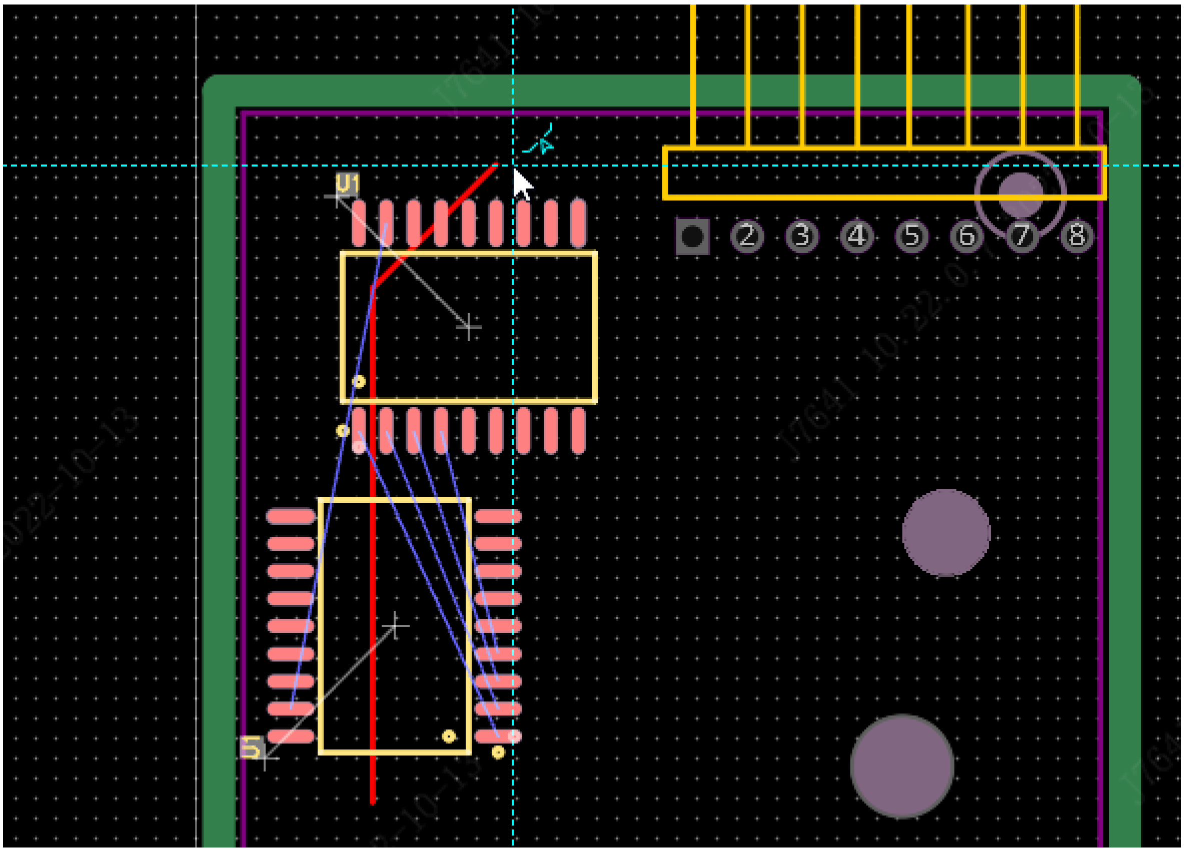
outside the polygon

- Elements within the drawn polygon are not selected, and all elements outside the polygon are selected



touches the line

- Elements that touch the line will be selected, elements that are not touched will not be selected

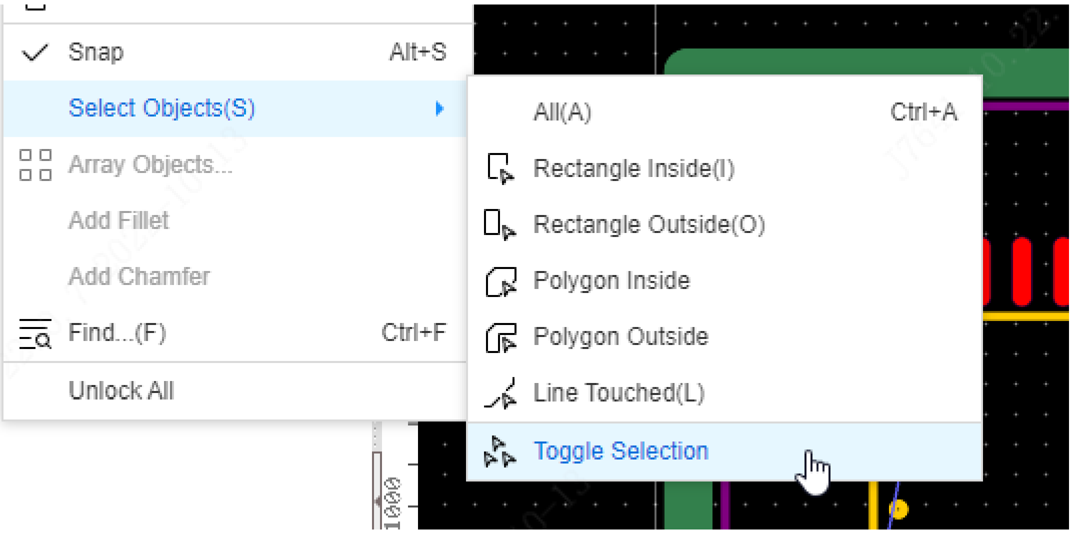


Connected Copper

After entering the function, click any graph element with network attribute, and the graph element with connection will be directly selected, regardless of whether it belongs to the same network

Toggle selection

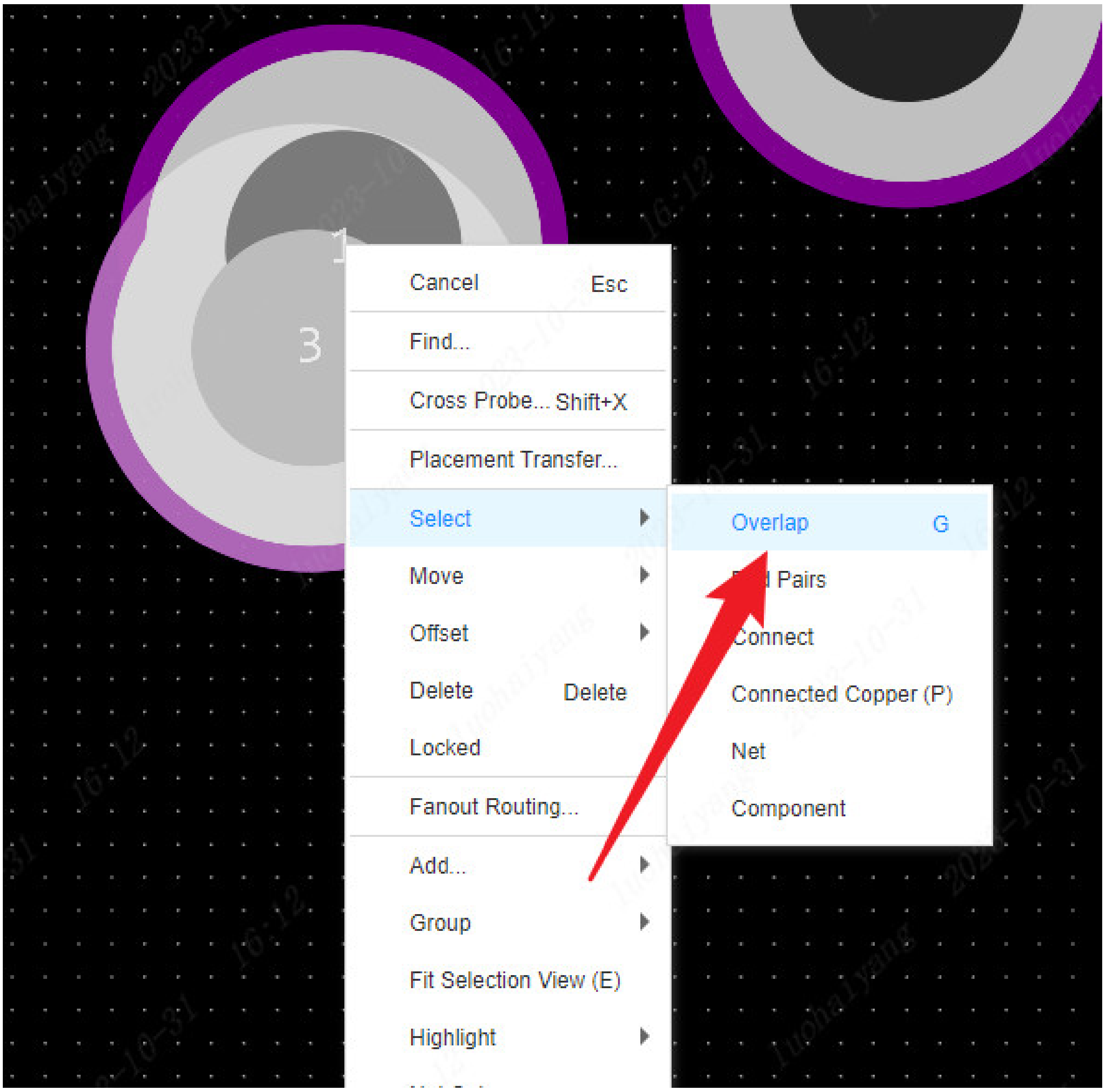
- Top Menu - Edit - Select objects - Toggle selection



Continuous selection is similar to `Ctrl` +mouse-click continuous selection.

Select overlapping objects

When two primitives overlap, the mouse click and pick may be difficult to select the desired one, you can use the select overlap function. Keyboard shortcut `G`



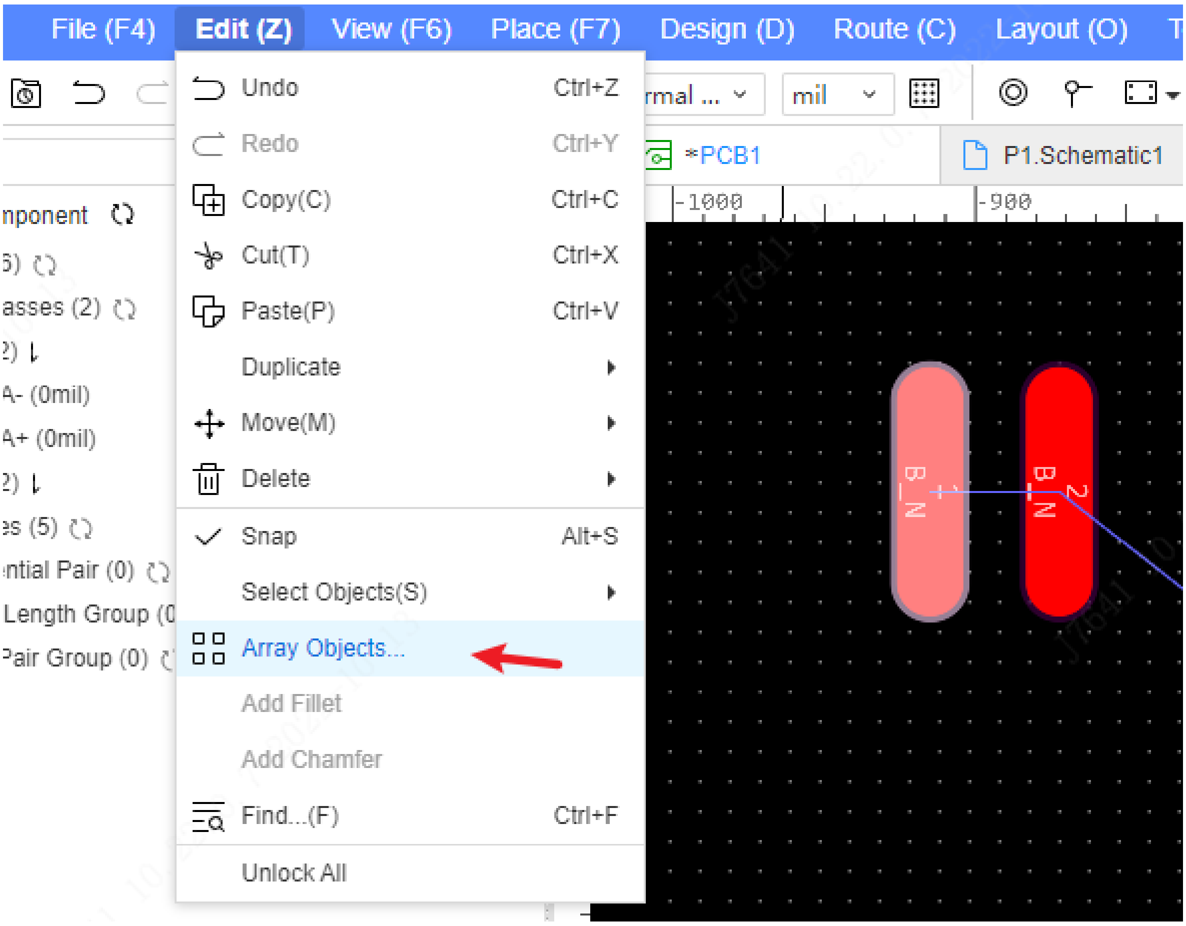
The shortcut keys can be used to continuously select the primitives around the mouse.

Array Object

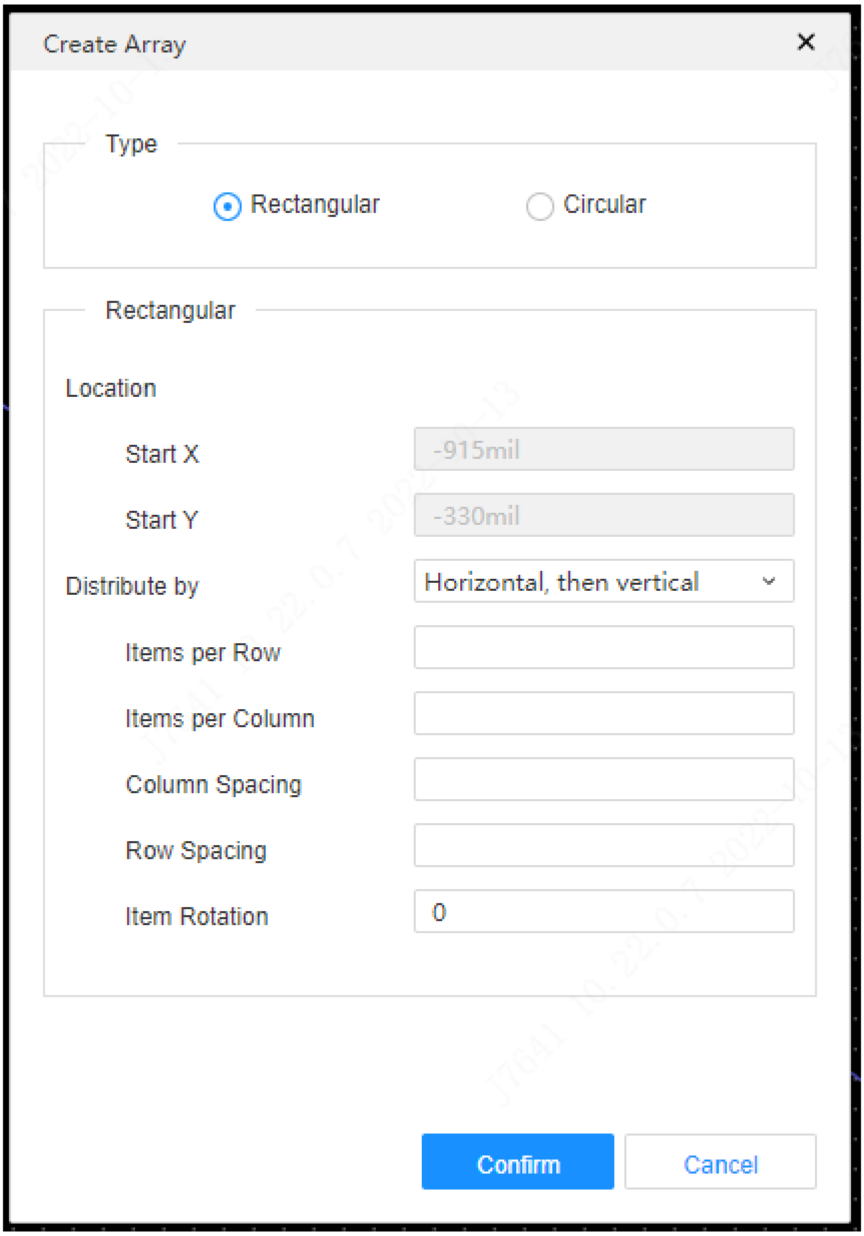
The array function of the device array and the footprint library is similar, but some more functions are added on the basis of the footprint library.

Instructions:

- Select object - Top Menu - Edit - Array Object



The property panel of the popup array, the default is the rectangle property panel.

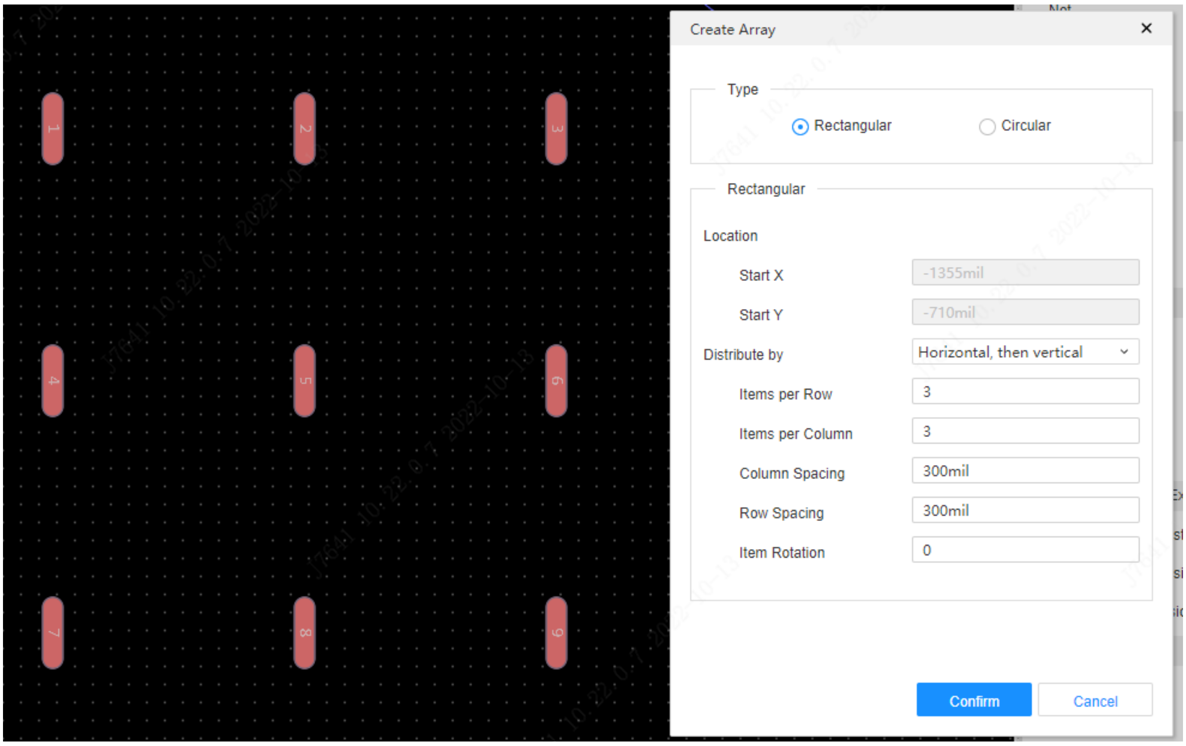


Rectangular array settings

Select Rectangle, the starting point of the array defaults to the selected device location.

- **Type**: choose the shape of the array
- **Position**: The position of the rectangular array is the position of the selected device by default, while the circular array needs to be selected by yourself.
- **Distribution Type**: Only the array sorting setting of horizontal first then vertical or vertical first horizontal can be selected.
- **Number of rows per row**: The number of rows in this array.
- **Number of columns per column**: Set the number of columns to write to the array.
- **Column Distance**: Set the distance between column arrays.
- **Line Spacing**: Set the distance of the array lines.
- **Element rotation angle**: Set the angle of the array after Place.

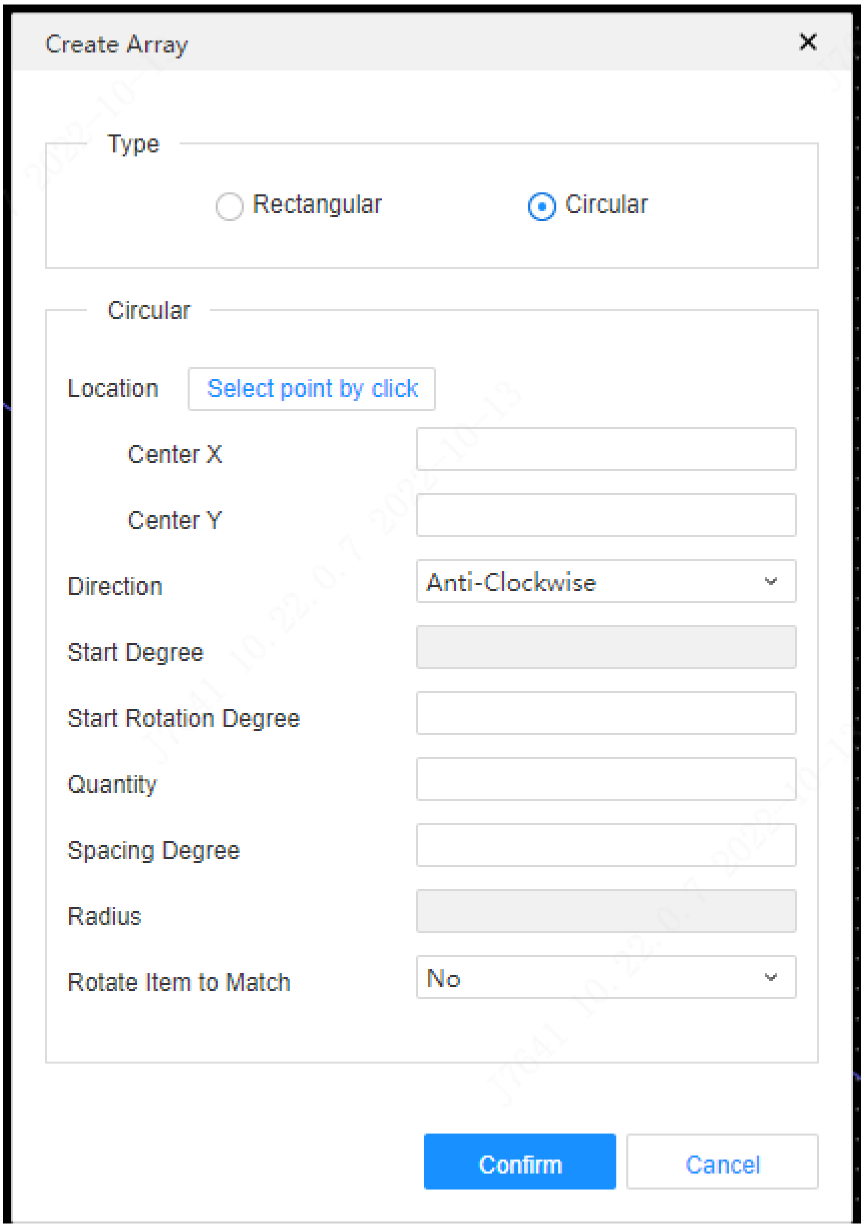
For the distribution type, we can choose horizontal first and then vertical or hammer first and then horizontal, and follow up on personal needs.



Properties of circular array

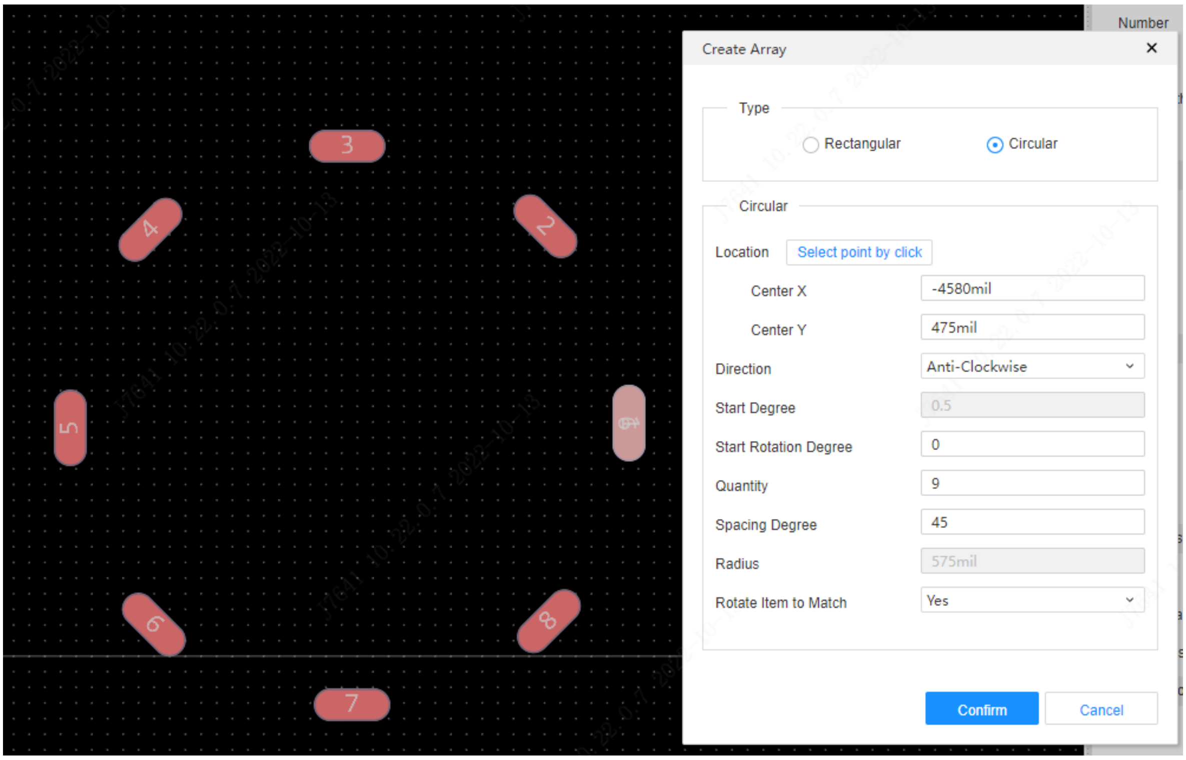
Set the pattern type to circle

Properties panel for circular arrays



- **Position**: Set the center point of the circular array.
- **Direction**: You can set the direction of the array to be counterclockwise or clockwise.
- **Starting rotation angle**: After selecting the position, the system will default to the starting angle.
- **Start rotation angle**: Set the rotation angle of the starting array object.
- **Number**: Set the number of array objects.
- **Spacing Angle**: Set the spacing of each array object.
- **Radius**: Default and selected radius, cannot be changed.
- **Select object to fit circle**: Select whether the device is suitable for use in a circular array.

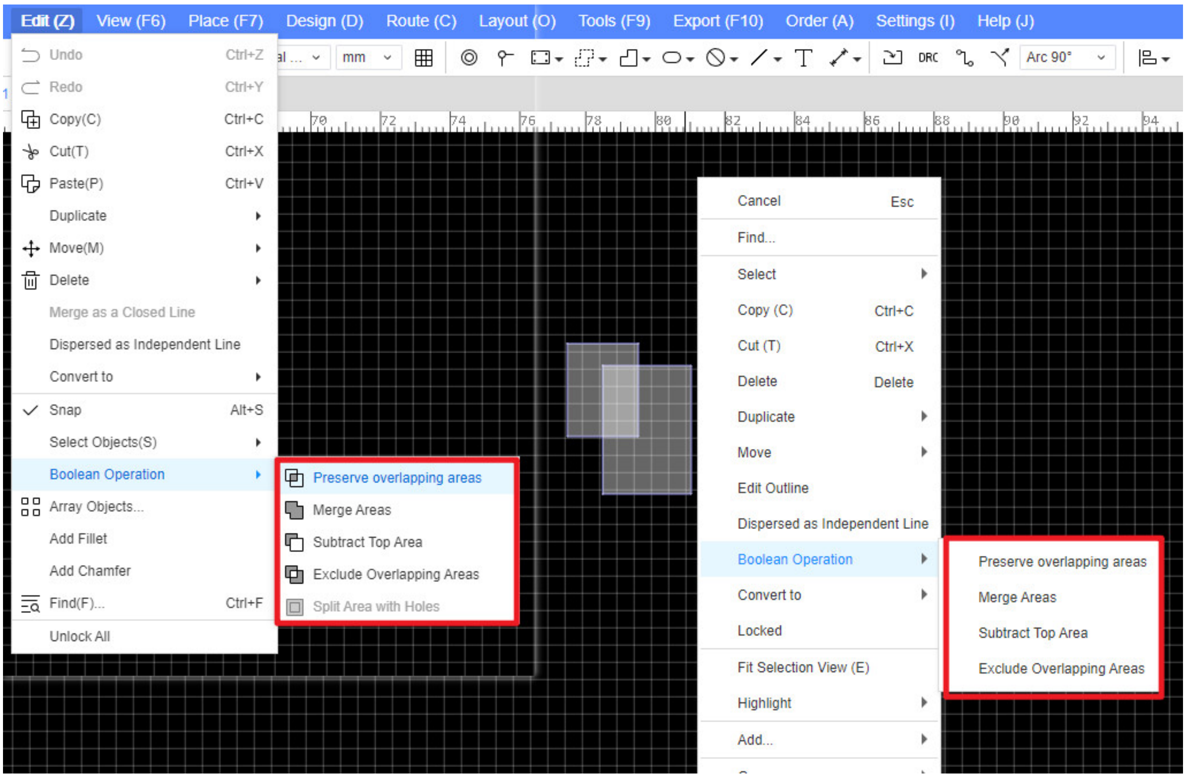
Select the center position of the circular array, which can be selected with the mouse or entered manually.



Boolean Operation

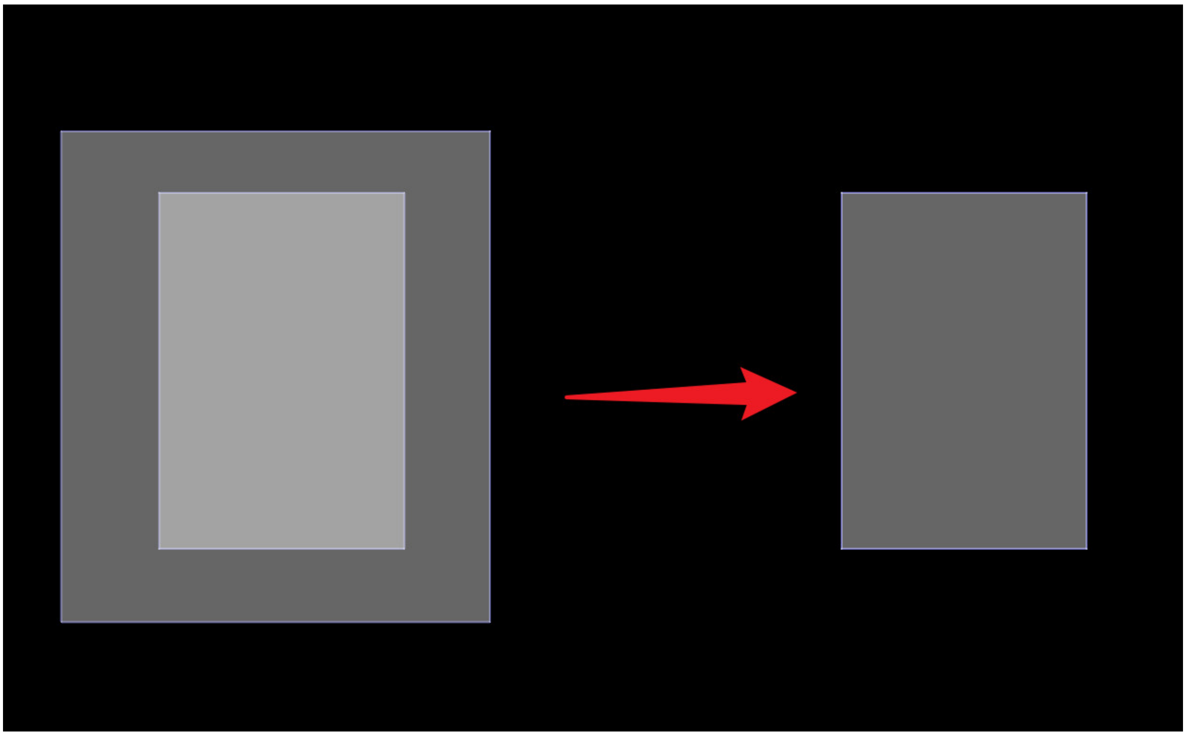
Operating entrance

- Top Menu -Edit -Boolean Operation
- Right -click menu -Boolean operation
- shortcut keys (need to be added by yourself)



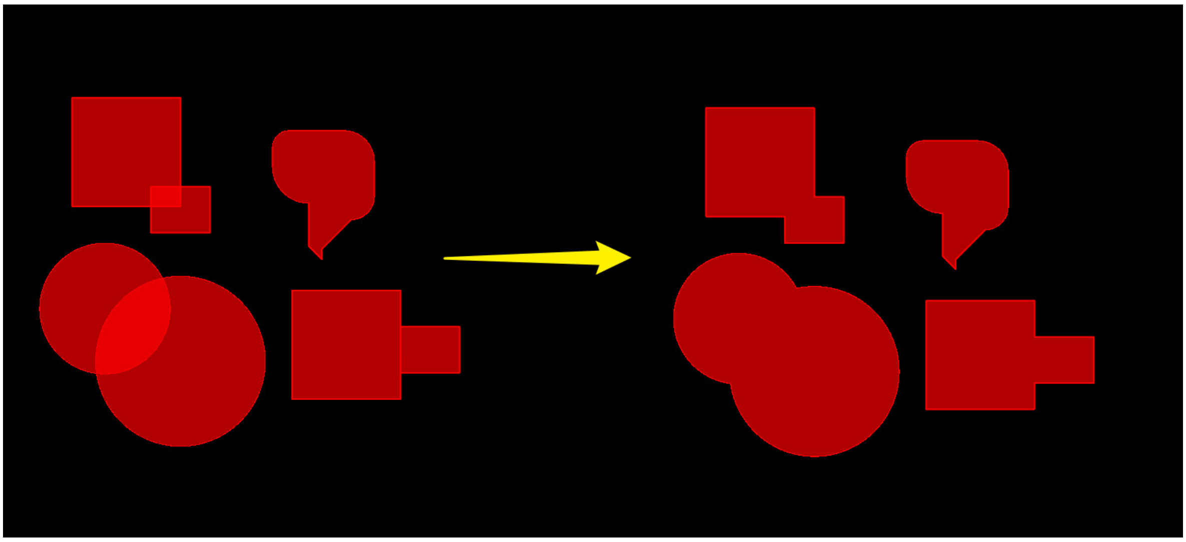
Preserve Overlapping Areas

Select this menu can only retain all the area overlap overlap. This operation currently supports all contour areas and FPC reinforcement boards



Merged Area

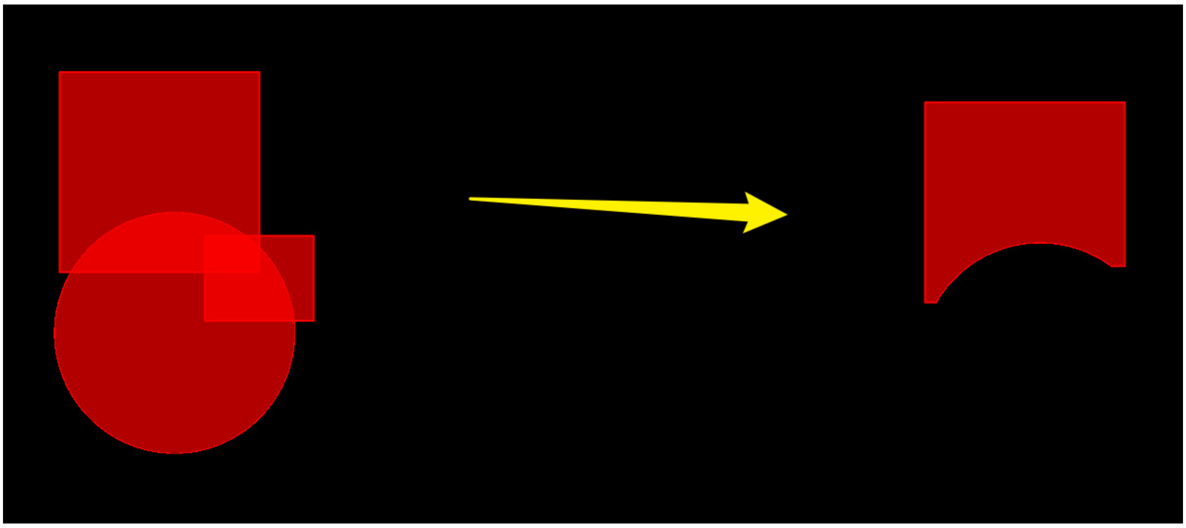
You can merge the maps close to each other or overlap into one. At present, all outline objects and FPC reinforcement boards support



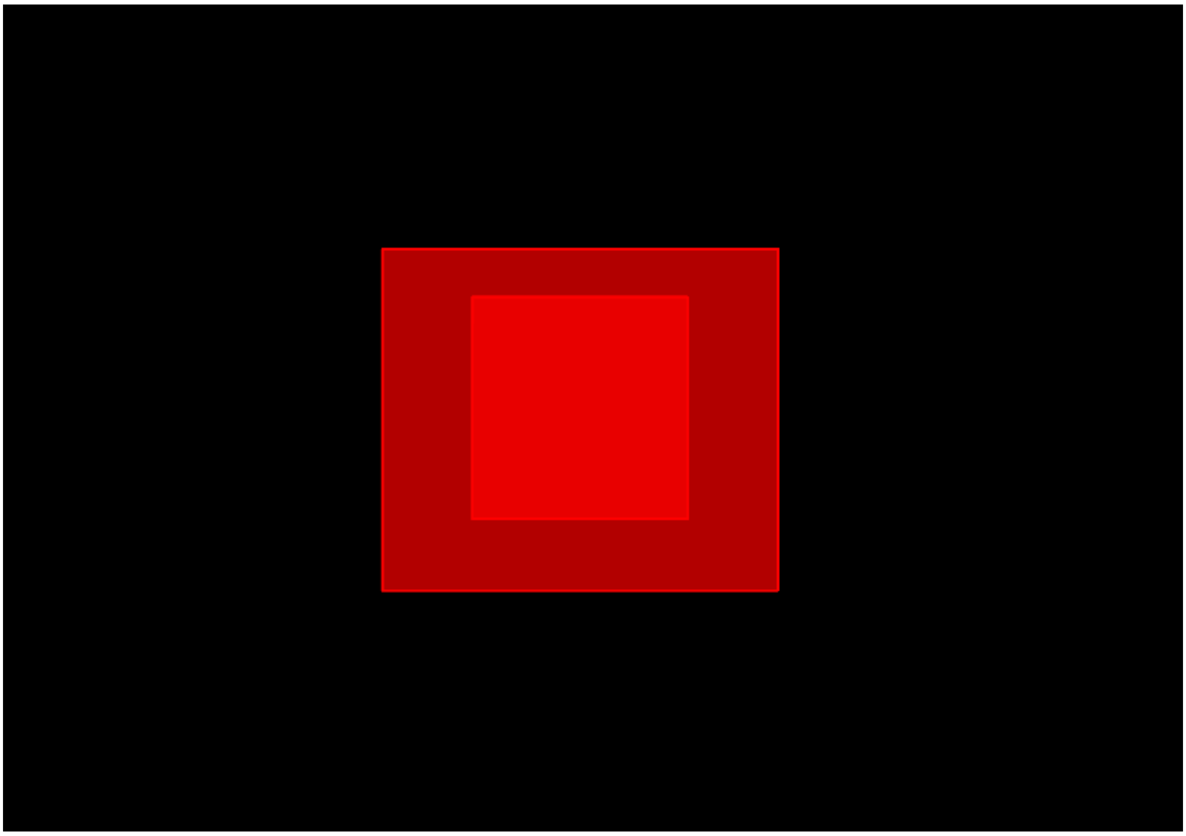
Subtract Top Area

You can selectively subtract the overlap of the image element, and currently only support the filling areact the overlap of the image element, and currently only support the filling area

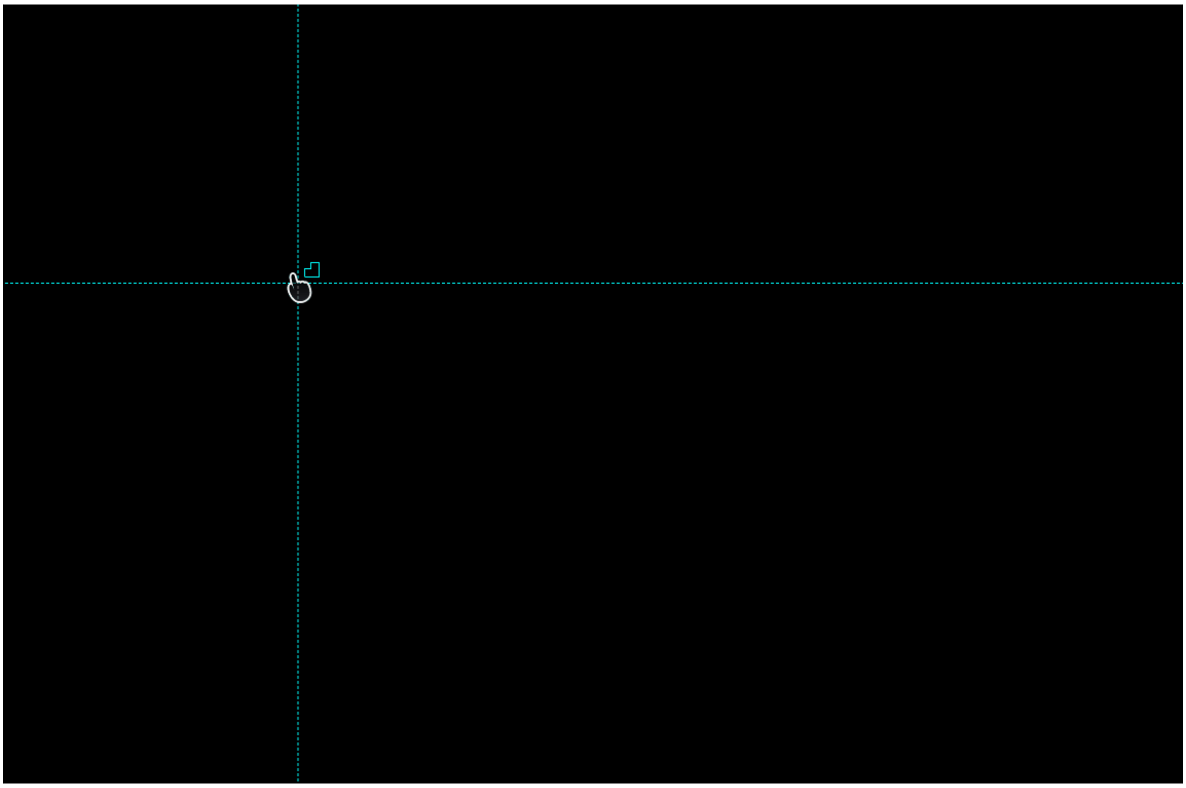
When choosing to subtract the top -level area, you can decide which part of the reserved part of the ctrl key is selected by holding down the CTRL key.



If you choose multiple charts in a one -time direct box, and the map is a completely contained relationship, the small is not the benchmark, subtract the small

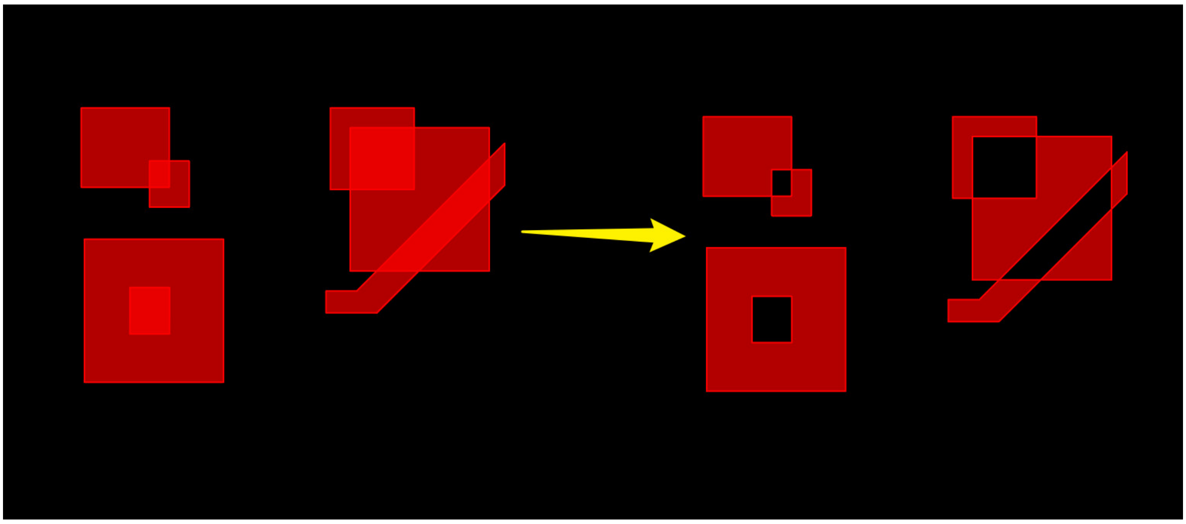


If you choose multiple charts at a one -time box, and the image is only partially intersecting, based on the order of the plans on the canvas, the first plans are set as the benchmark.



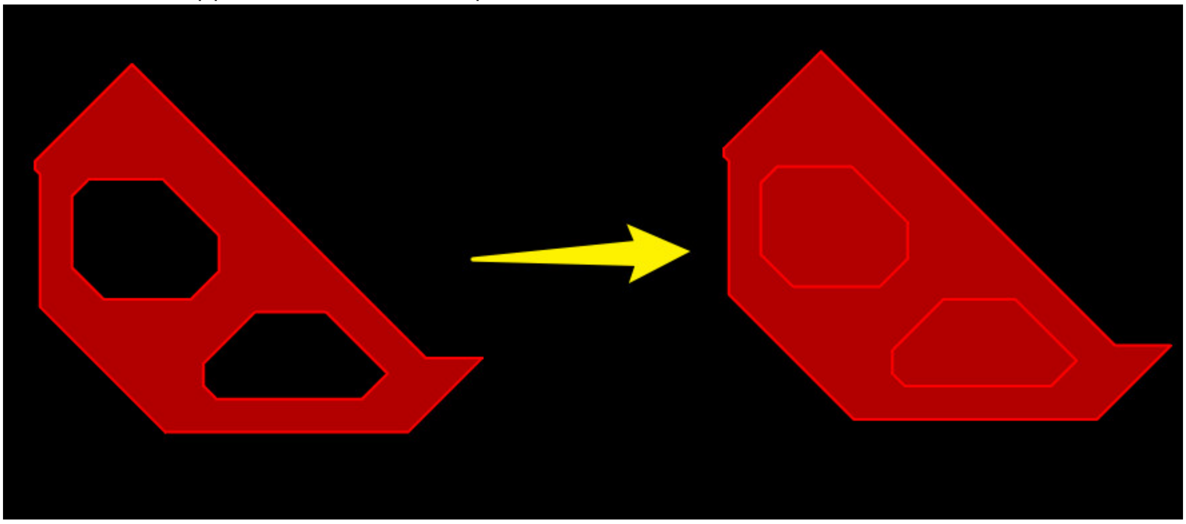
Exclude Overlapping Areas

You can subtract all the overlap parts of the chart, and currently only support the filling area



Split the Area with Holes

The graphics with holes will be split into a large map with the most accurate appearance and the most accurate appearance and the shape of the cave.



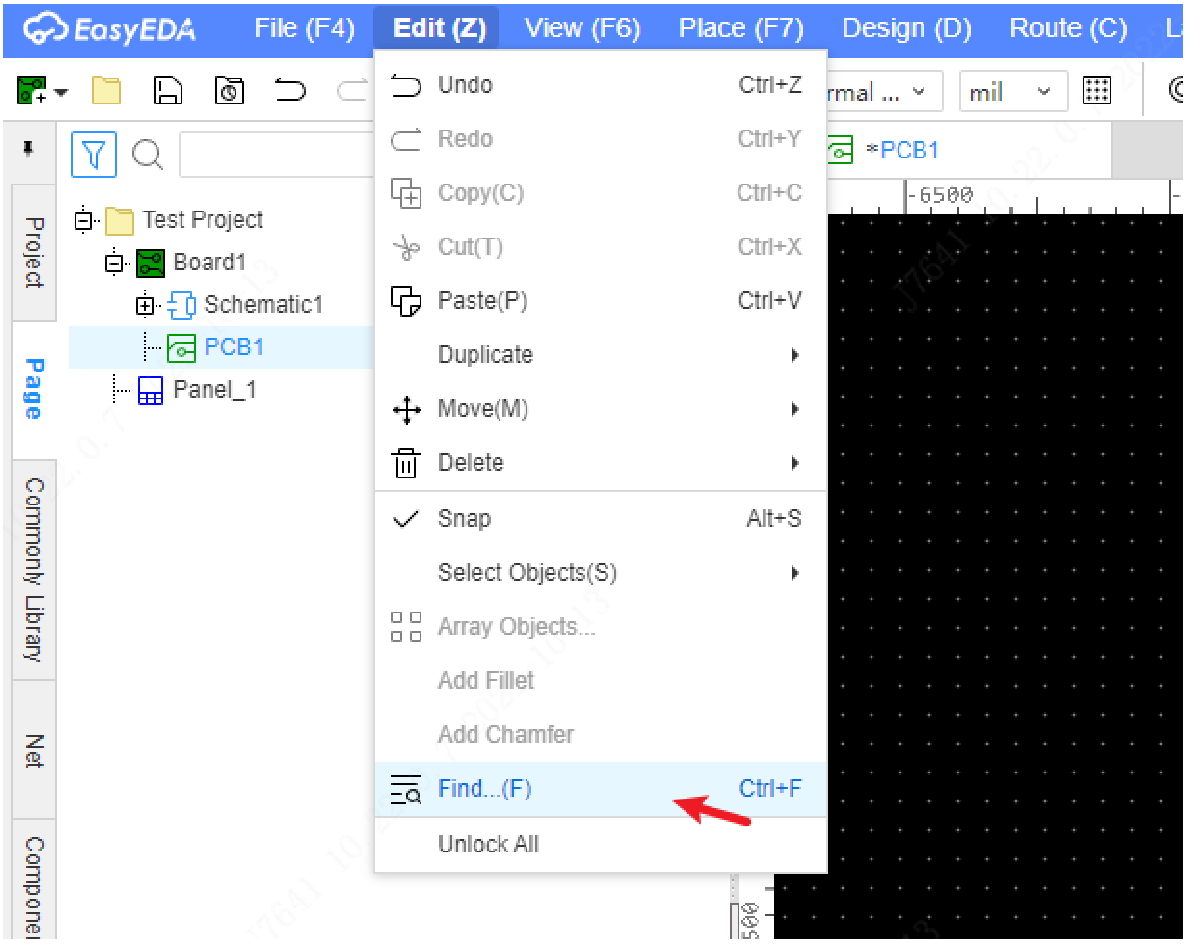
Find

Find components, properties, names, etc. in the PCB interface. The search format can use wildcards and expressions to search.

- Use wildcards. When checked, all text input boxes support wildcards `*?`. For example, to search for tags: `BAT?` or `BAT*`, you can search for all tags starting with BTA. `*` can be used in place of zero, one or more characters, while `?` can only be used in place of one character.
- Case sensitive. Search by case. For example query: `max232` and `MAX232` are two different search terms.
- Use expressions. Supports inputting regular expressions in the text input box for character matching. For example, search: `/u/`, you can find all attributes with u.
  Please check the usage of specific regular expressions: https://www.w3school.com.cn/js/js_regexp.asp

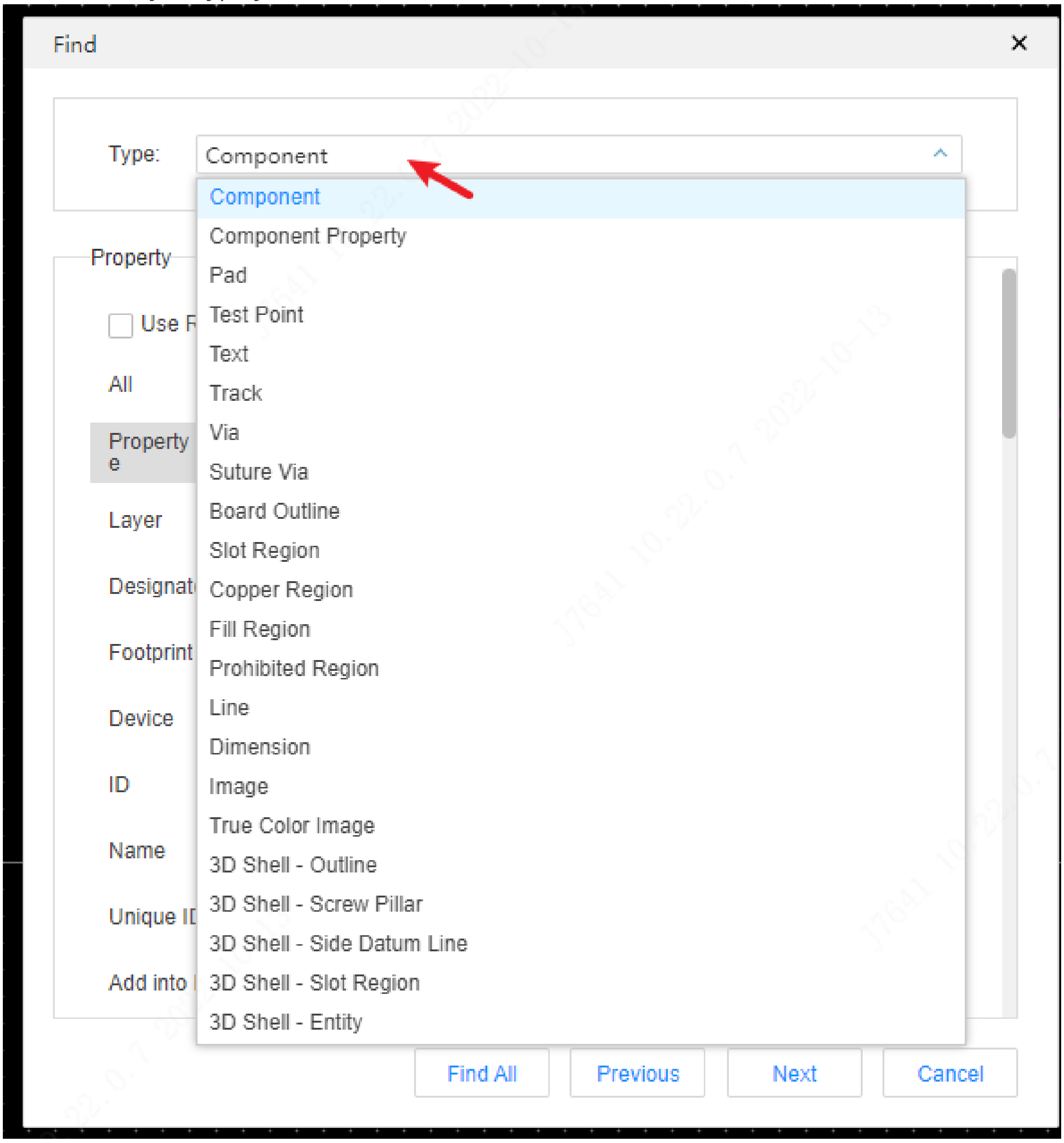
How to find:

- Top Menu - Edit - Find or Shortcut `Ctrl + F`

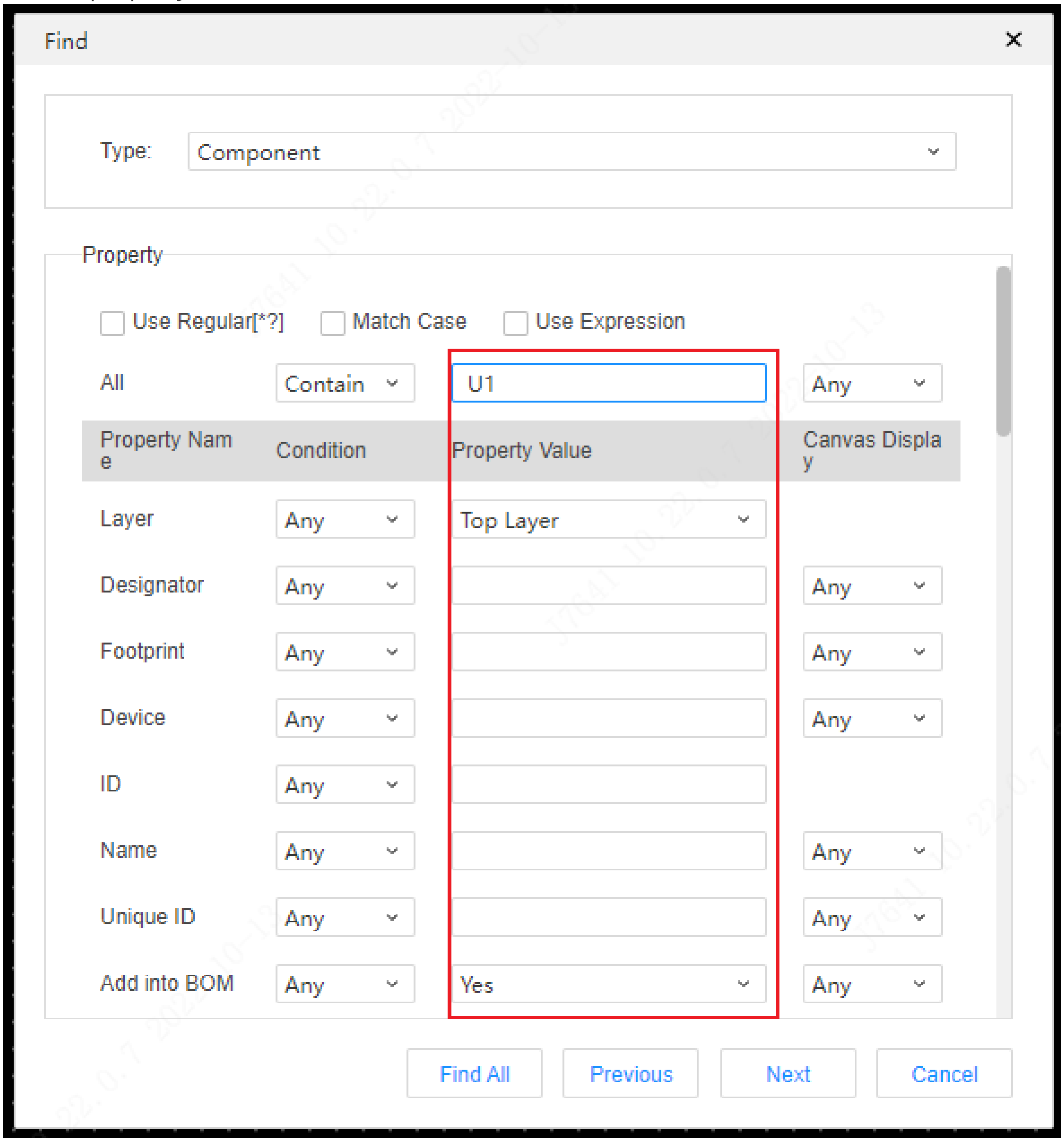


EasyEDA
File (F4)
Edit (Z)
View (F6)
Place (F7)
Design (D)
Route (C)
Undo Ctrl+Z
Redo Ctrl+Y
Copy(C) Ctrl+C
Cut(T) Ctrl+X
Paste(P) Ctrl+V
Duplicate
Move(M)
Delete
Snap Alt+S
Select Objects(S)
Array Objects...
Add Fillet
Add Chamfer
Find...(F) Ctrl+F
Unlock All
Test Project
Board1
Schematic1
PCB1
Panel_1
Project
Page
Commonly Library
Net
*PCB1
mil
6500

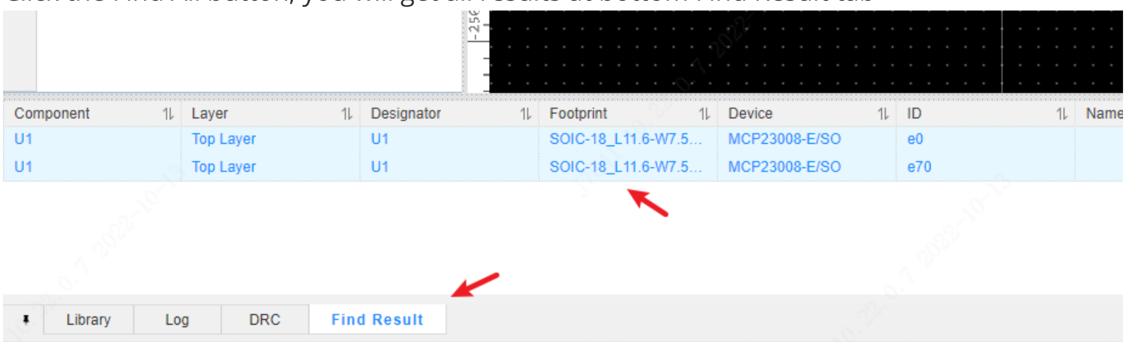
1. Select the object type you want to find



2. Set the property value



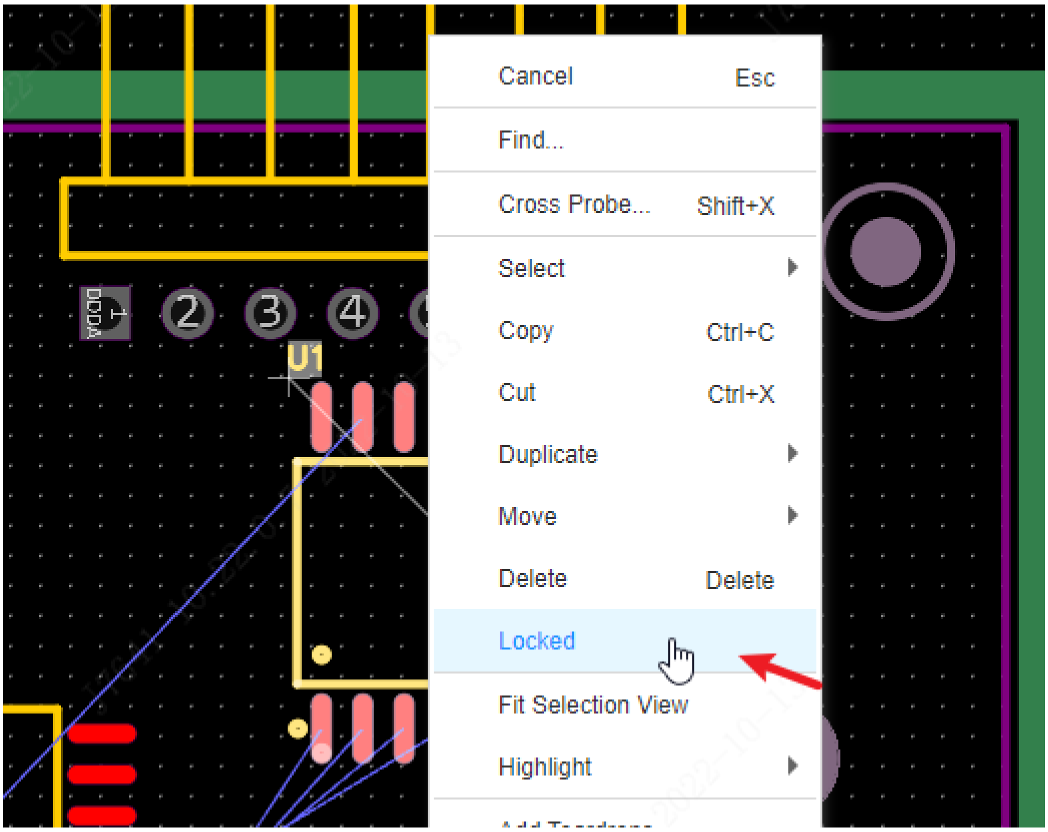
3. Click the Find All button, you will get all results at bottom Find Result tab



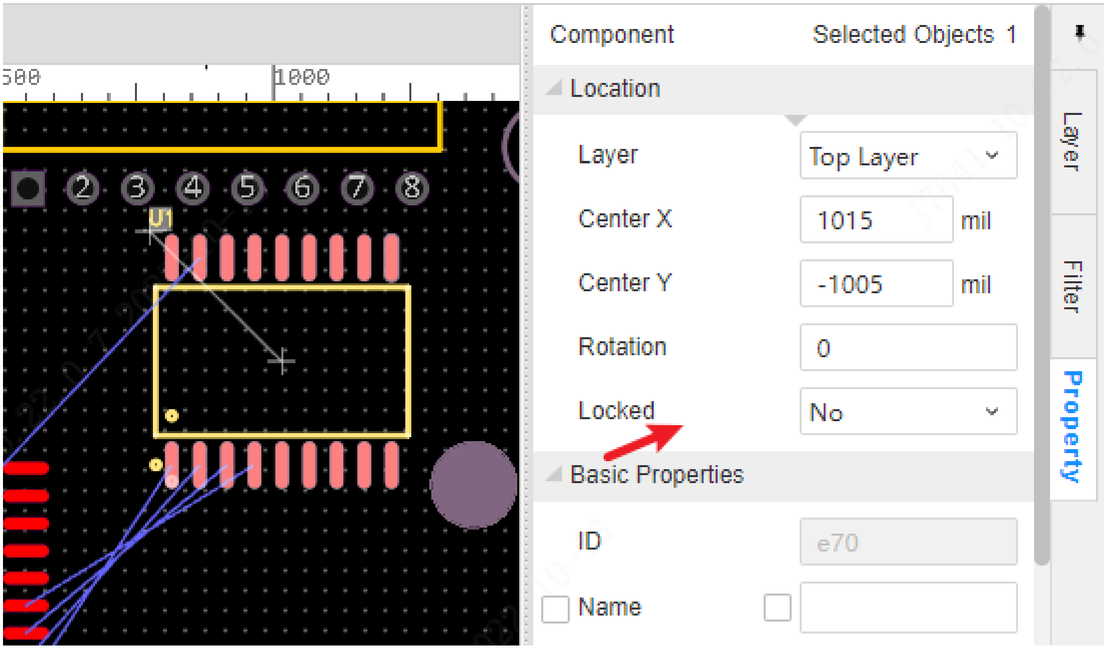
you can click the list to locate the object.

Lock/Unlock

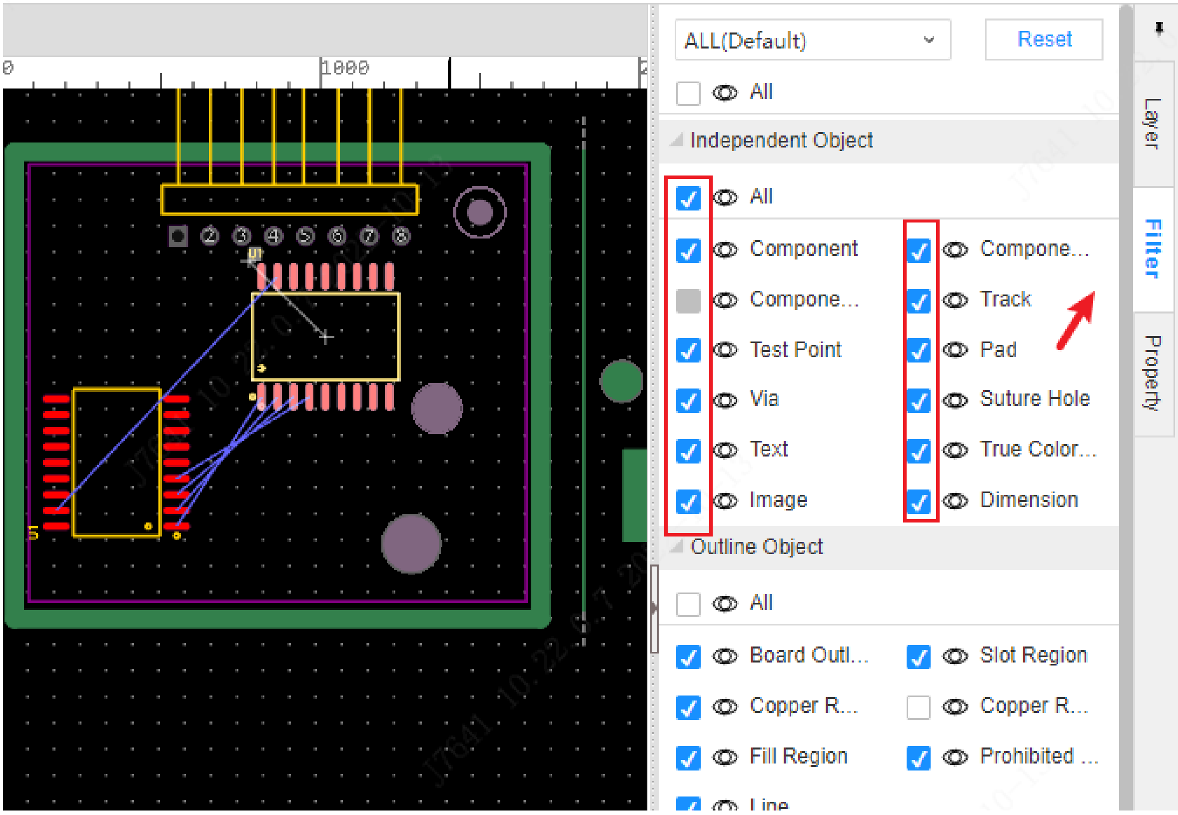
Locks the selected element so that it cannot be moved or deleted. Click the component to be locked, right-click to select Lock or select Yes in **Lock** in the left panel to lock the component.



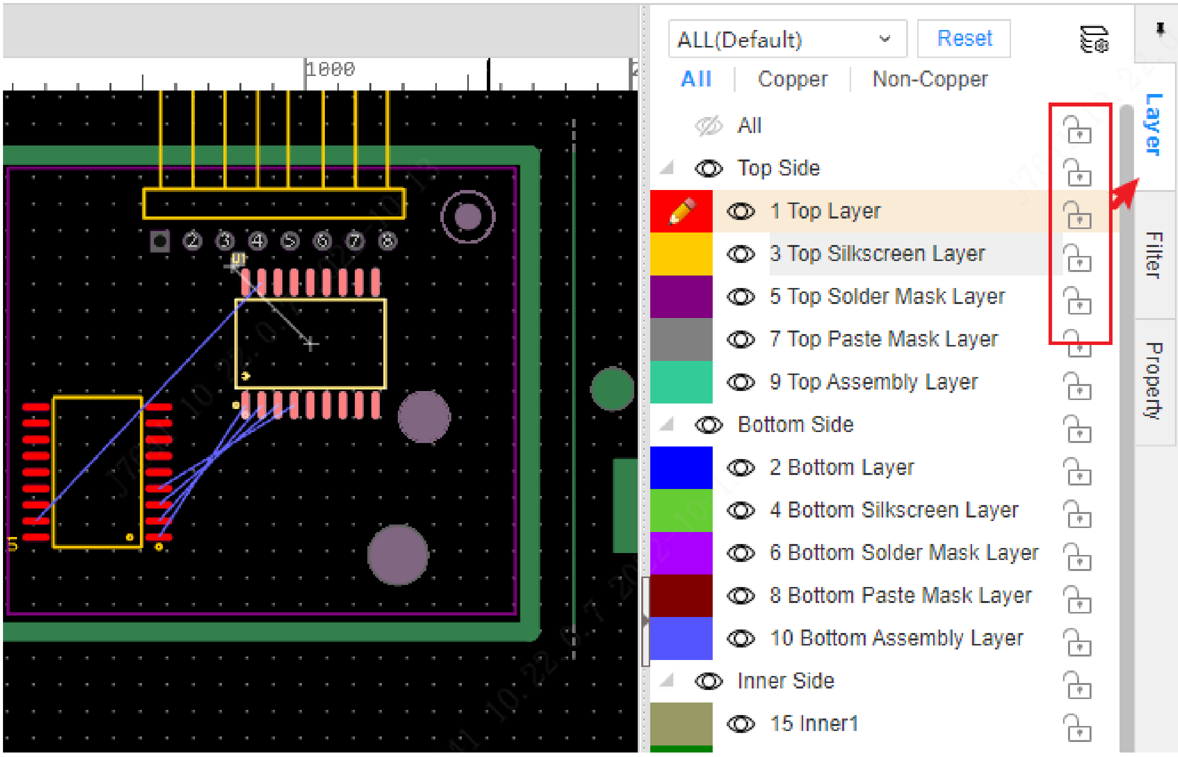
Single Element Lock: Set the Lock property on its properties to Yes.



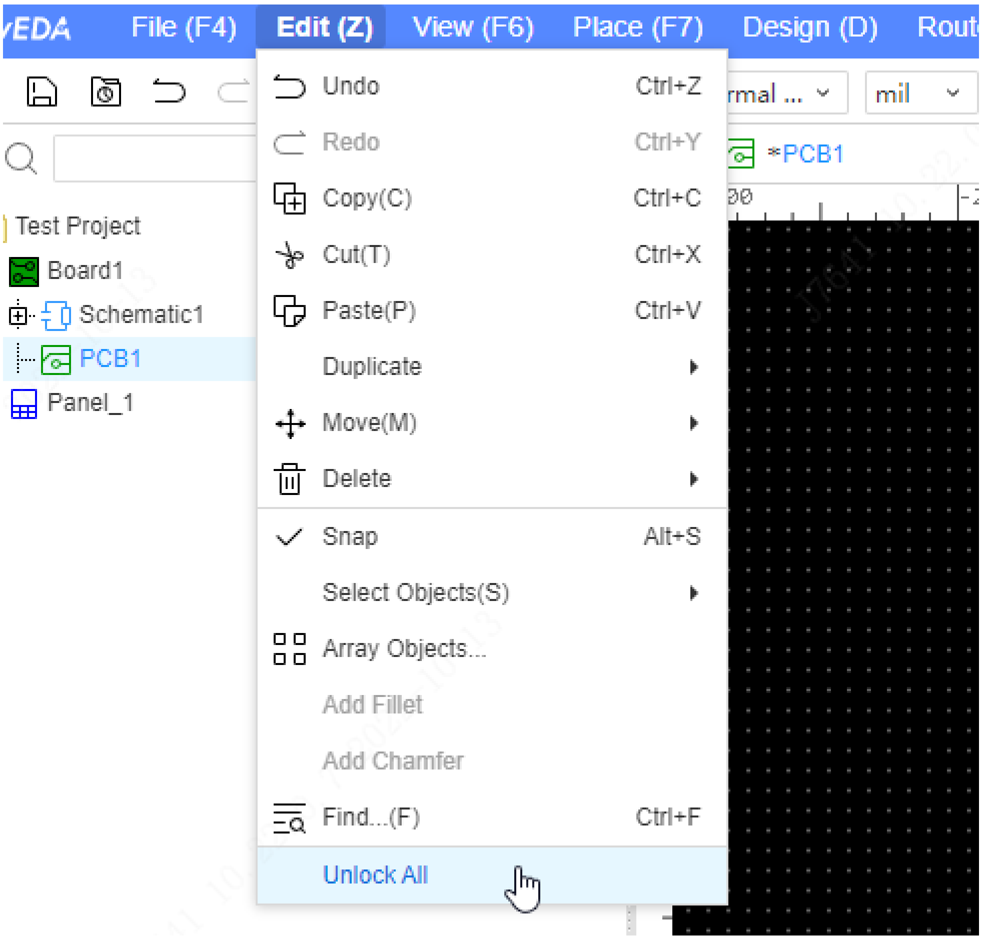
Batch lock: in the filter panel on the right, directly uncheck the object corresponding to the primitive or state to lock it;



In the layer panel, you can also lock the corresponding layers in batches.



If you want to unlock all objects, you can try unlock menu

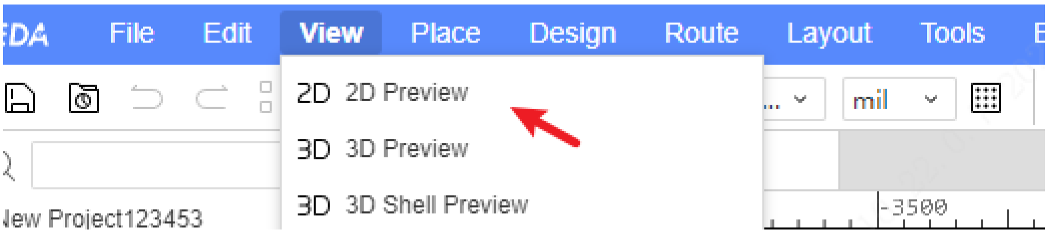


2D Preview

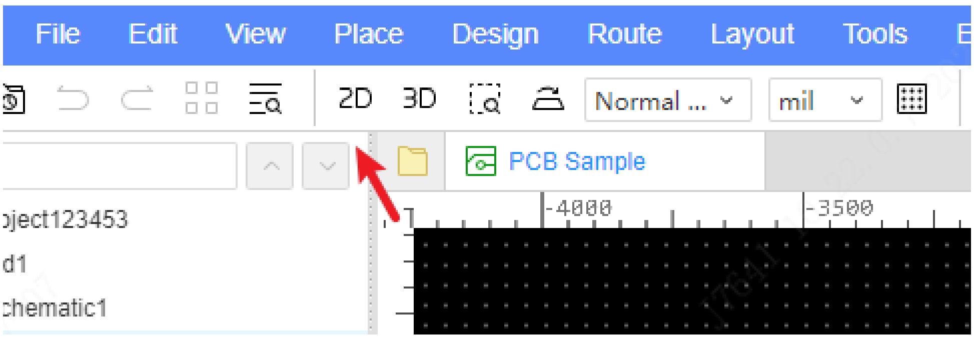
Generate a 2D preview of the designed board

Steps:

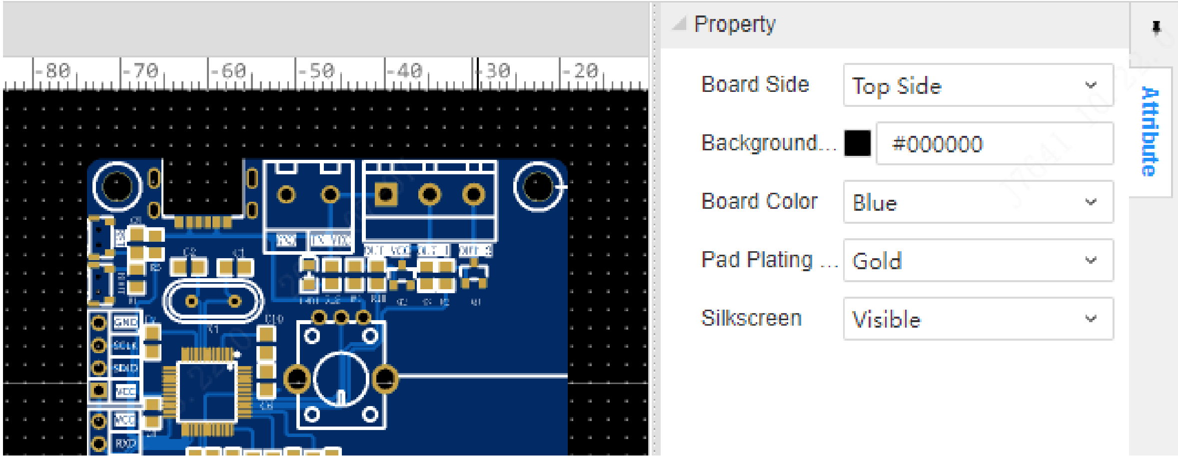
- Top Menu - View - 2D Preview



- Top Toolbar - Click on 2D Preview



Turn on the 2D previewed PCB, the 2D previewed PCB is similar to the produced PCBA.



In the right panel of the 2D interface, you can modify the color of the current board to simulate the generated PCBA.

Board Side: Preview top side or bottom side

Background Color: The background color setting of the 2D mode interface

Board Color: Color setting of PCB board

Pad Plateing Color: Preview of PCB pad painting

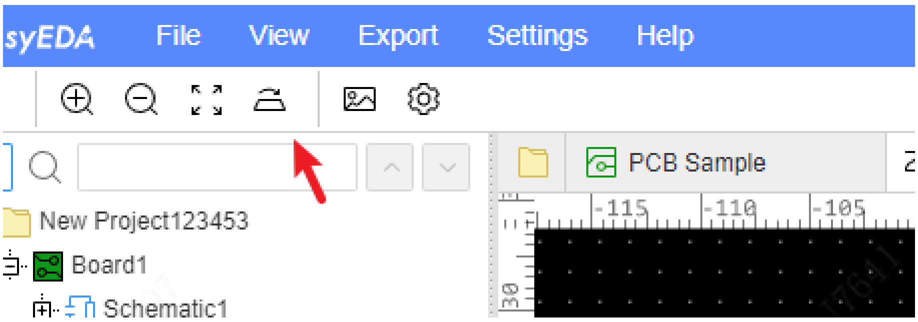
Silkscreen: Can be changed to show or hide

Note: The preview of this 2D function is a preview of the simulation image of the generated file and cannot be used as a real object. Please see the specific results according to the generated results.

Flip the board

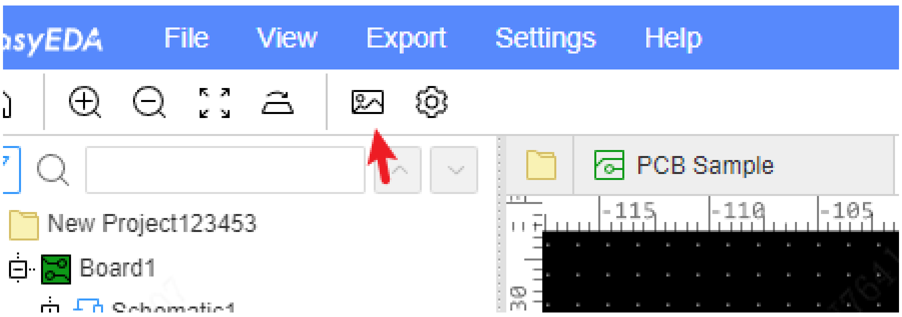
Flip the pcb board under the 2D preview of the board

Top Menu - View - Flip Board or Shortcut `Shift` + `F`



Export Image:

You can export the 2D preview as an image

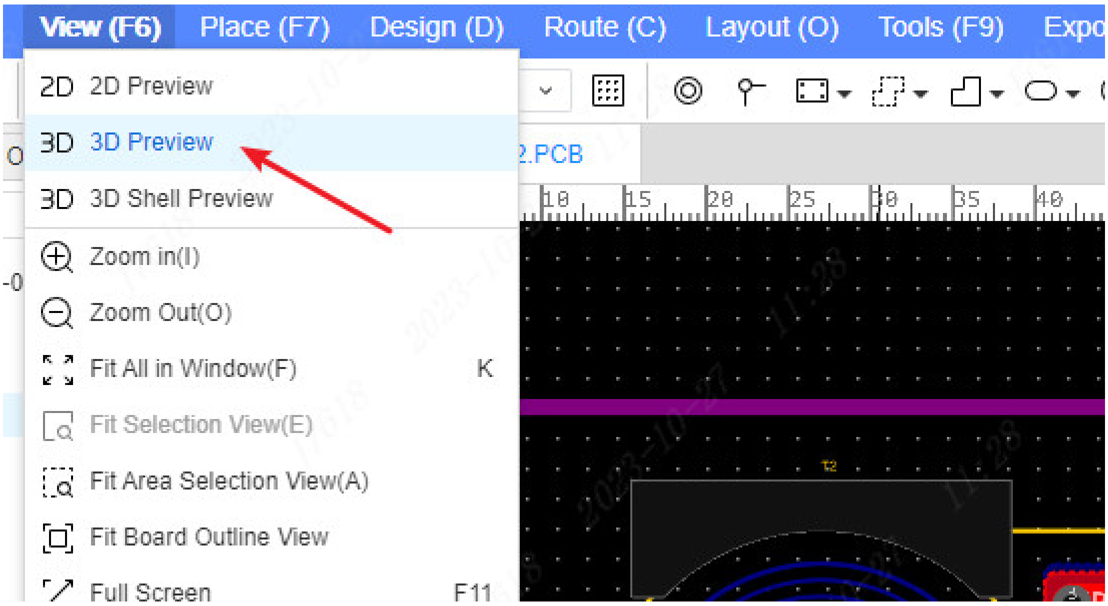


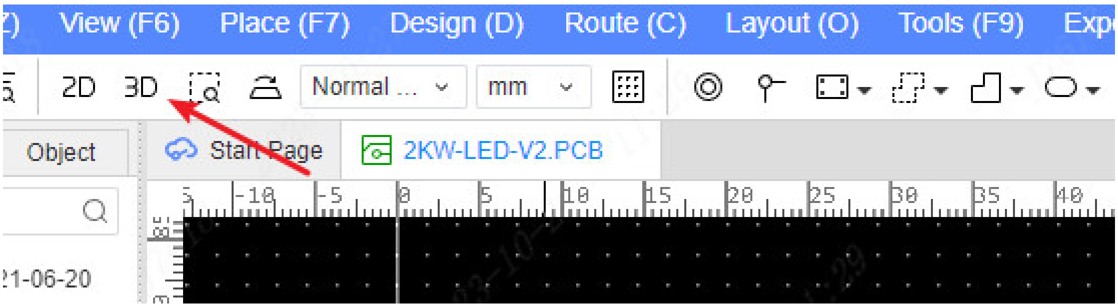
3D Preview

Generate a 3D preview of the designed board

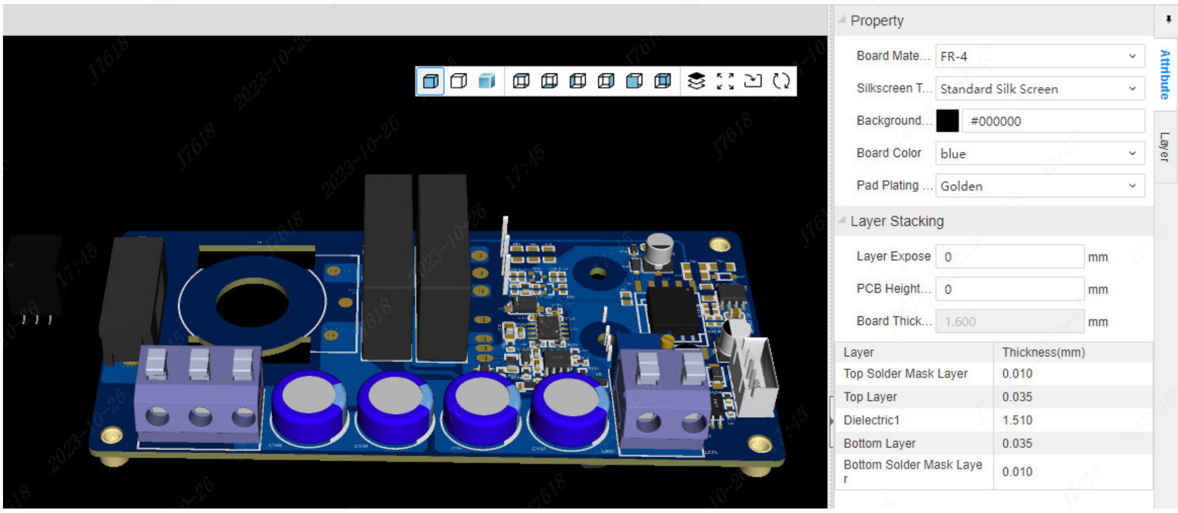
Steps:

- Top Menu - View - 3D Preview
- Top Toolbar - Click on 3D Preview icon





Turn on the 3D previewed PCB, the 3D previewed PCB is similar to the produced PCBA.

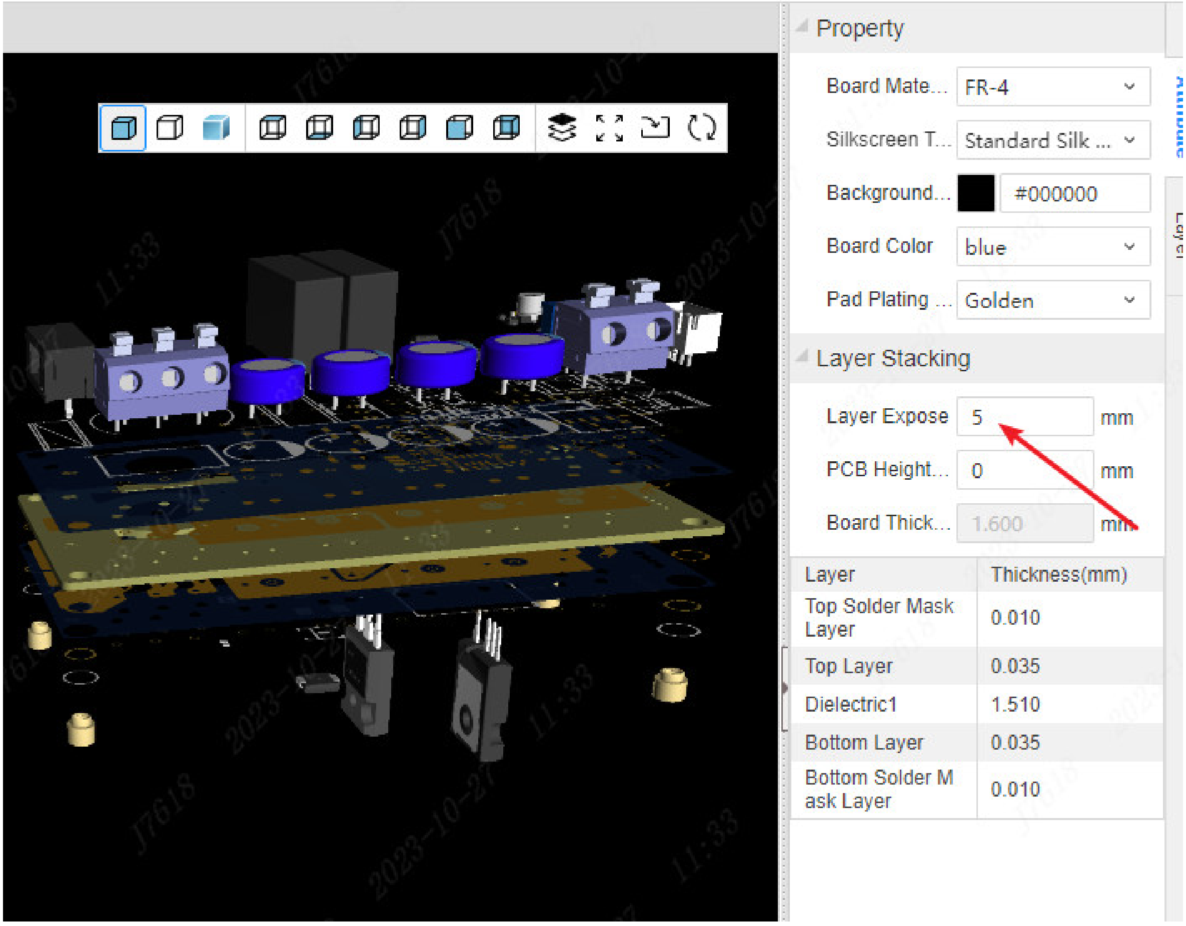


In the right panel of the 3D interface, you can modify the color of the current board to simulate the generated PCB and 3D model. If the components of the PCB are not bound to the 3D model of the component, you need to bind the 3D library first in the PCB: Top Menu - Tools - 3D Model Manager.

Property description:

- **Board Material**: The material of the PCB plate base plate, including glass fiber board, FPC, aluminum substrate, copper -based plate, ceramic plate, and iron fluorine.

- **Silkscreen Technology**: It is divided into standard silk screen and colorful silk screen. colorful silk screen are the unique process of JLC, which can produce color PCBs.
- **Background Color**: The background color setting of the preview interface
- **Board Color**: The color setting of the PCB board. Seven color settings are supported.
- **Pad Plating Color**: Color preview of PCB pad spraying, gold or silver
- **Layer Expose**: The spacing of each layer, the settings can be used to view the graphics of each layer separately.

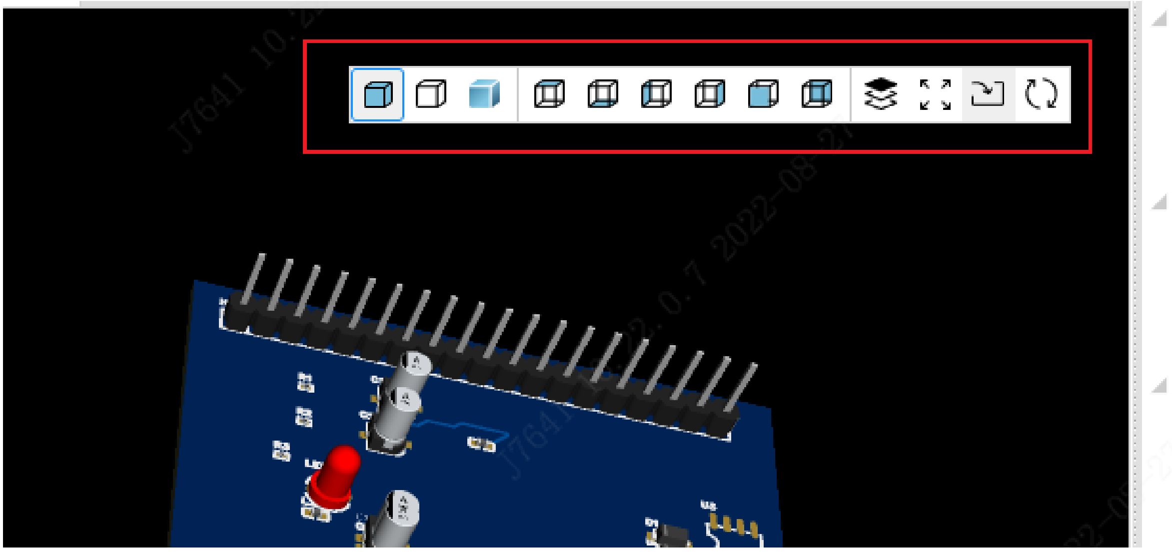


- **PCB height from the bottom of the casing**: Supports setting the height of the PCB board to the bottom of the casing and previewing the position.
- **Board thickness/layer thickness information**: This is displayed according to the layer thickness set by the layer stack in the layer manager of the PCB. If you need to modify it, please modify it in the Layer Manager: Top Menu - Tools - Layer Manager - Layer Stack.

Preview toolbar:

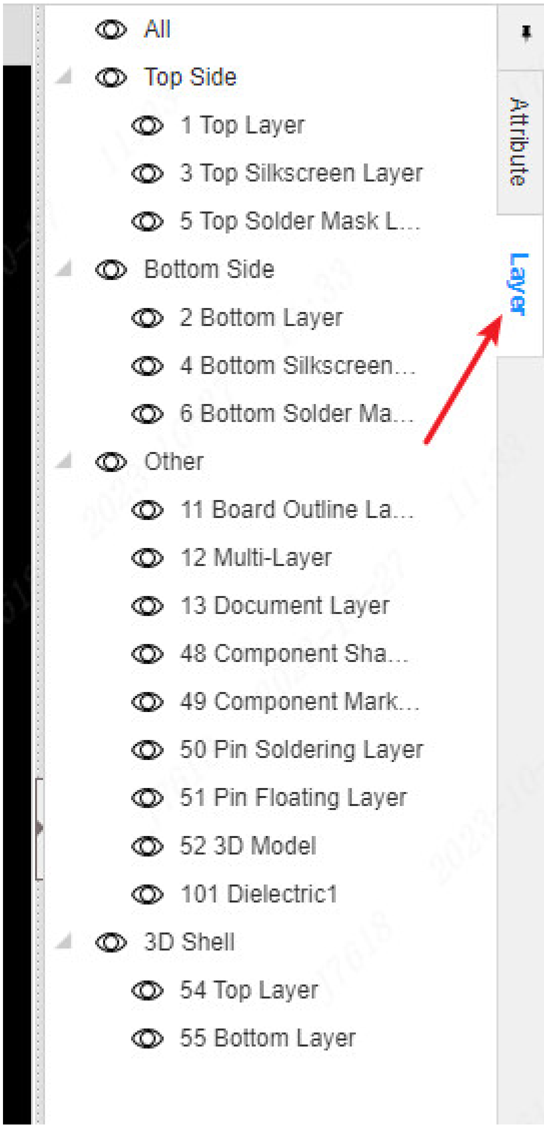
The preview toolbar supports a variety of functions. When the mouse is suspended on the icon, the corresponding function name can be prompted.support:

- Normal view, contour view, Gerber view, contour view is only used for 3D shells.
- The top surface, bottom surface, left side, right side, front, back, back
- Explosion: 3D shell preview use
- Adapt to all: The current preview adapts to the view area
- Import change: Introduction to PCB changes
- Refresh: Return to the default angle



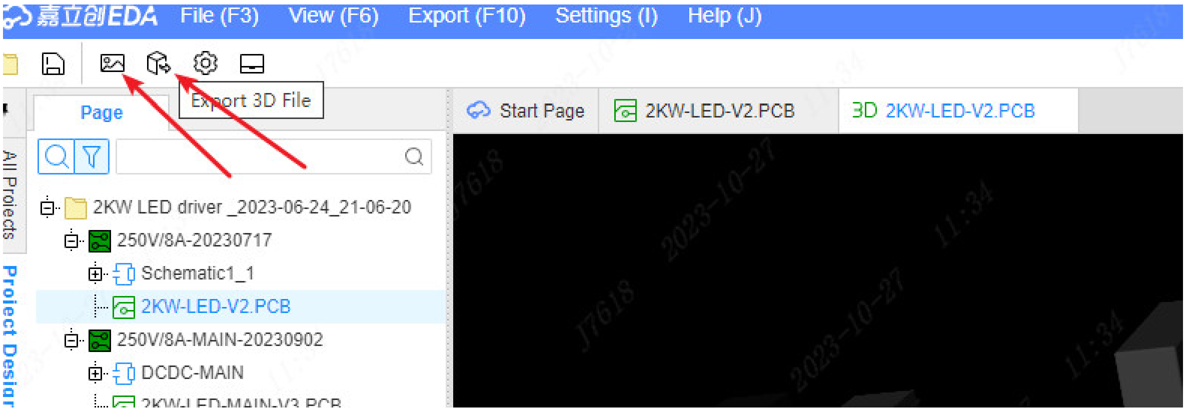
Layer switching:

Support to hide some unwanted layers on the right side. The 3D shell can also be hidden here.

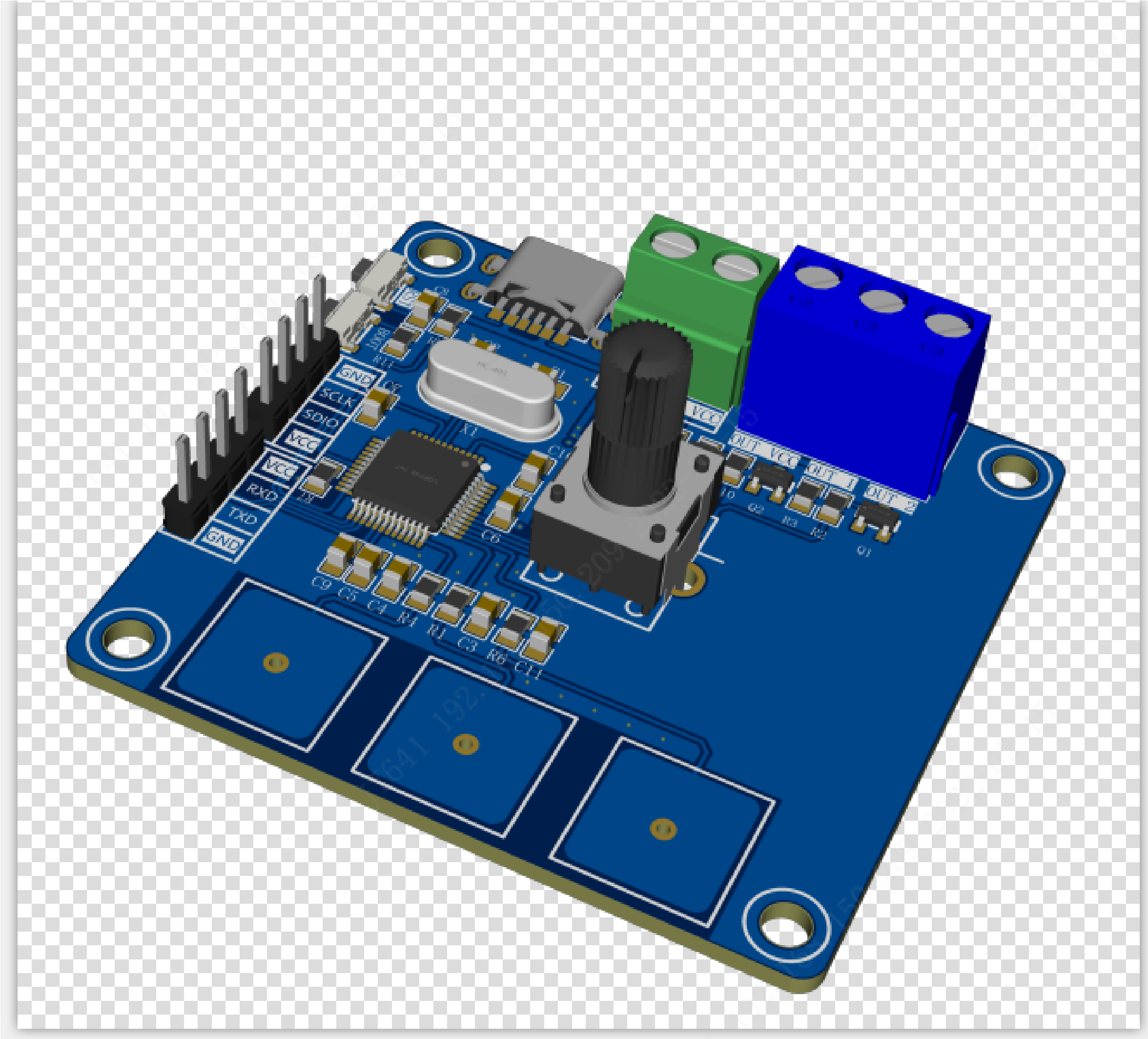


Top toolbar:

The top toolbar supports exporting pictures and 3D models.



The default width of the exported image is 2160, which is exported according to the current canvas state, and the background is transparent.



Notice:

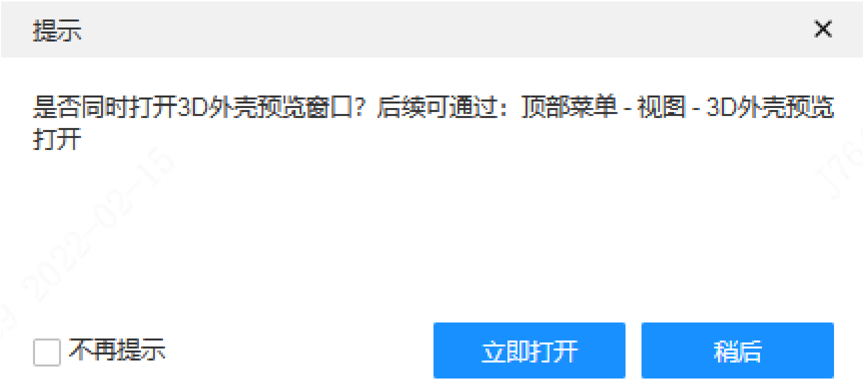
1. The preview of this 3D function is a preview of the simulation image of the generated file, which cannot be used as a real object. Please refer to the production results for specific results.
2. If you need a 3D preview of a separate window, please use the "3D Shell Preview" function.

3D Shell Preview

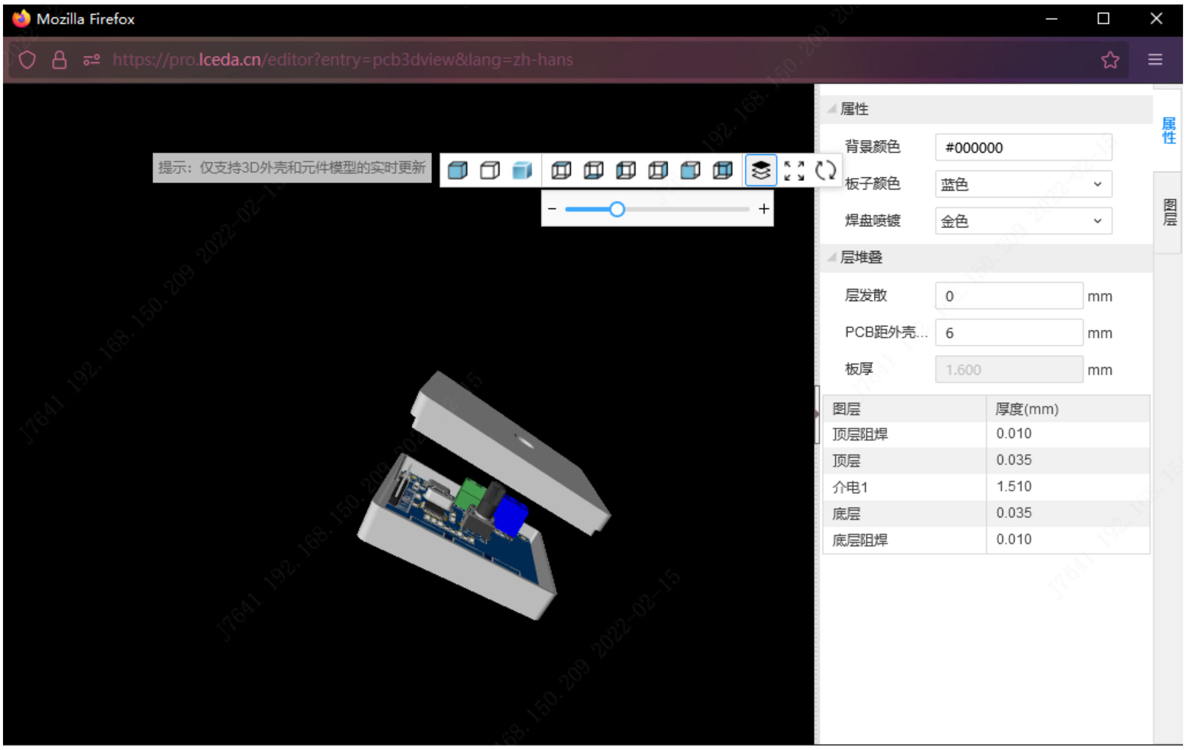
EasyEDA Pro provides a synchronous preview of the 3D shell, which is convenient for quick preview in the process of drawing the 3D shell to check whether the drawing is correct.

Entrance:

1. When designing the 3D shell, if the 3D shell preview is not opened, it will prompt whether to open the 3D shell preview, which is convenient for drawing the 3D shell.



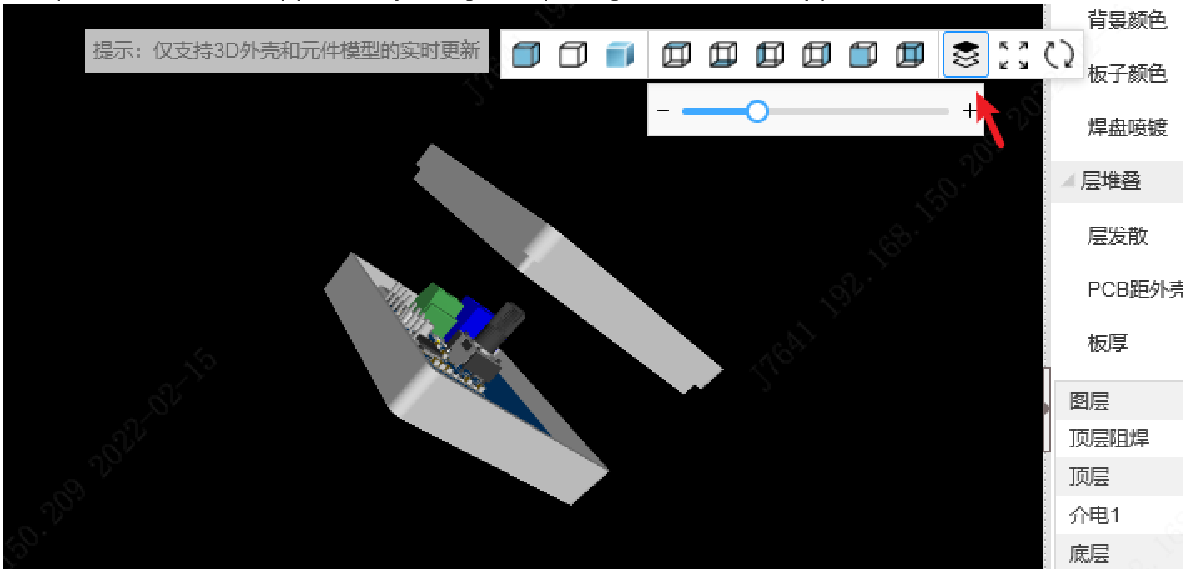
2. On the top menu - View - 3D Shell Preview.



The properties and layer settings are the same as the 3D preview settings.

Supports synchronizing the setting of "PCB height from the bottom of the casing" back to the PCB.

The preview toolbar supports adjusting the spacing between the upper and lower shells:



View Zoom

Zoom in and out of the view of the PCB interface

view zoom out

- Top Toolbar - View - Zoom Out
- Shortcut Z
- Mouse wheel back zoom

view zoom

- Top Toolbar - View - Zoom In
- Shortcut A
- Scroll the wheel forward to zoom in

fit all

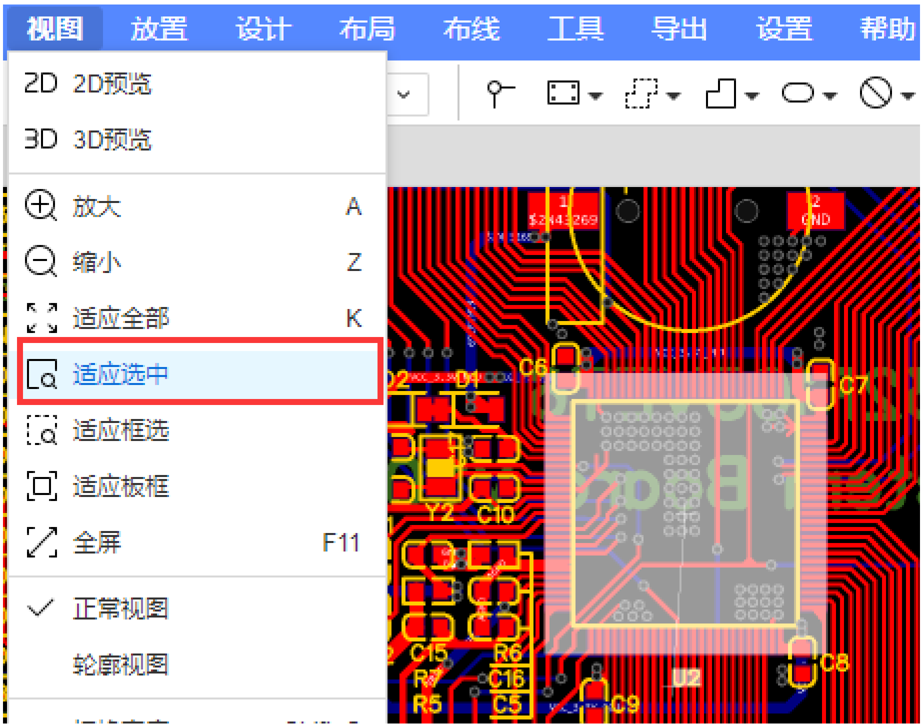
Adapt to the PCB design interface, follow the frame and components to make adaptation adjustments, even if the PCB zoom is minimal, click the shortcut key K to zoom to the current PCB.

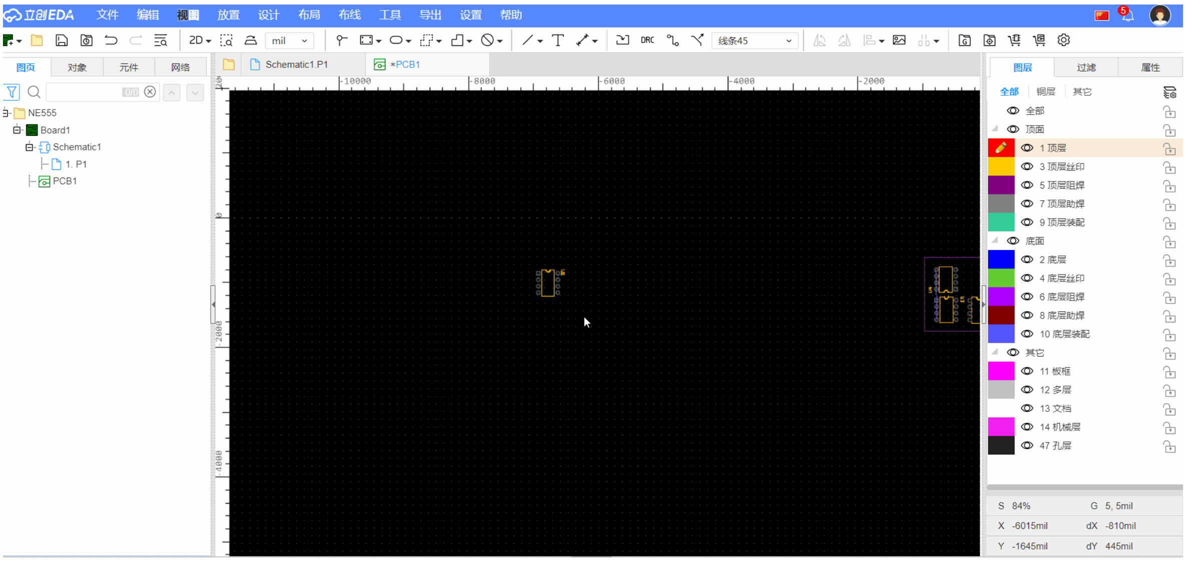
- Top Toolbar - View - Fit All
- Shortcut K

Adapt to selected

Make the selected element larger

- Select Device - Top Menu - View - Fit Selected
- select device - right mouse button - adapt to selection

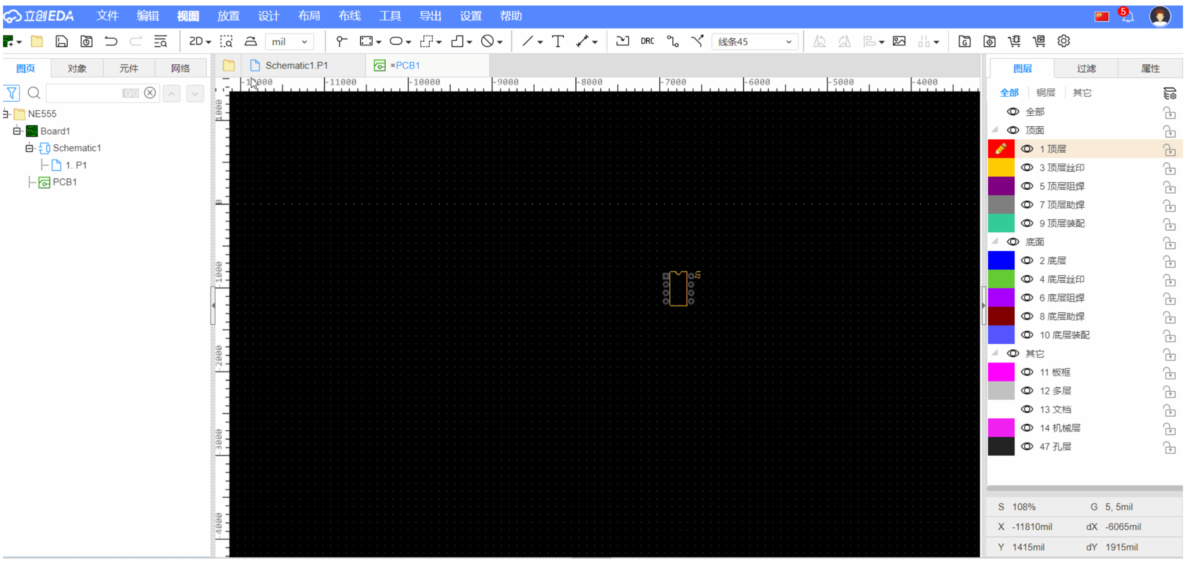




Adapt to box selection

Enlarge the selected area

- Top Menu - Edit - Fit Box Selection



Fit to board frame

Scale to fit the board frame.

- Top Menu - View - Fit to Board Frame

full screen

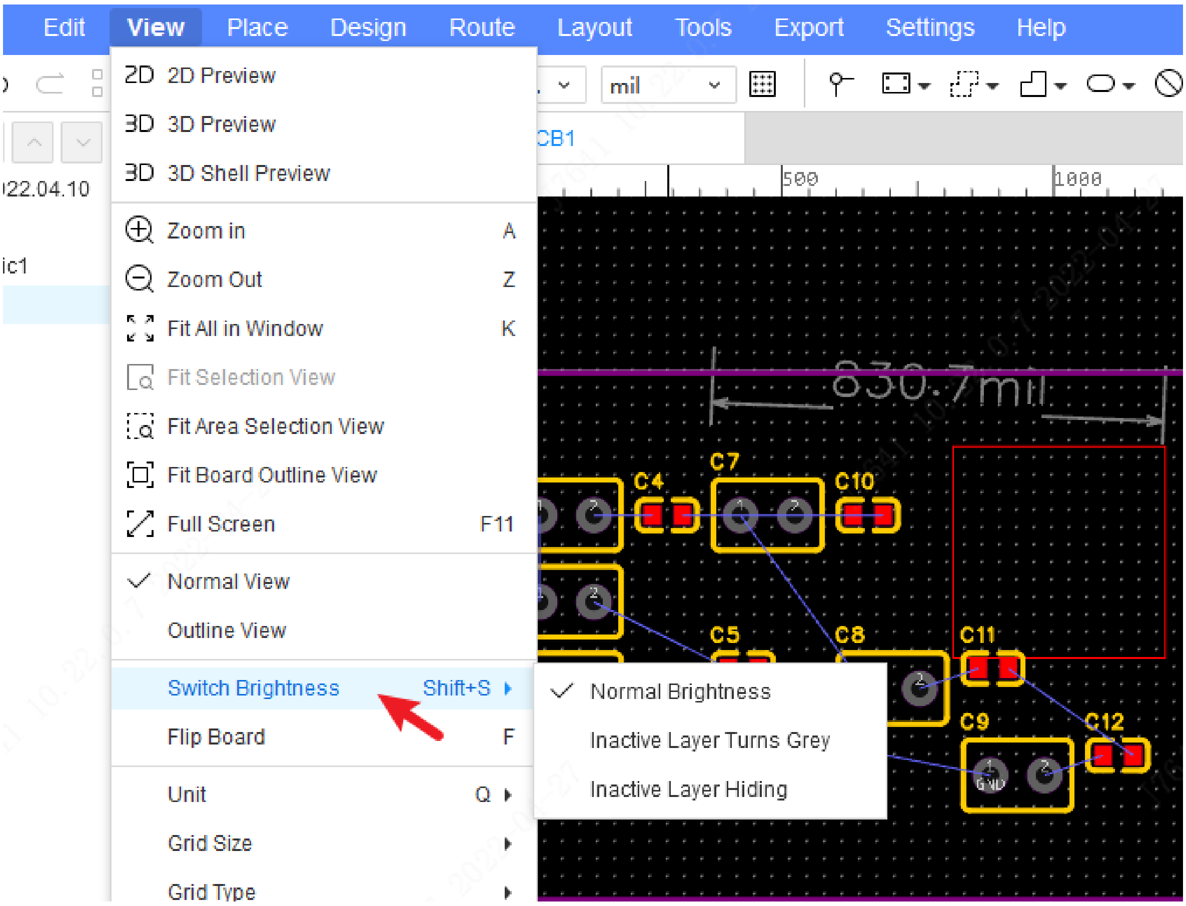
Enlarge the editor as a whole according to the size of the screen.

- Top Menu - View - Full Screen
- Shortcut F11

Switch Brightness

Toggles the brightness of the current layer, dims the elements of other layers, and displays the elements of the current layer alone.

- Top Menu - View - Switch Brightness
- Use shortcut keys `Shift` + `S`



Flip Board

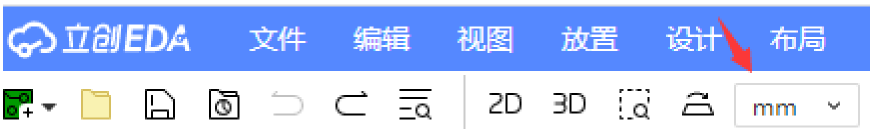
Turn the PCB over for easy routing,

- Top Menu - Edit - Flip Board
- Shortcut key `F`

Unit Switch

Switch the unit setting of the PCB interface. Currently, only two units of mm and mil are supported for switching.

- Top Menu - View - Unit
- Shortcut `Q`
- Top toolbar



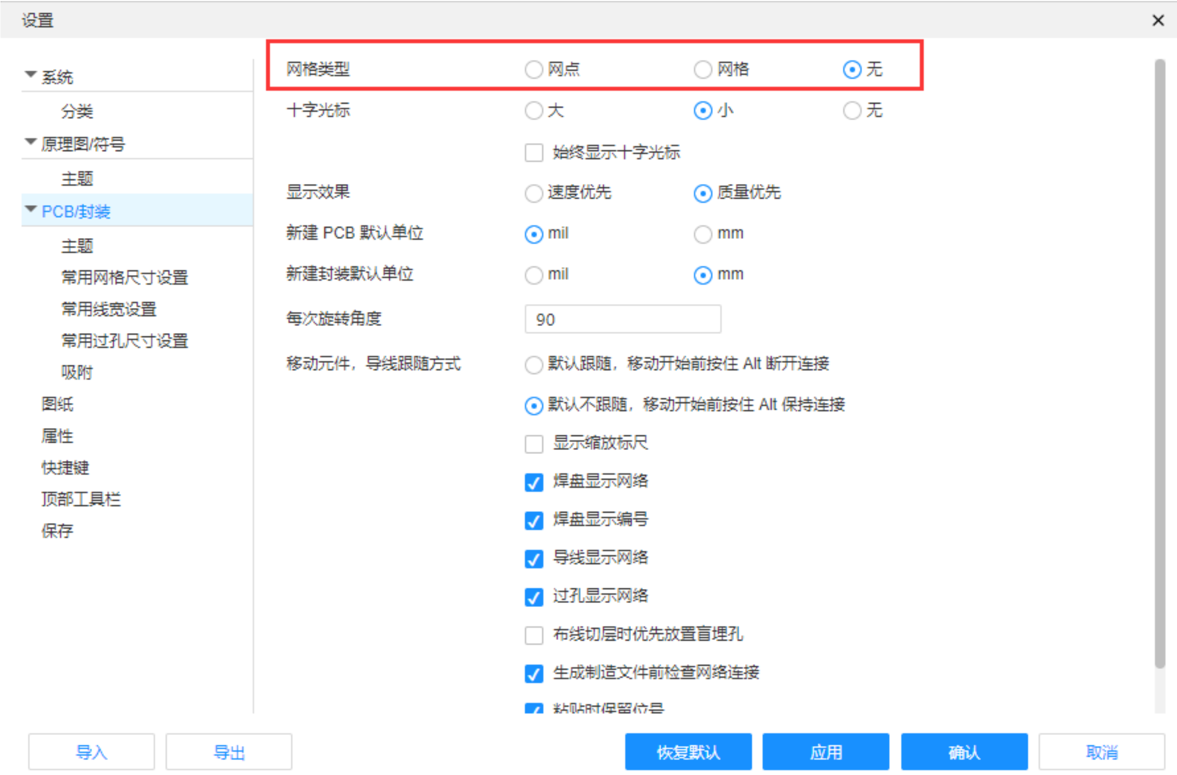
Grid Size

grid size

Modify the size of the PCB interface grid.

Grid switching

Toggle Grid to Grid, Dot, or None.



Modify grid size

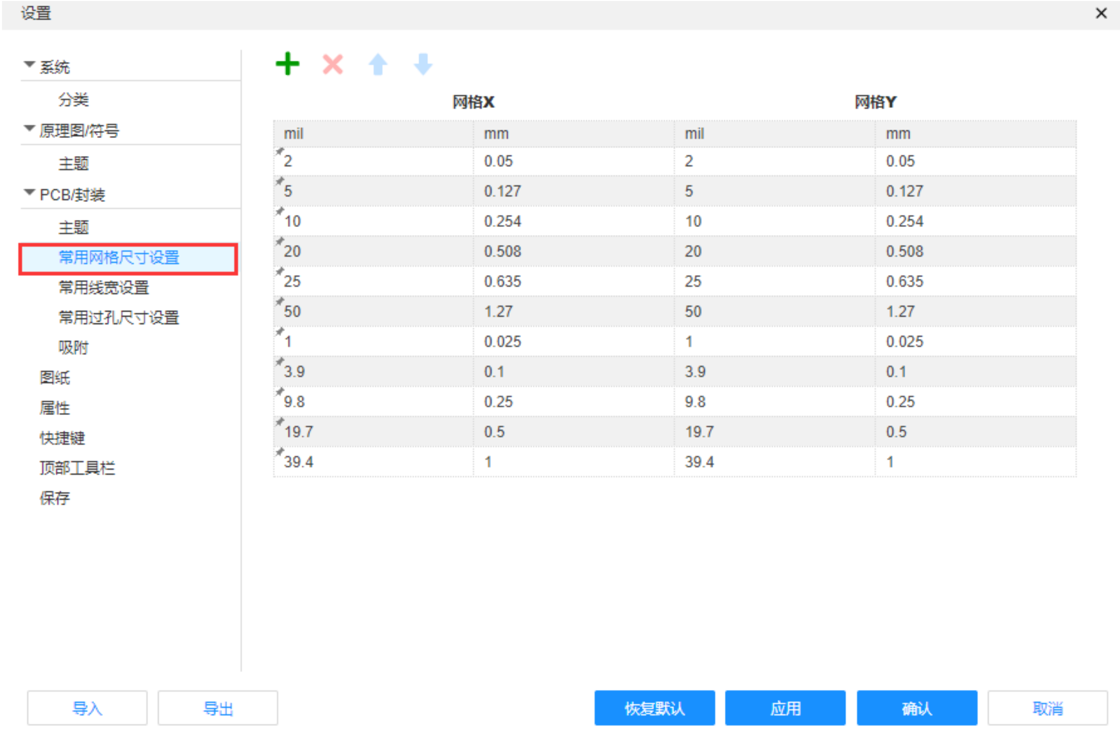
- Top Menu - View - Grid
- Right mouse button - Grid


视图 放置 设计 布局 布线 工具 导出 设置
2D 2D预览
3D 3D预览
放大 A
缩小 Z
适应全部 K
适应选中
适应框选
适应板框
全屏 F11
正常视图
轮廓视图
切换亮度 Shift+S
翻转板子 Shift+F
单位 Q
网格
飞线
高亮
顶部工具栏
左侧面板
右侧面板
底部面板 S
X, Y: 2, 2mil(0.05, 0.05mm)
X, Y: 5, 5mil(0.127, 0.127mm)
X, Y: 10, 10mil(0.254, 0.254mm)
X, Y: 20, 20mil(0.508, 0.508mm)
X, Y: 25, 25mil(0.635, 0.635mm)
X, Y: 50, 50mil(1.27, 1.27mm)
X, Y: 1, 1mil(0.025, 0.025mm)
X, Y: 3.9, 3.9mil(0.1, 0.1mm)
X, Y: 9.8, 9.8mil(0.25, 0.25mm)
X, Y: 19.7, 19.7mil(0.5, 0.5mm)
X, Y: 39.4, 39.4mil(1, 1mm)
常用网格设置...

Common grid settings

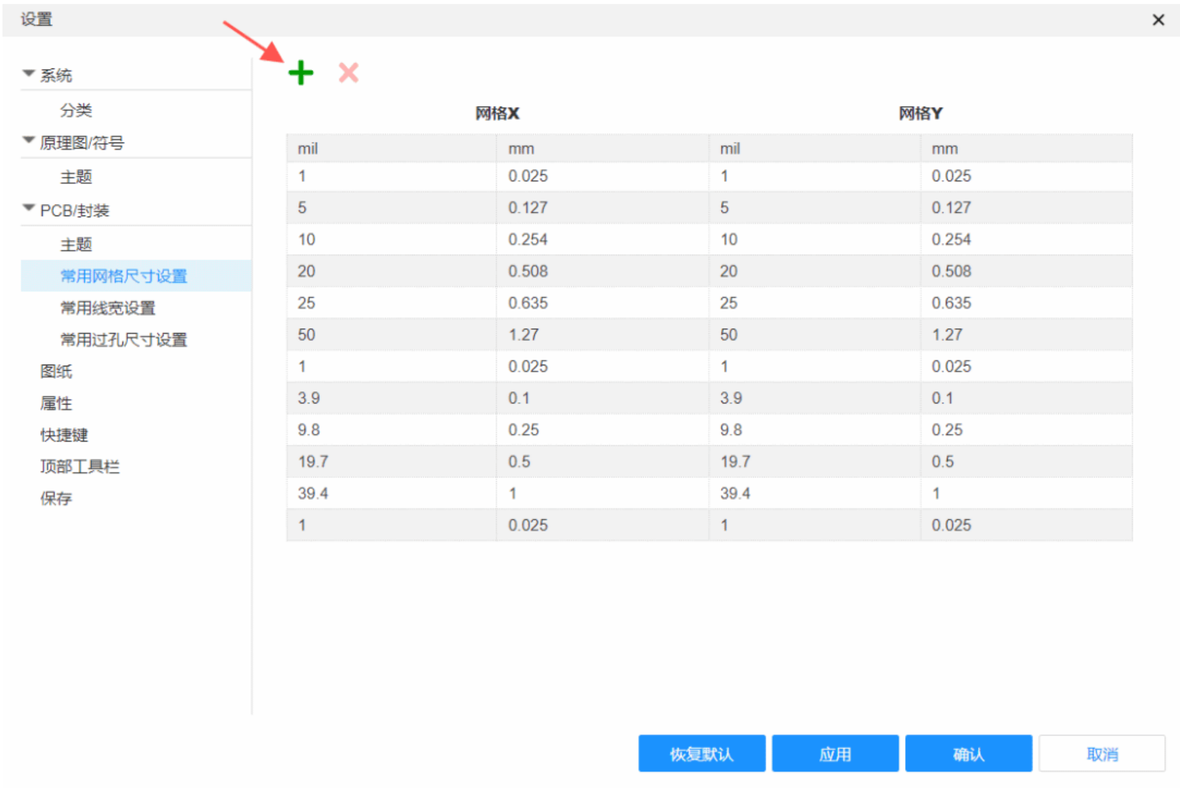
- Top Menu - Settings - PCB/footprint - Common Grid Sizes
- Top Menu - View - Grid - Common Grid Sizes


视图
放置
设计
布局
布线
工具
导出
设置
2D 2D预览
3D 3D预览
放大 A
缩小 Z
适应全部 K
适应选中
适应框选
适应板框
全屏 F11
正常视图
轮廓视图
切换亮度 Shift+S
翻转板子 Shift+F
单位 Q
网格
飞线
高亮
顶部工具栏
左侧面板
右侧面板
底部面板 S
X, Y: 2, 2mil(0.05, 0.05mm)
X, Y: 5, 5mil(0.127, 0.127mm)
X, Y: 10, 10mil(0.254, 0.254mm)
X, Y: 20, 20mil(0.508, 0.508mm)
X, Y: 25, 25mil(0.635, 0.635mm)
X, Y: 50, 50mil(1.27, 1.27mm)
X, Y: 1, 1mil(0.025, 0.025mm)
X, Y: 3.9, 3.9mil(0.1, 0.1mm)
X, Y: 9.8, 9.8mil(0.25, 0.25mm)
X, Y: 19.7, 19.7mil(0.5, 0.5mm)
X, Y: 39.4, 39.4mil(1, 1mm)
常用网格设置...

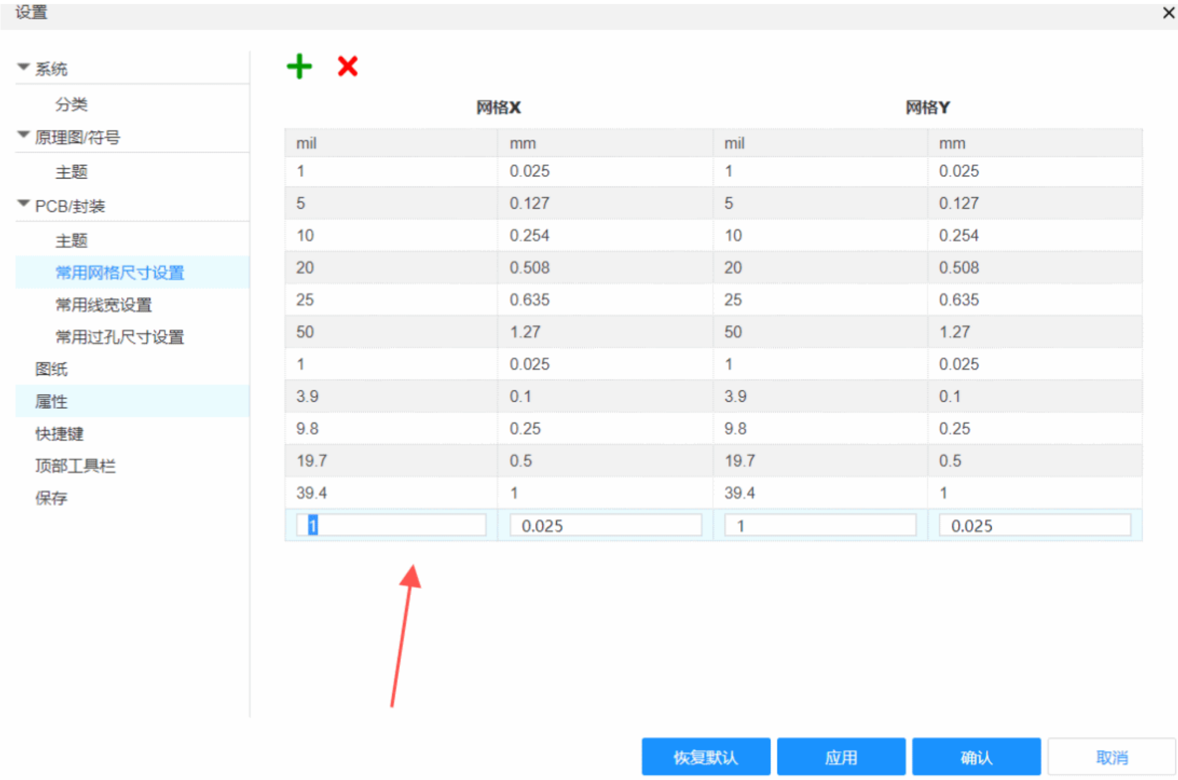


Add grid settings

In the grid settings popup, click the "+" sign at the top to add

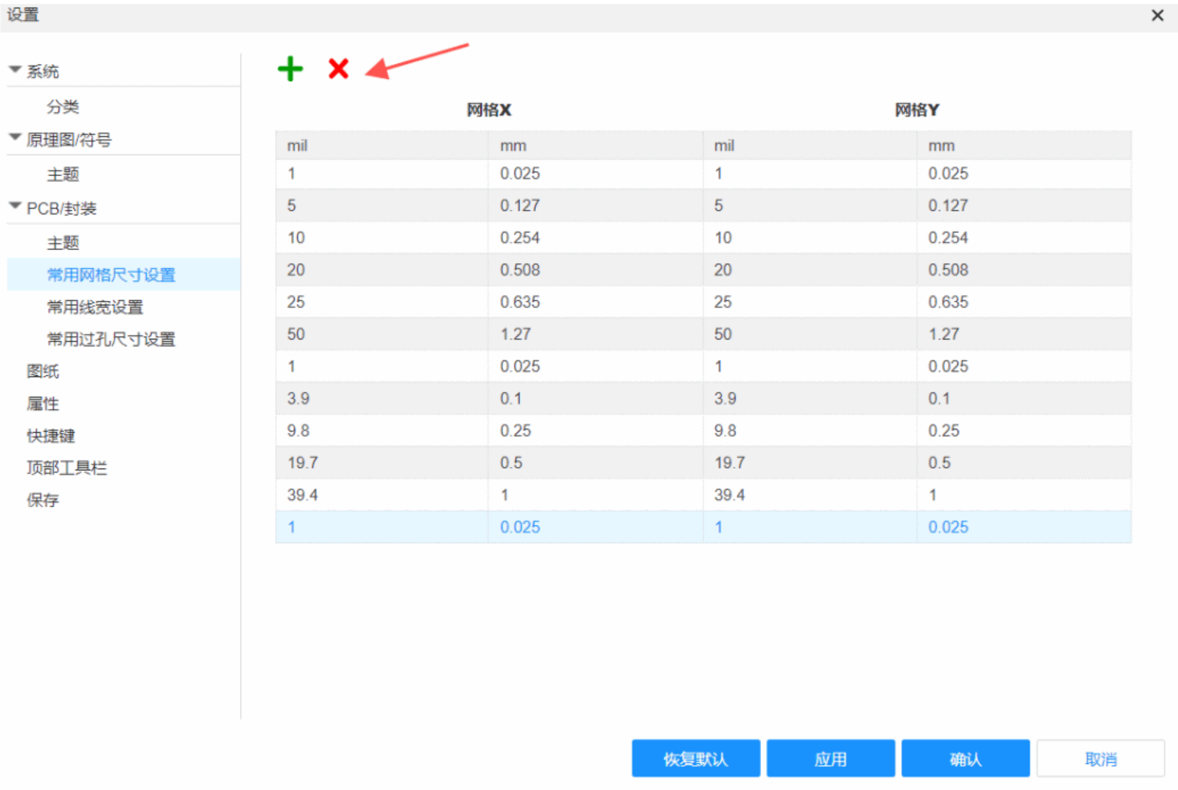


Double-click the grid property setting you just added, and click OK after modification to save the current configuration.



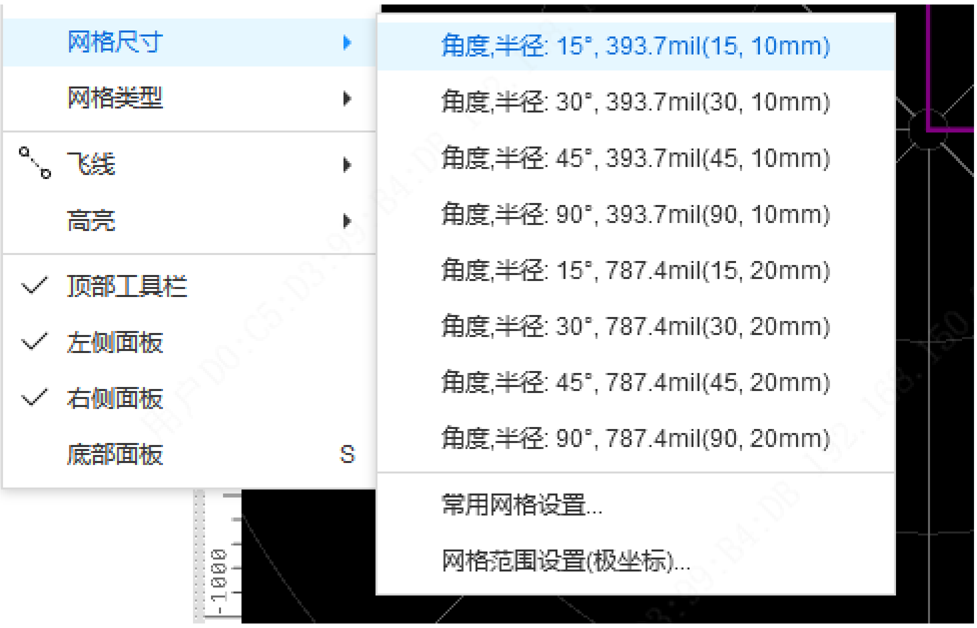
delete grid settings

Select the grid size setting you want to delete, and click the "X" at the top of the grid settings popup to delete

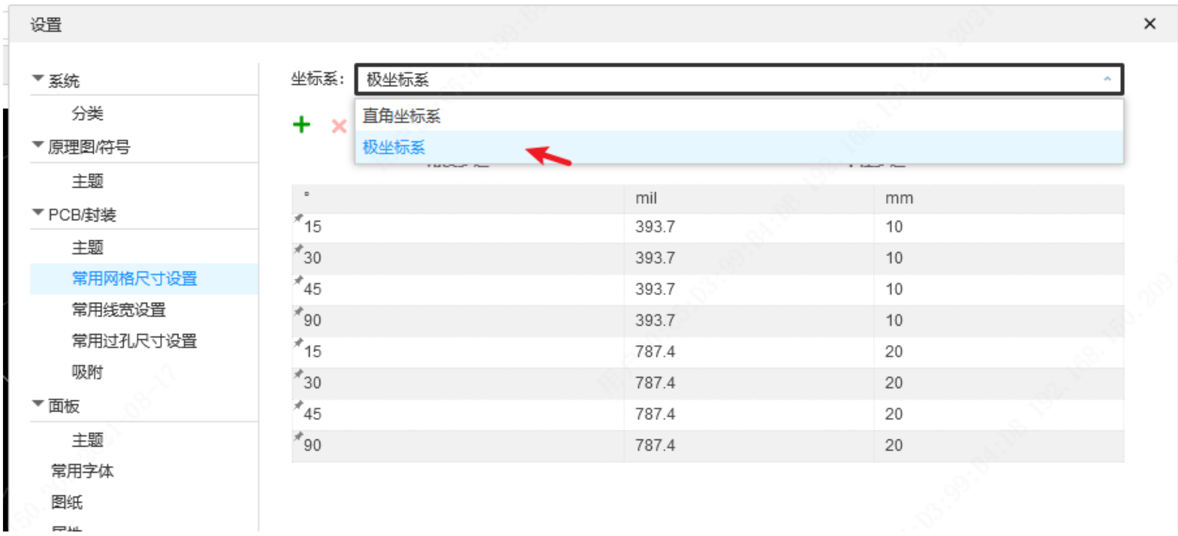


Polar coordinate system

Polar grid size list



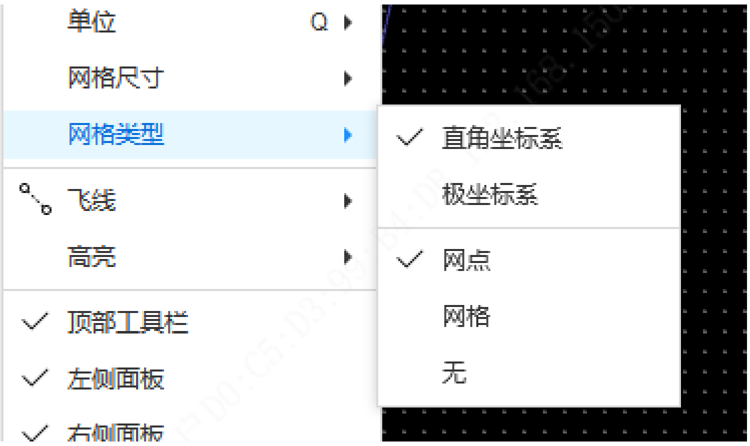
Supports setting the range size of polar coordinate system



Grid Type

grid type

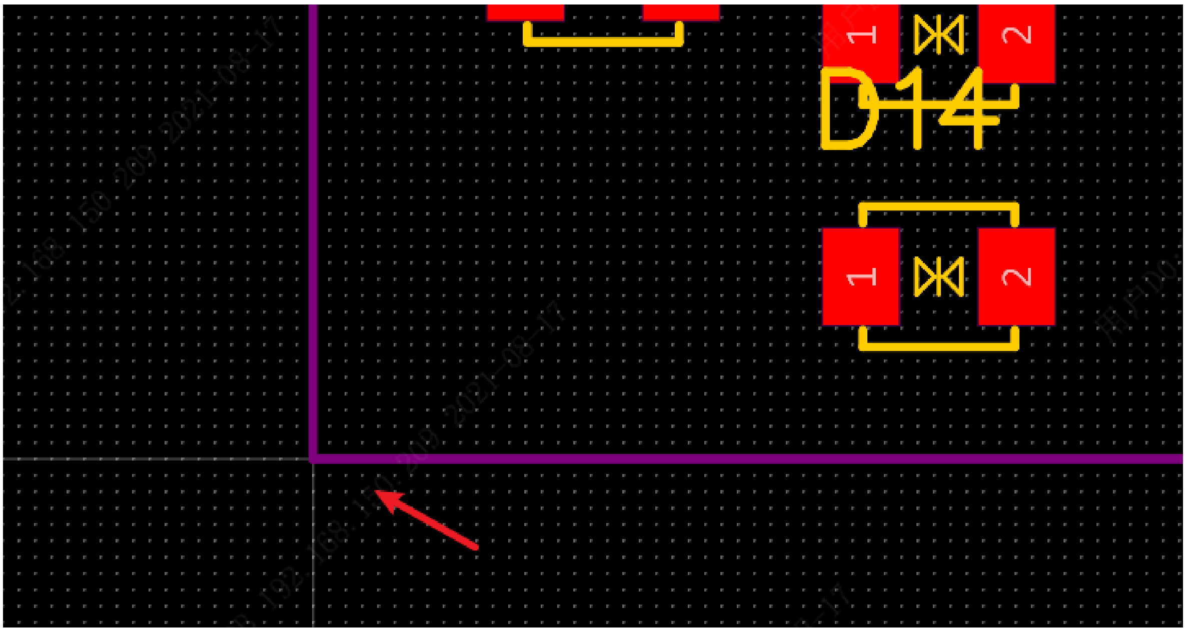
Supports switching between two grid types: Cartesian coordinate system, polar coordinate system



You can also set whether the grid in the canvas is grid lines, grid points, or no grid display.

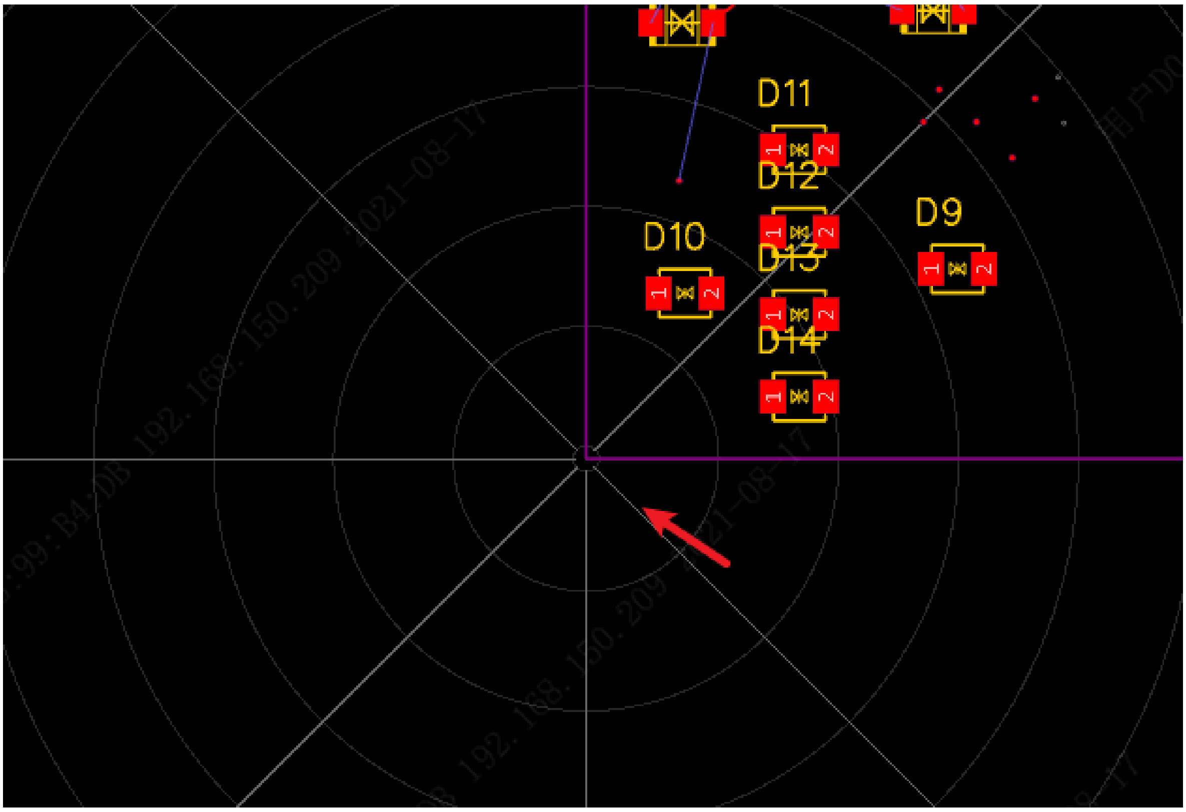
Cartesian coordinate system

The Cartesian coordinate system is the most commonly used, and ordinary boards can use this coordinate system to complete the design.



Polar coordinate system

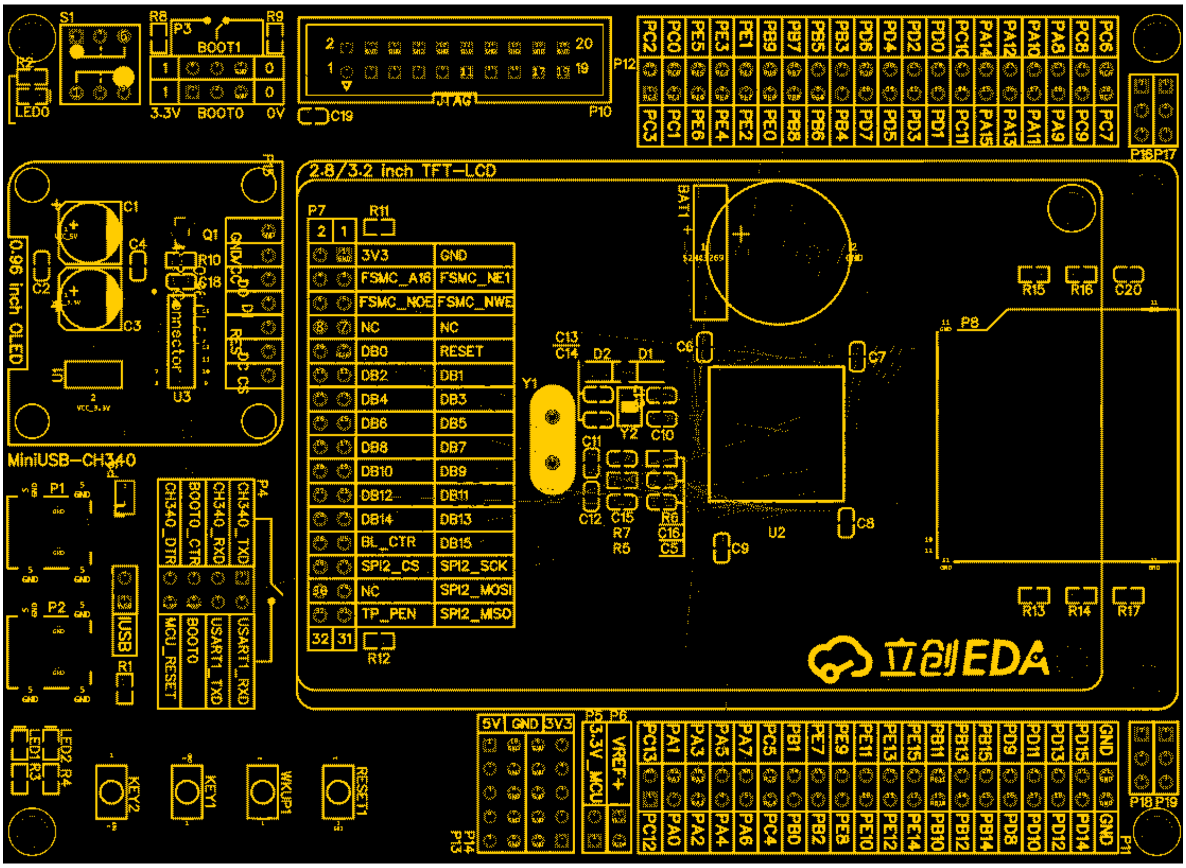
The polar coordinate system is a coordinate system that takes the center of the circle as the origin and expands outward in a fan shape. It is generally used when designing annular boards and layouts.



Ratline

Flying leads are generated based on the same network.

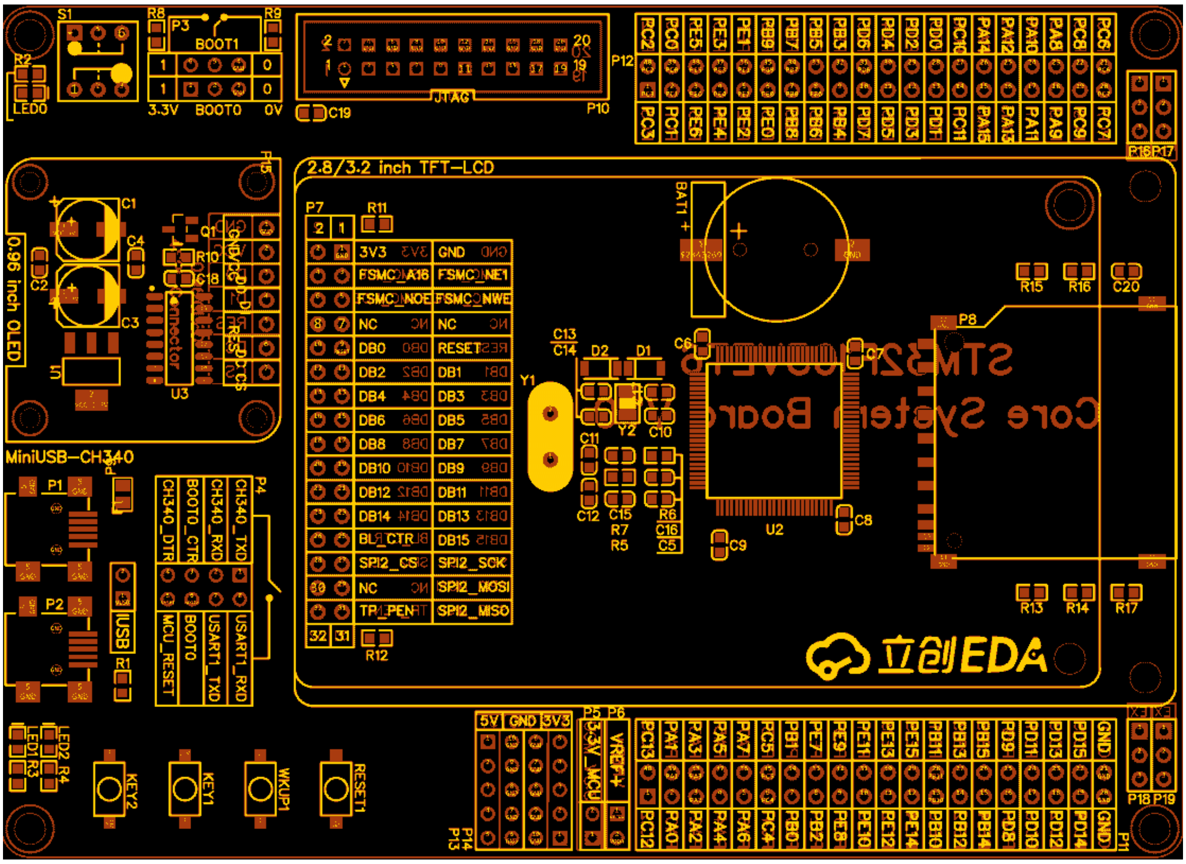
Flying leads appear when two footprints have the same pad network, indicating that the two pads can be connected by wires. EasyEDA Pro supports hide and display of device flying leads



Before the ratline is not hidden

hide

- **Top Menu - View - ratlines - Hide All**

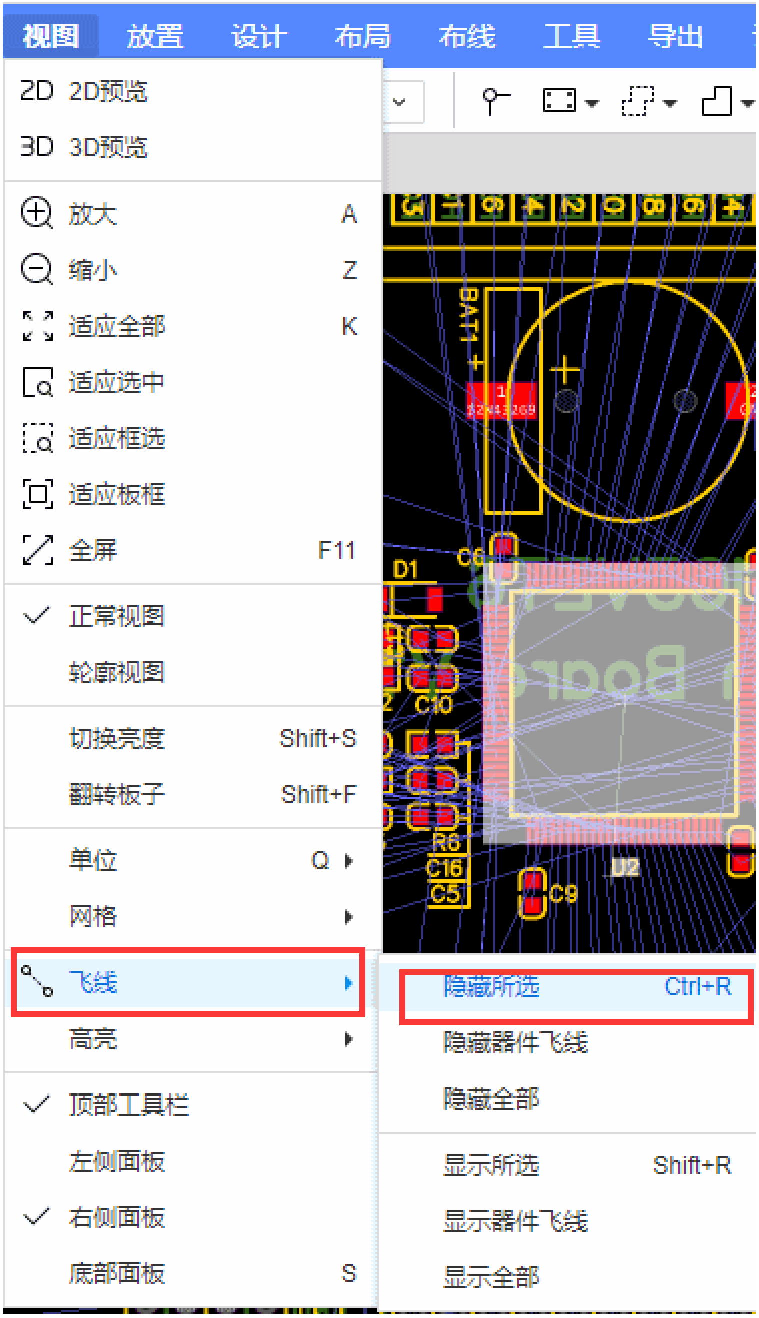


show

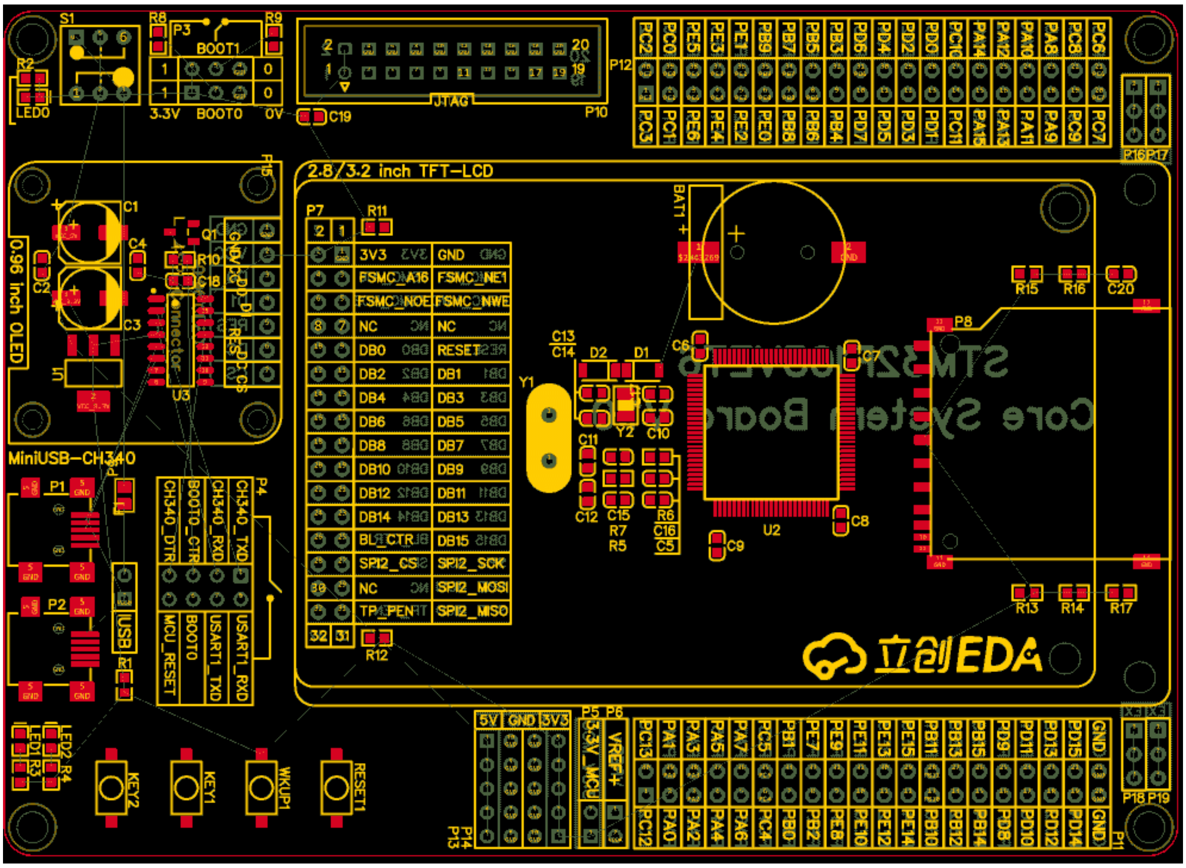
- **Top Menu - View - ratline - Display**, you can display the hidden spin.

Single device flying lead hidden

- **Selected Device - Top Menu - Flying Leads - Hide Selected**, after clicking, the flying leads of the selected device will be hidden separately.



视图
放置
设计
布局
布线
工具
导出
2D 2D预览
3D 3D预览
放大 A
缩小 Z
适应全部 K
适应选中
适应框选
适应板框
全屏 F11
正常视图
轮廓视图
切换亮度 Shift+S
翻转板子 Shift+F
单位 Q
网格
飞线
高亮
顶部工具栏
左侧面板
右侧面板
底部面板 S
隐藏所选 Ctrl+R
隐藏器件飞线
隐藏全部
显示所选 Shift+R
显示器件飞线
显示全部



show all flying leads

Highlight

EasyEDA Pro supports device and network highlight selection.

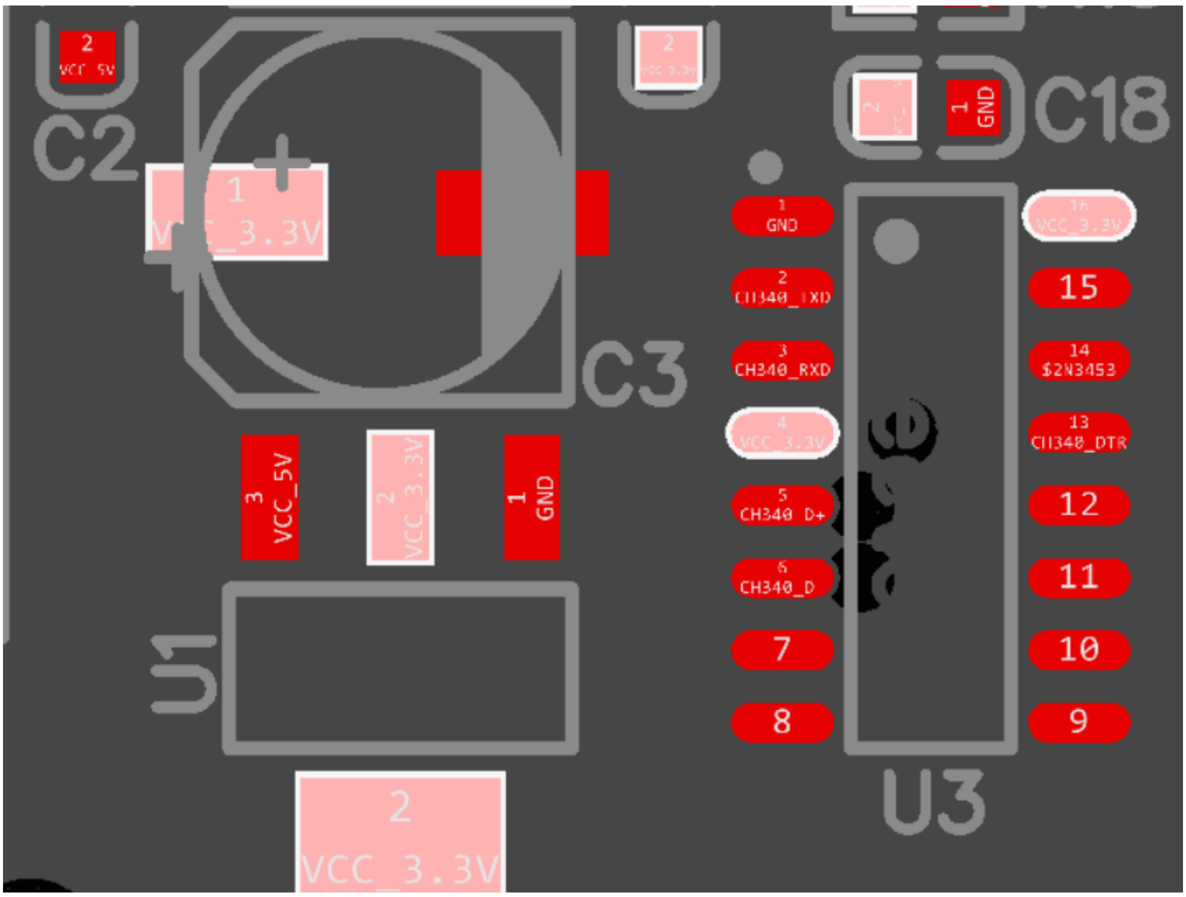
Highlight operation steps:

- select pad or net - top menu - view - highlight - highlight net
- Select the element to be highlighted - Right click - Highlight
- Shortcut H

Highlight all


视图
放置
设计
2D 2D预览
3D 3D预览
放大 A
缩小 Z
适应全部 K
适应选中
适应框选
适应板框
全屏 F11
正常视图
轮廓视图
切换亮度 Shift+S
翻转板子 Shift+F
单位 Q
网格
飞线
高亮
顶部工具栏
左侧面板
右侧面板
底部面板 S

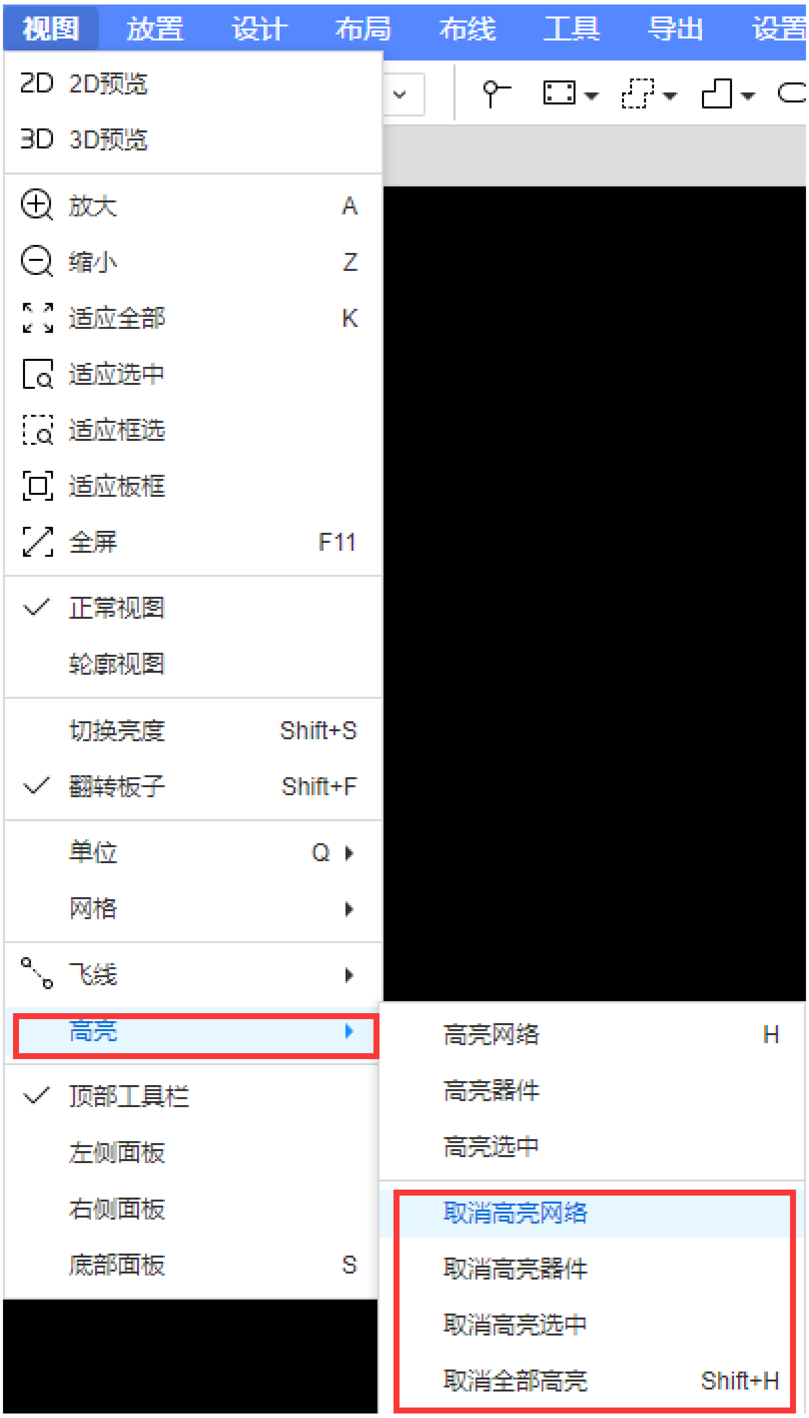
You can highlight all the same nets selected in the PCB.



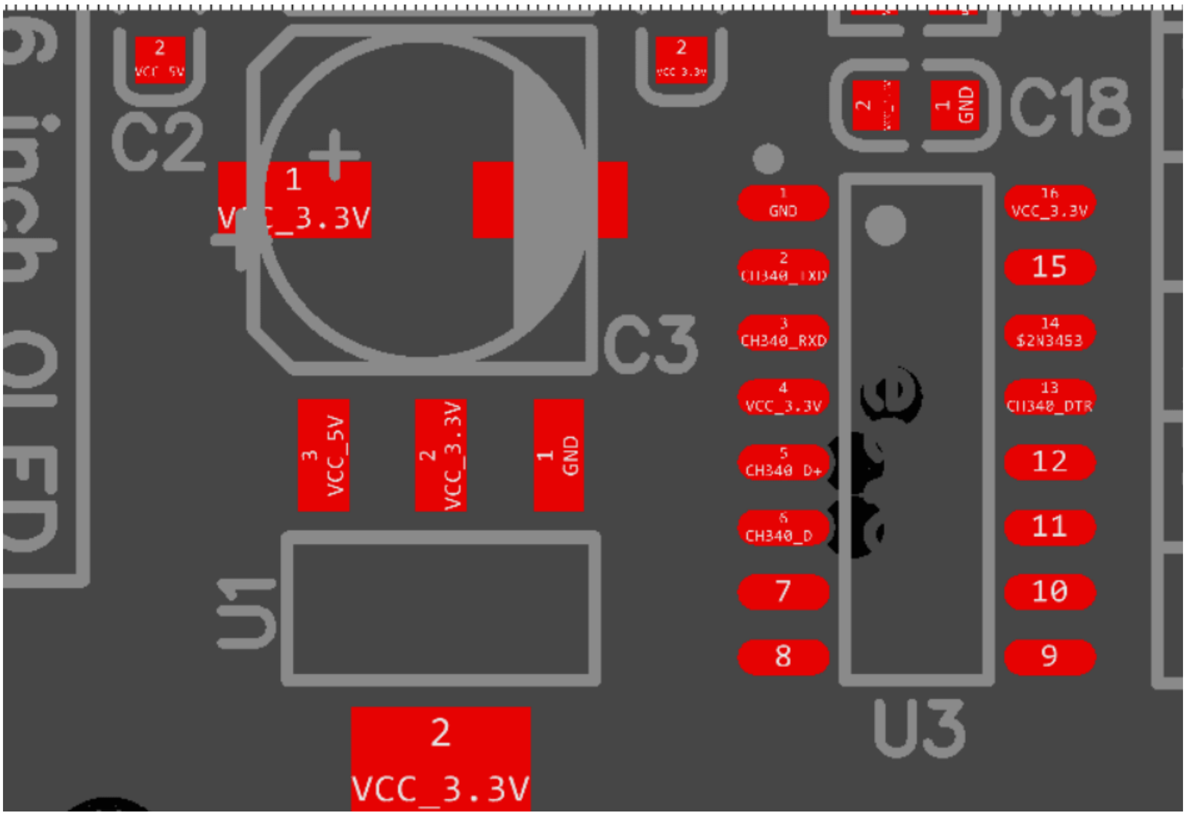
un-highlight

- **Select the pad you want to cancel the net - top menu - view - highlight - unhighlight all or right click - highlight - unhighlight all**

You can also use the shortcut key shift + H to cancel all highlighting



unhighlight all



Highlight devices

- Select Device - Top Menu - View - Highlight - Highlight Device
- Selected device - right click - highlight - highlight device
- Left View - Components - Find the corresponding device and double click


视图
放置
设计
布局
布线
工具
导出
2D 2D预览
3D 3D预览
放大 A
缩小 Z
适应全部 K
适应选中
适应框选
适应板框
全屏 F11
正常视图
轮廓视图
切换亮度 Shift+S
翻转板子 Shift+F
单位 Q
网格
飞线
高亮
顶部工具栏
左侧面板
右侧面板
底部面板 S
高亮网络 H
高亮器件
高亮选中
取消高亮网络
取消高亮器件
取消高亮选中
取消全部高亮 Shift+H

Cancel highlighted device

- Top Menu - View - Highlight - Un-Highlight Device
- Right click - Highlight - Cancel highlight device
- Shortcut `Shift`+`H`

highlighted

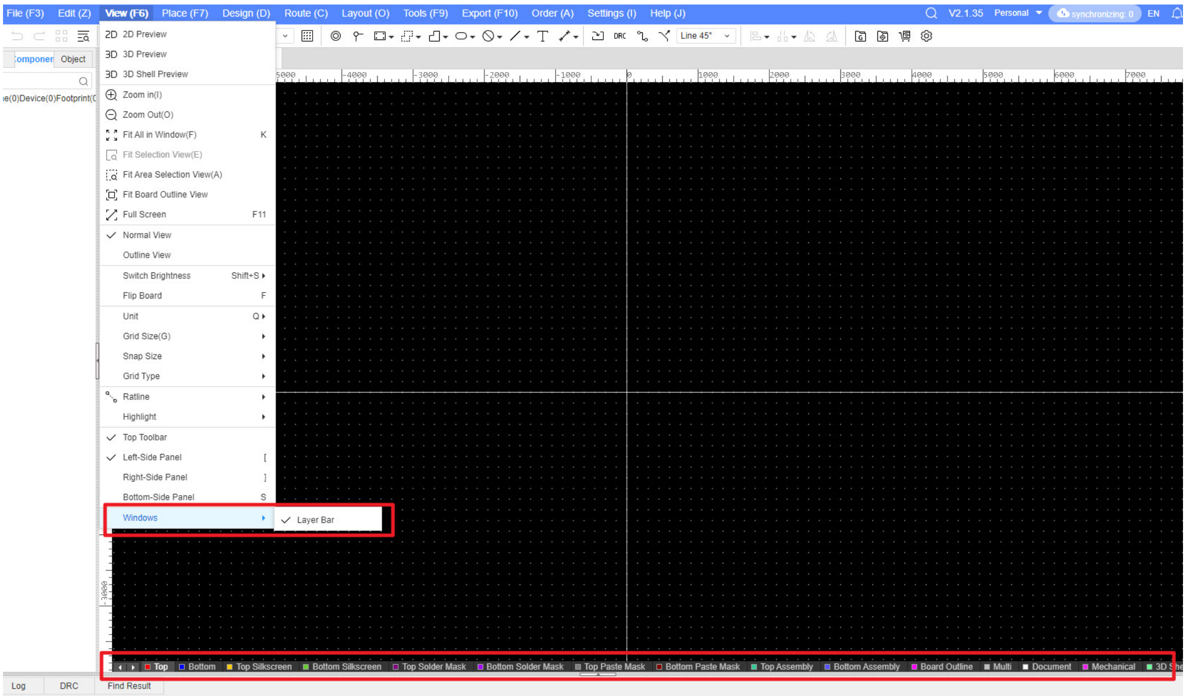
- The operation steps of the highlighted device are the same as the operation steps of the highlighted device, but other elements can be selected by the highlighted selection.

Windows

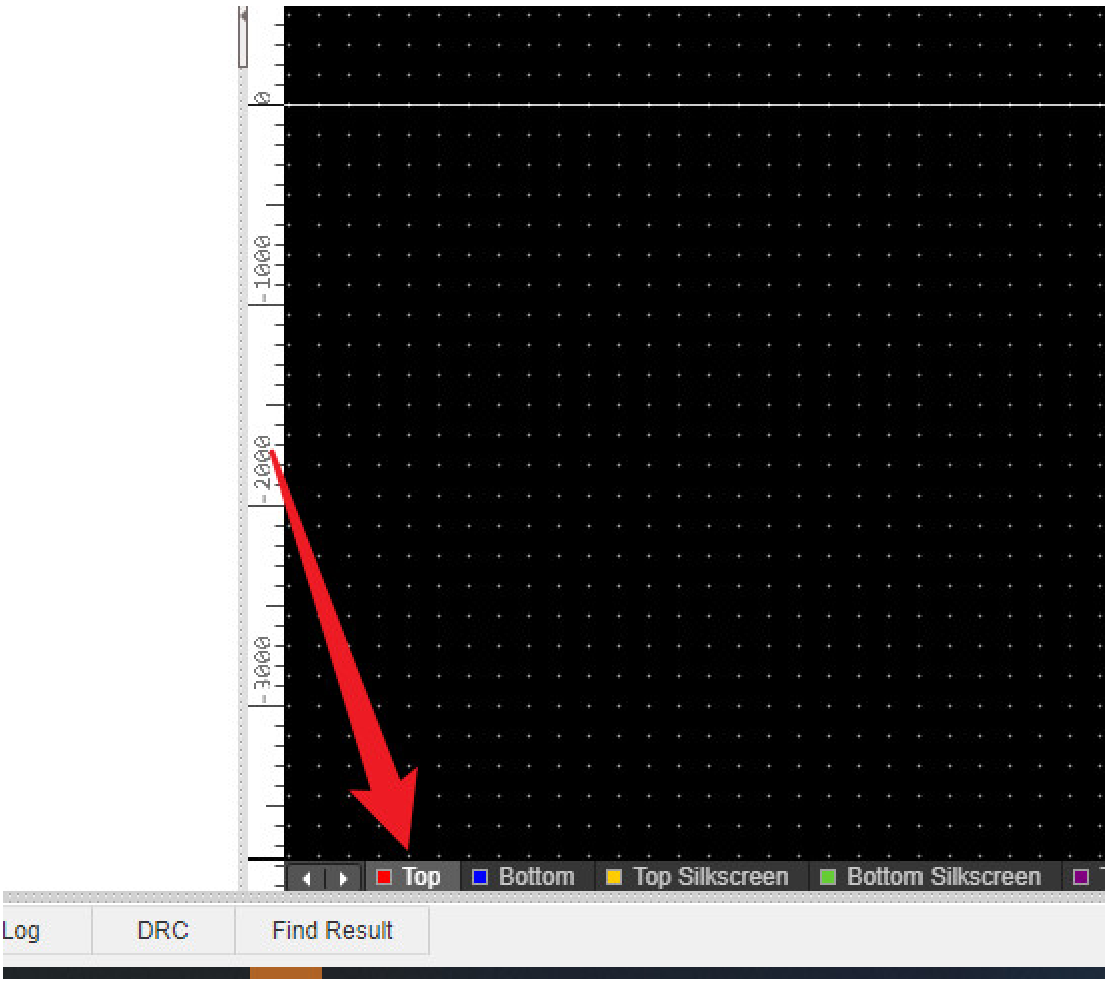
Windows

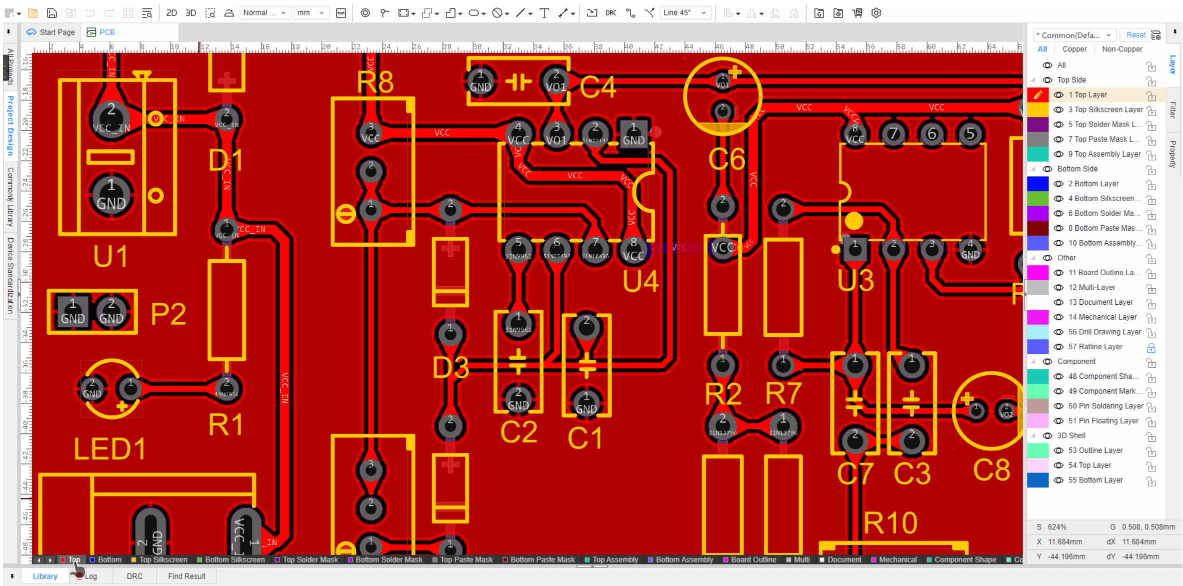
Layer Bar

The layer bar display can be turned on or off in the window under the View menu

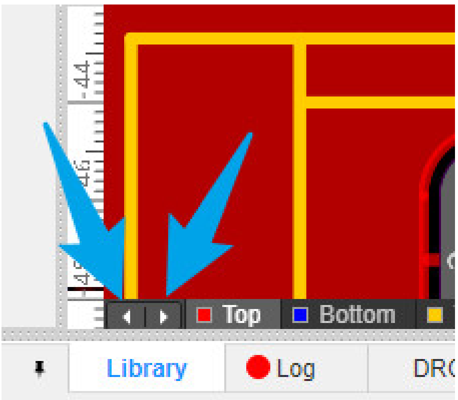


Open the Layers bar.You can switch between currently active layers by clicking the Layers button

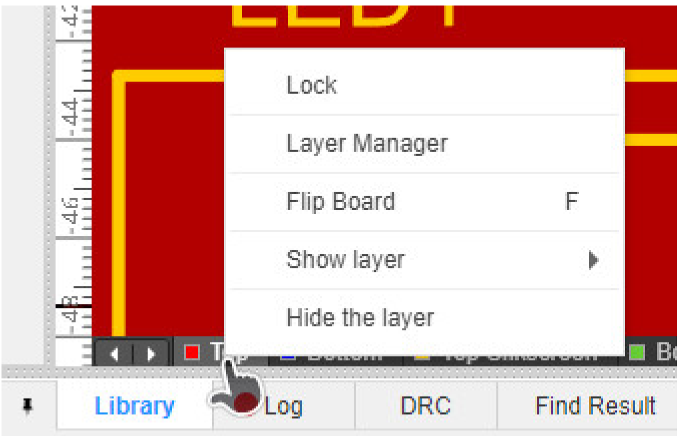




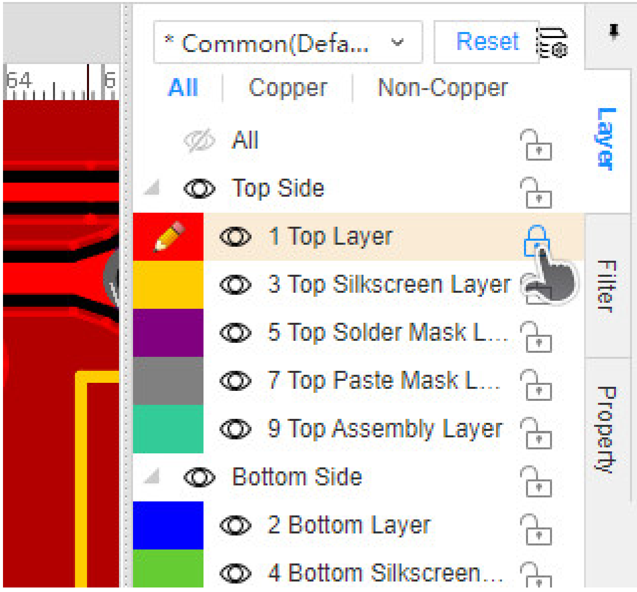
You can also switch by clicking the button on the left



Right-click on a layer button.You can open the right-click menu

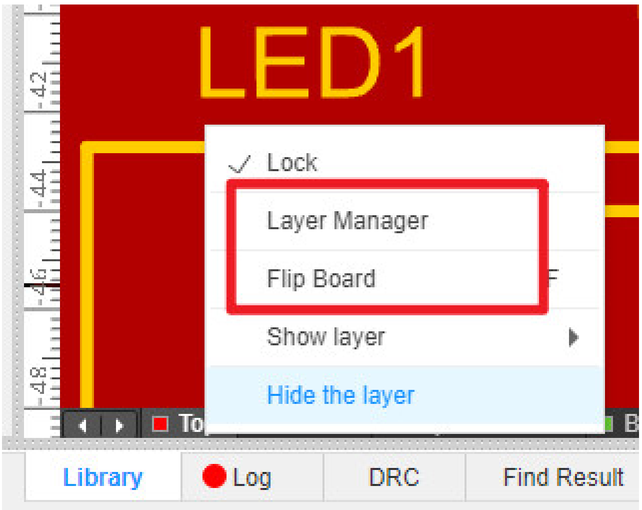


lock： Click Lock to lock this layer， The contents of this layer cannot be selected after locking.The function is the same as the lock function of the original right layer panel

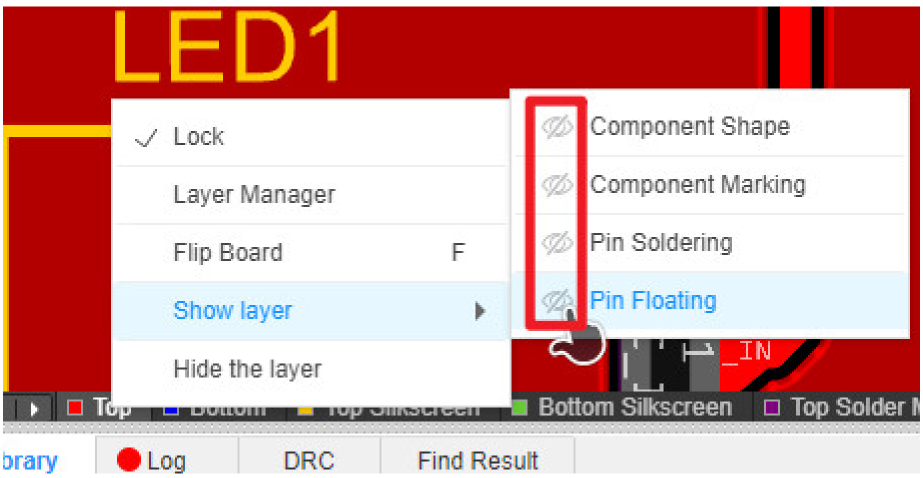


Layer manager： Click on Layer Manager to open the Layer Manager window，

Flip board： Click the flip board to flip the view of the PCB left and right



show layerr： When the mouse moves to the menu of the display layer (this menu will not be displayed when no layer is hidden), the second menu of the hidden layer will appear on the right. Click the eye icon to control the display and hide of the corresponding layer, and the right-click menu will not disappear after clicking. Clicking directly on a layer's name will flip the layer's display state and close the right-click menu

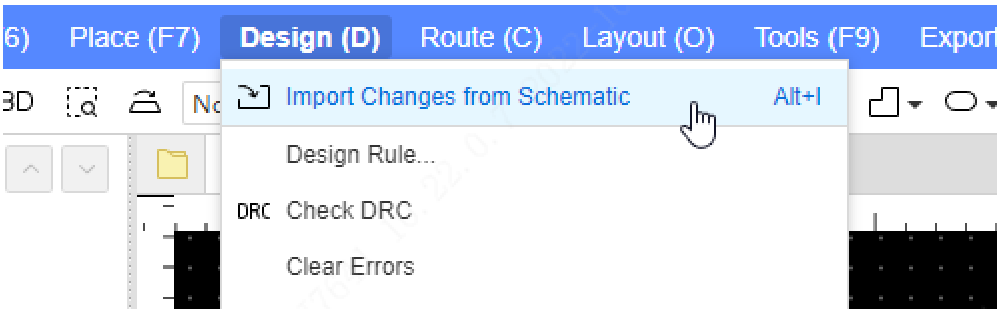


Hide the layer： Click Hide to hide the layer you just right-click on

device

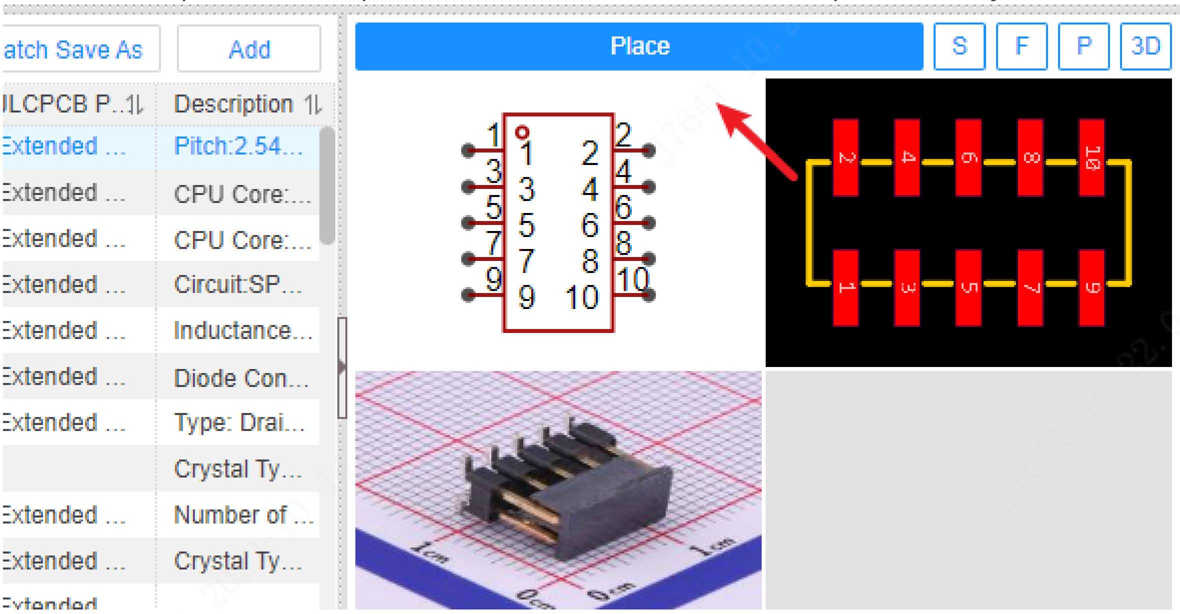
Place the device

1.When the schematic diagram is transferred to the PCB, or when the PCB is imported and updated, the device will be automatically placed in the PCB.



When the schematic diagram is transferred to the PCB, the relative position of the components in the schematic diagram will be placed in the PCB by default, which is convenient for quick component layout search.

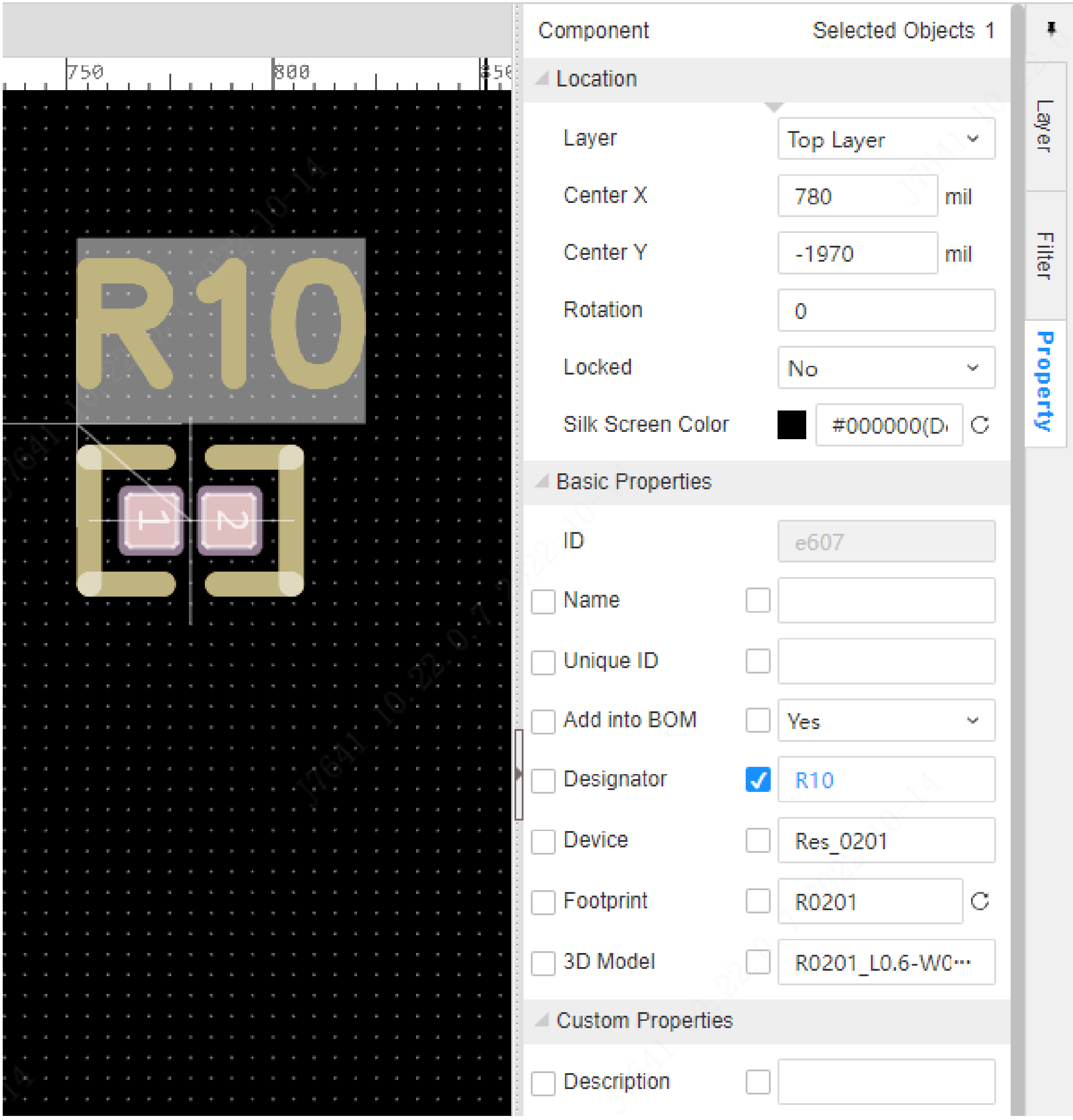
2.You can also place new components on the PCB in the bottom component library.



After selecting the component with the mouse, drag to place the component position.You can also use the automatic layout function of the layout menu, which is currently an experience version and is under development.

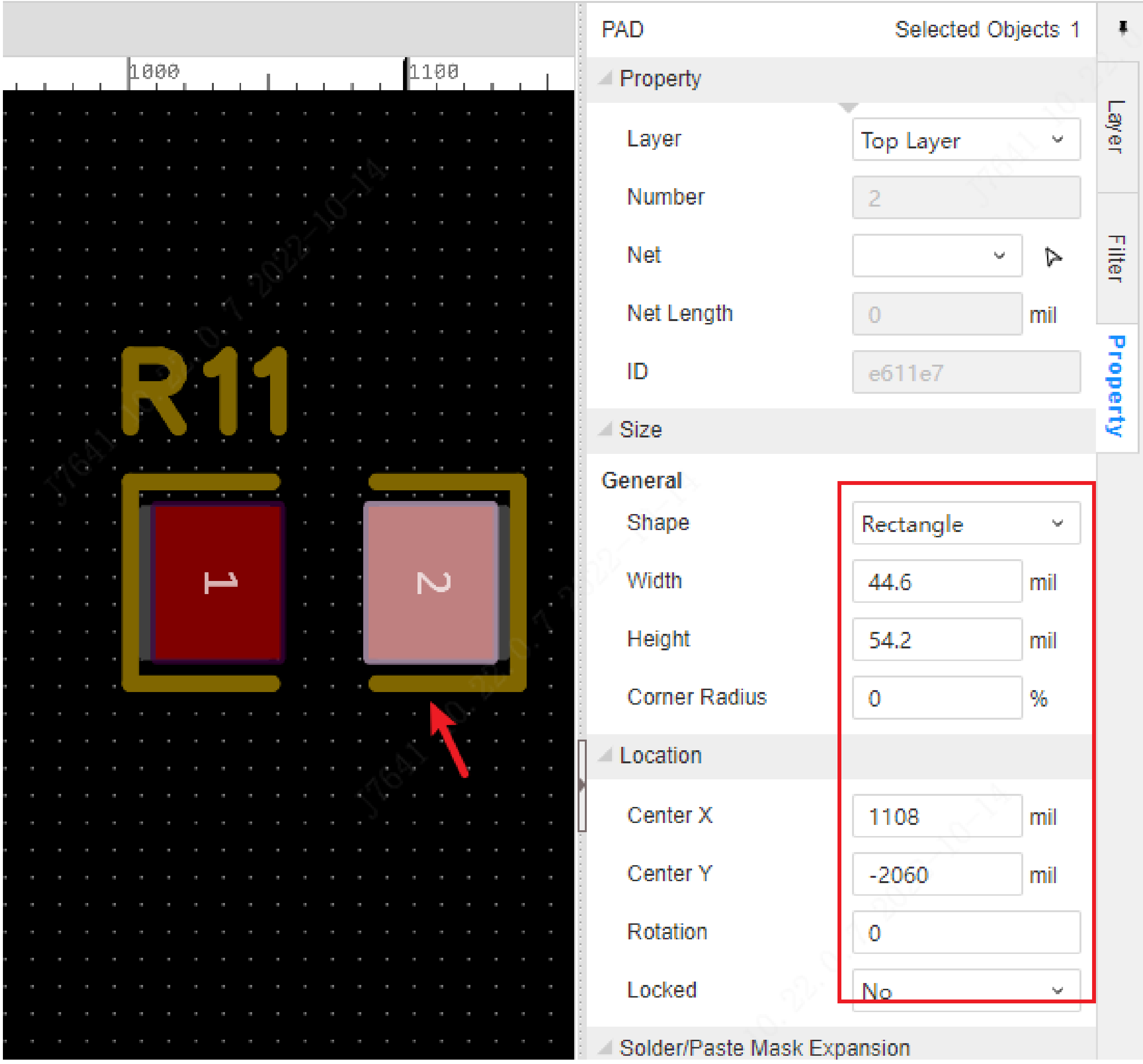
Device Properties

After clicking a component, you can view and modify the properties of the component in the property panel on the right

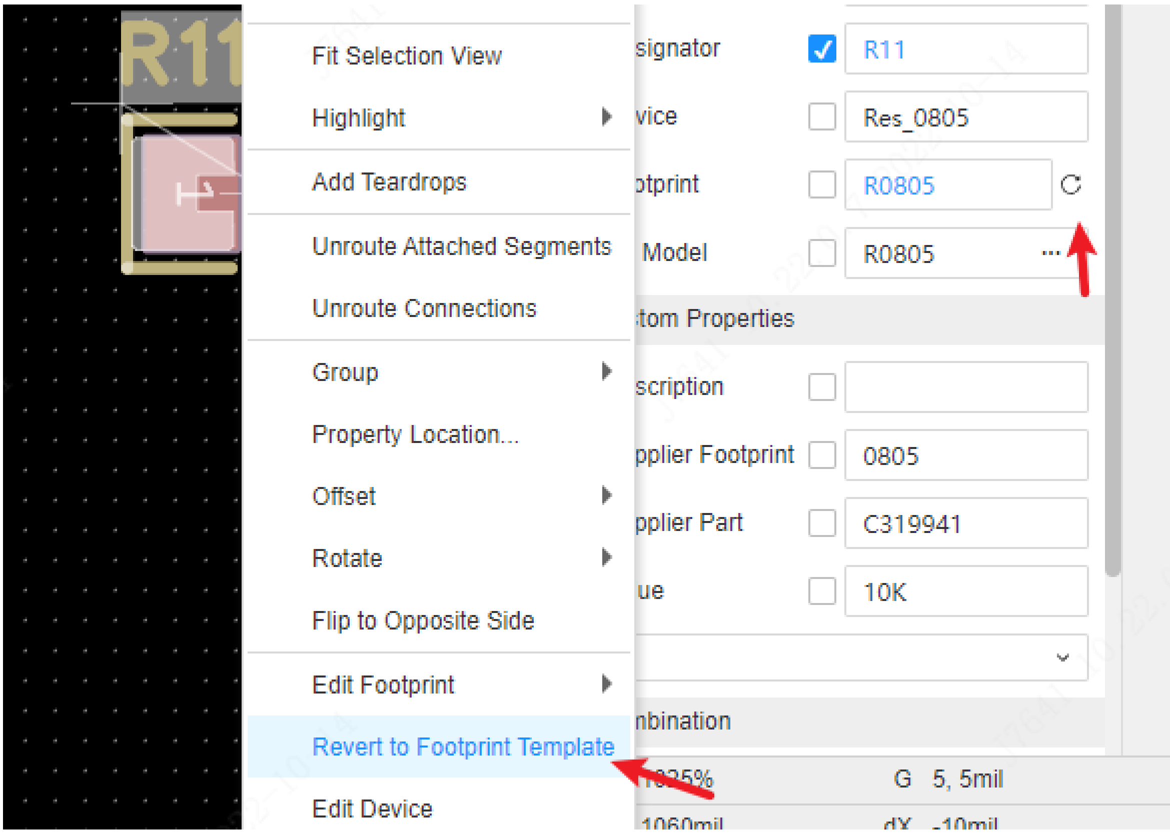


Edit Footprint

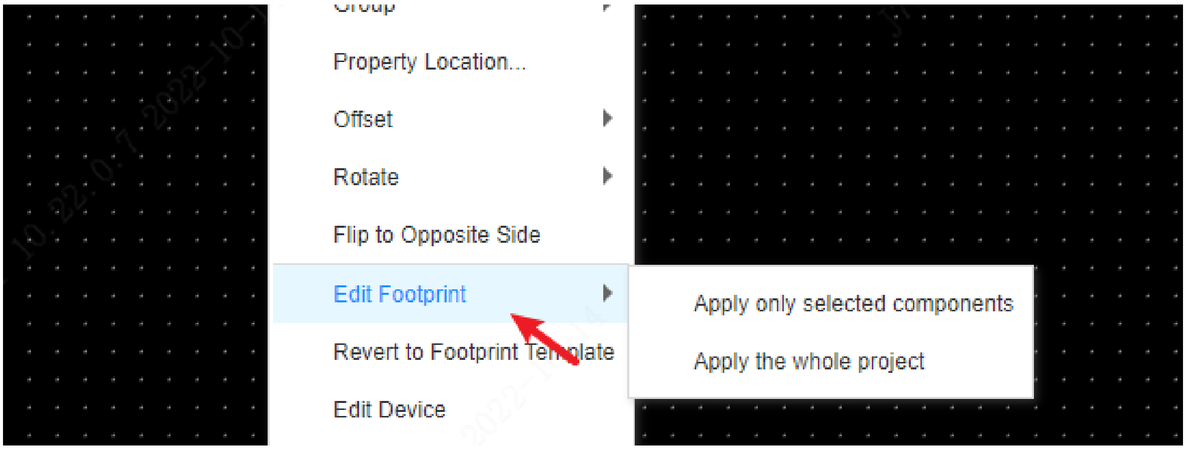
If you need to modify the pads of the package directly in the PCB, you can click the pads in the package to modify the properties (such as modifying the pad size, coordinates), and the modification will not affect other identical packages.When right-clicking to edit a mask, the mask template before modification will be opened.



If you need to restore the original package template, after selecting the component, click the restore button icon of the package attribute in the property panel on the right to restore the modification, right-click menu to revert footprint template.

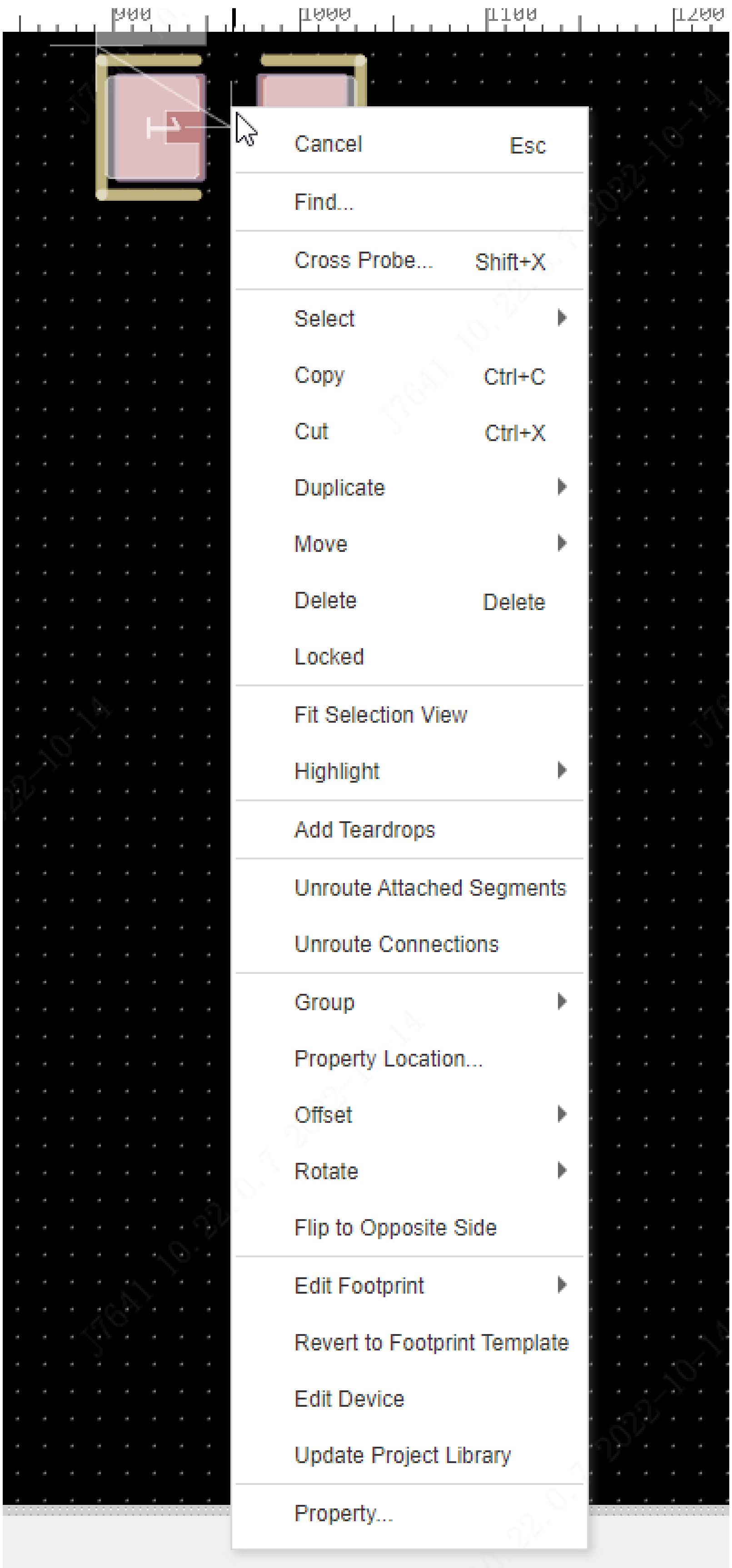


You can also right-click on the package and edit the package.This method can directly modify the project library template and support synchronous modification of the global package.Editing the package will open the package editing interface, which will be automatically synchronized to the PCB after saving.



right click menu

The encapsulated right-click menu supports a lot of functions and can be used according to your own needs.


900
1000
1100
1200
Cancel Esc
Find...
Cross Probe... Shift+X
Select
Copy Ctrl+C
Cut Ctrl+X
Duplicate
Move
Delete Delete
Locked
Fit Selection View
Highlight
Add Teardrops
Unroute Attached Segments
Unroute Connections
Group
Property Location...
Offset
Rotate
Flip to Opposite Side
Edit Footprint
Revert to Footprint Template
Edit Device
Update Project Library
Property...

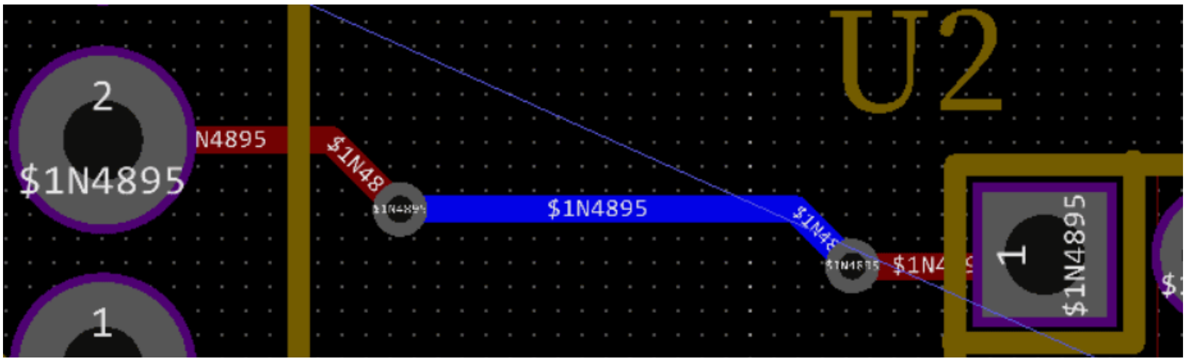
Via

Via

Vias are also called metallization holes. In the double-sided and multi-layer boards, in order to connect the printed wires between the layers, a common hole, that is, a via hole, is drilled at the intersection of the wires that need to be connected on each layer. The parameters of the via hole mainly include the outer diameter of the hole and the size of the drill hole. When you need to draw a double-layer board or a multi-layer board, you can place the via hole to make the top layer and the bottom layer conduct.

- Place two vias on the trace, then you can switch the trace between the two vias to another layer, or remove it.
- During the routing process, use the shortcut key to change layers to automatically add vias. Shortcut `V`.

Note: The default cover oil for system vias



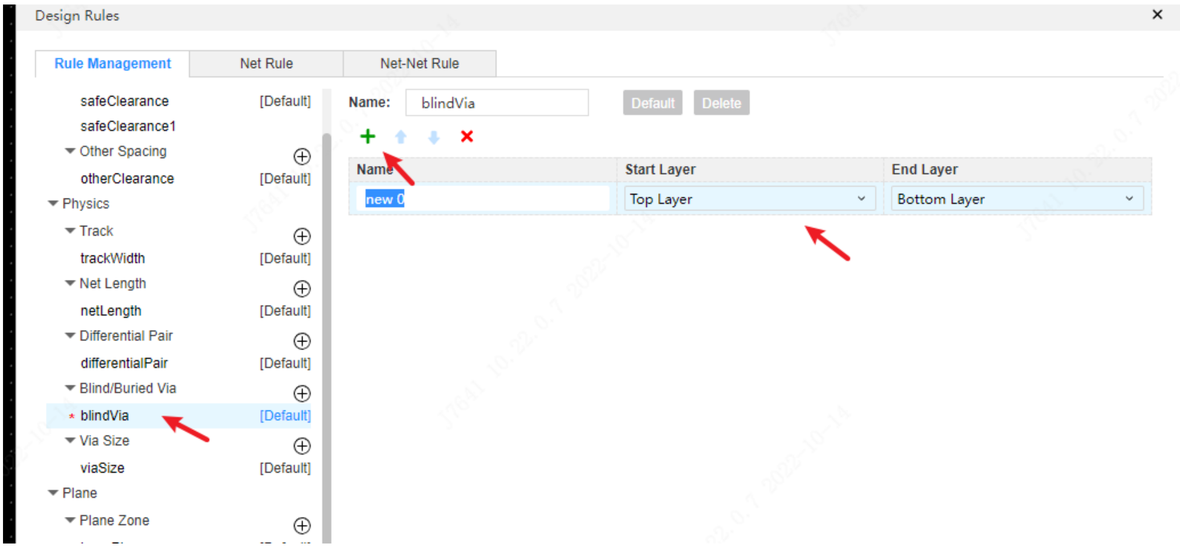
Blind and Buried Vias

EasyEDA Pro already supports Place of blind and buried vias. Before the operation, let me introduce what blind and buried vias are.

Blind Vias: In the design of multi-layer board, one side of the hole is on the surface of the board, and then leads to the inside of the board. Blind vias are vias that connect the surface layer and the inner layer without penetrating the entire page. Blind vias refer to vias that connect between inner layers and are not visible on the surface of the finished board.

Buried vias: Buried vias refer to vias made in the inner layer, which are invisible to the surface and bottom layer, and are used for inner layer signal interconnection. Generally used in mobile phones and PDA boards. Buried vias can reduce the probability of signal interference, maintain the continuity of the characteristic impedance of the transmission line, and save the wiring space, which is suitable for high-density and high-speed circuit board design.

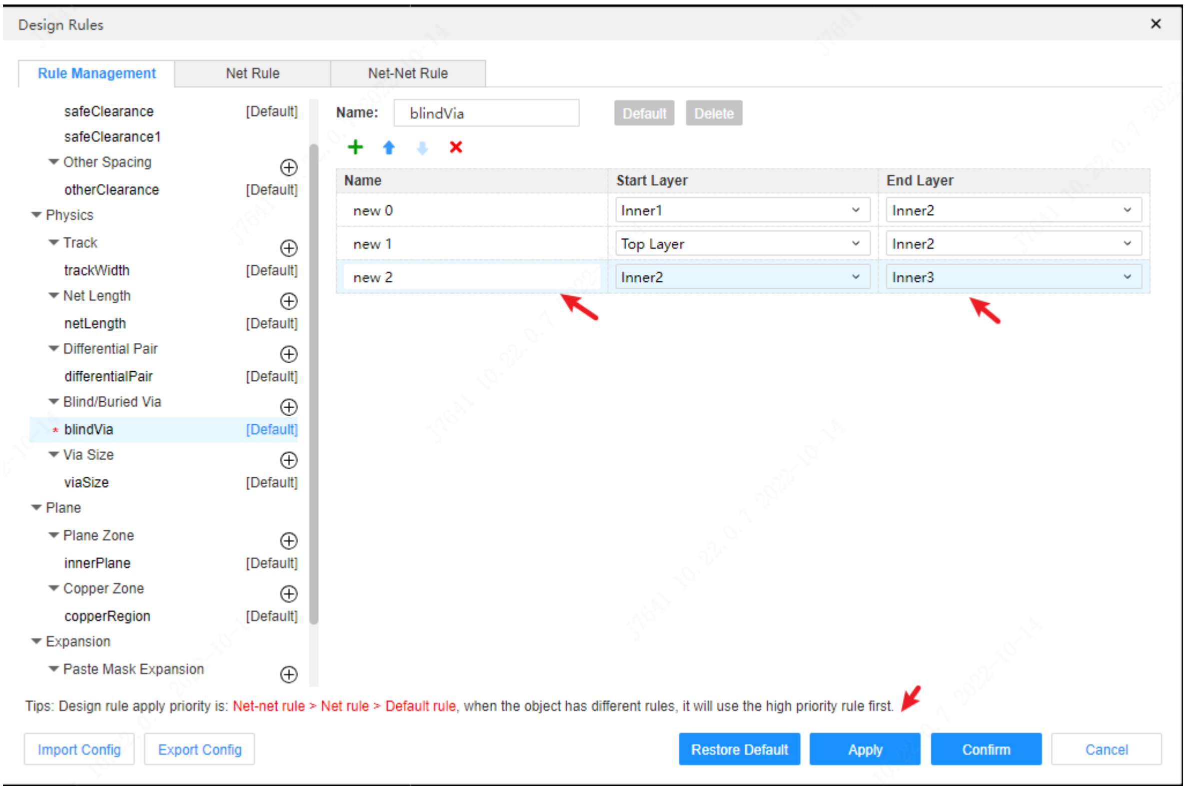
Before placing blind and buried vias, you need to add blind and buried vias in the rule settings. The premise of setting blind and buried vias is that they must be added on two-layer boards or more.



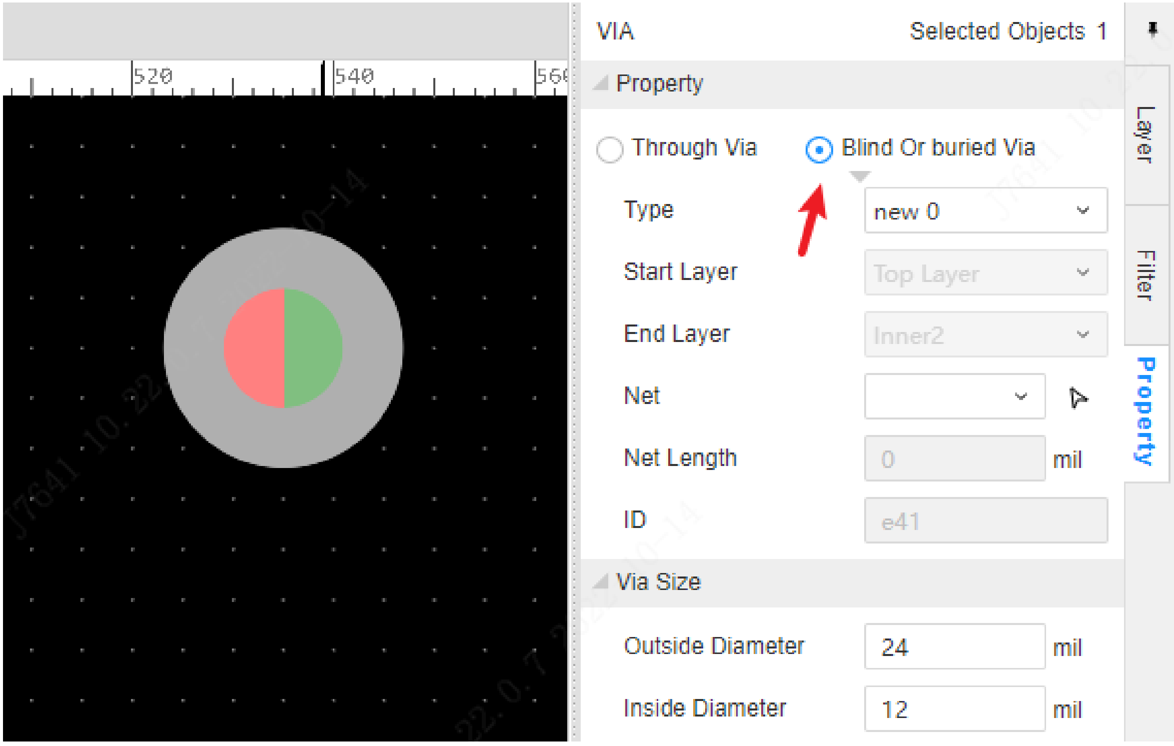
Click "+" to add blind buried via,

After adding, name the blind and buried vias, and select the start and end layers.

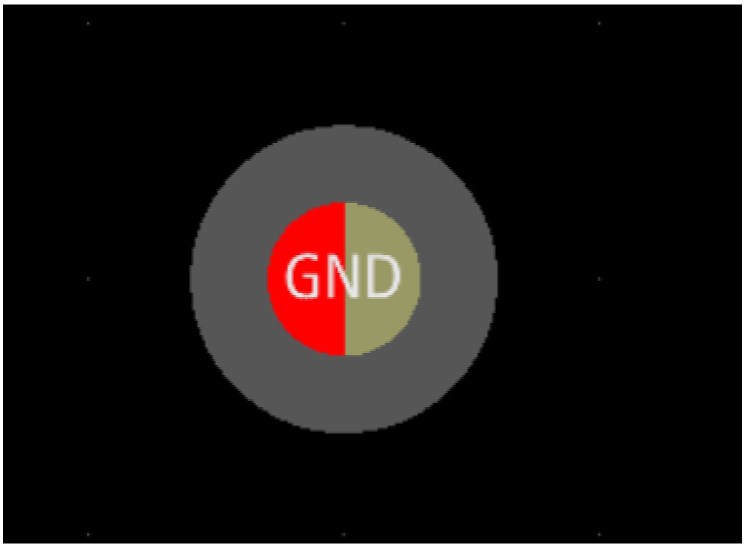
Note: When the start layer is the top layer, the end layer cannot be the bottom layer. **After adding buried vias and buried vias** Click OK to complete the setting of the rules for blind and buried vias.



Before placing the blind-buried via, you need to place a passer, find the properties of the via on the right panel, and select the blind-buried via just added.



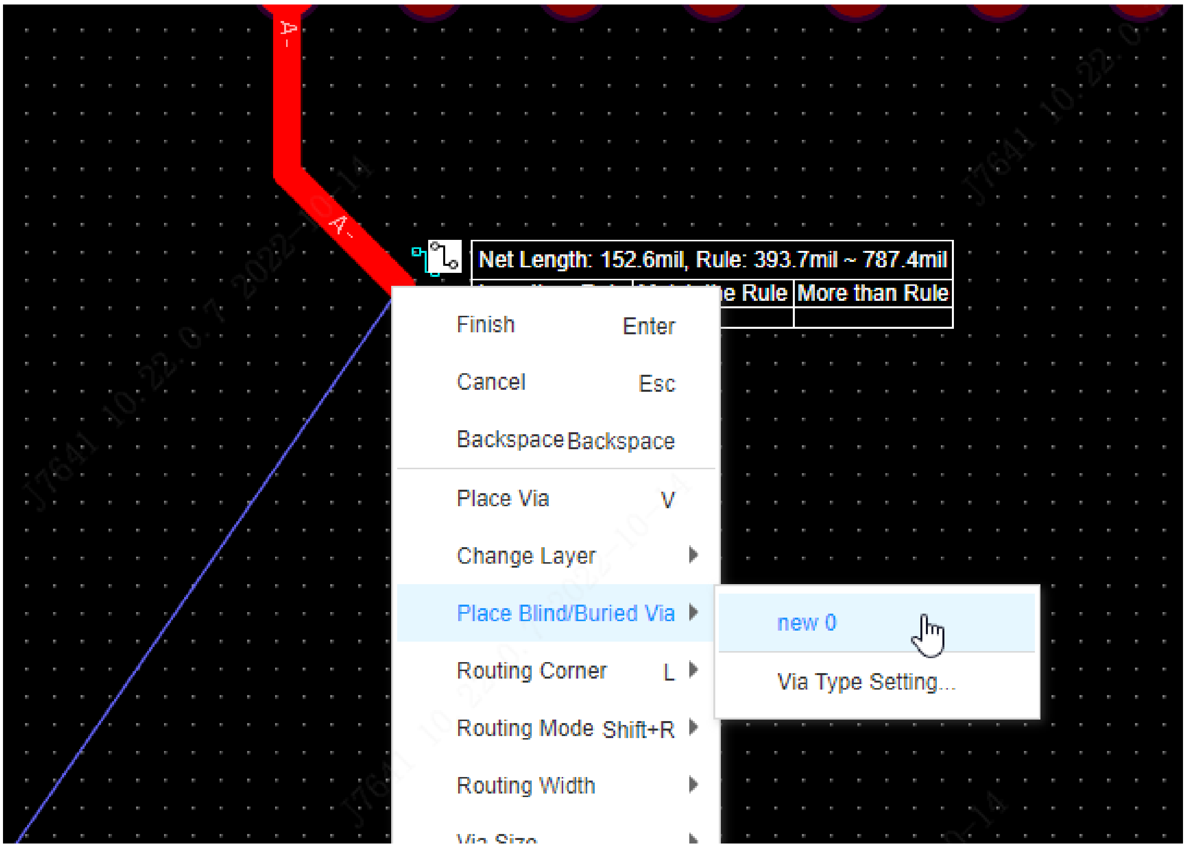
Blind vias: Top layer to inner layer 1



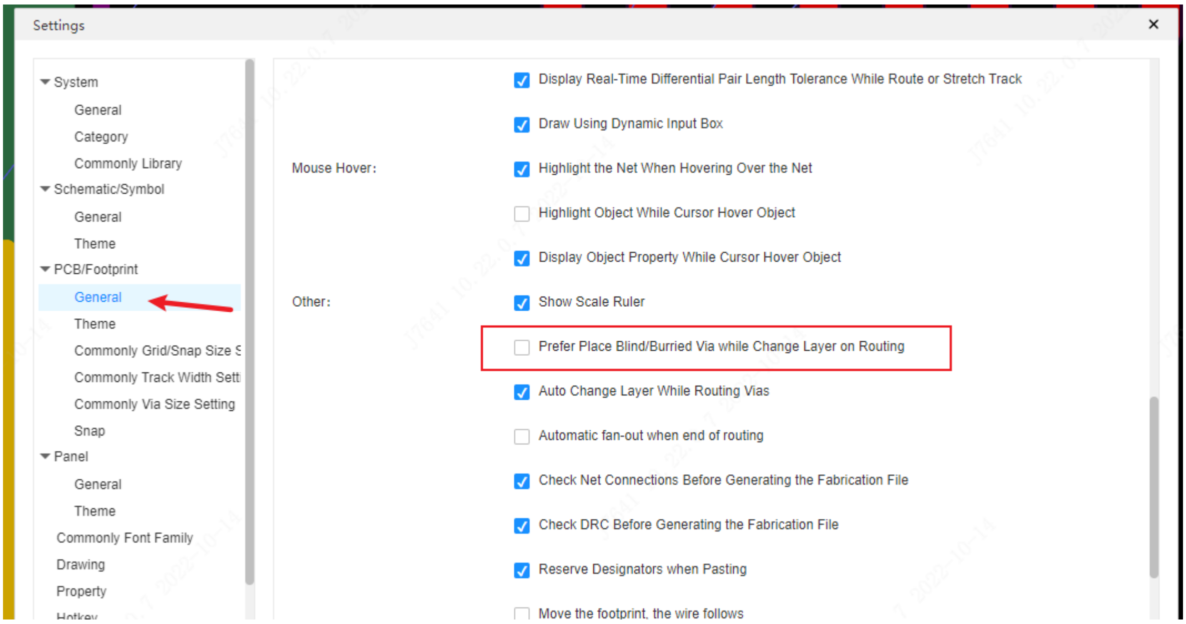
Buried vias: Inner layer 1 to inner layer 2.

Place blind and buried vias in routing mode

Right-click to add blind-buried vias in wiring mode. When placing blind-buried vias, you need to switch to the corresponding layer to place them.



Blind and buried vias can also be placed preferentially when routing layers in the system settings.

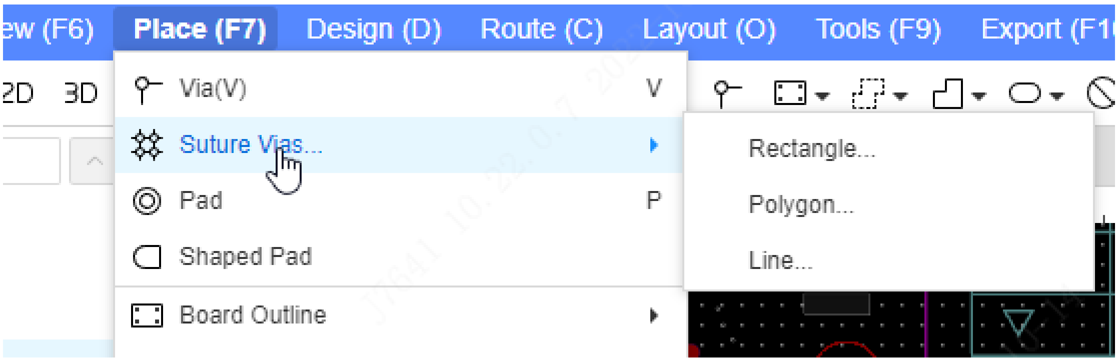


Suture Via

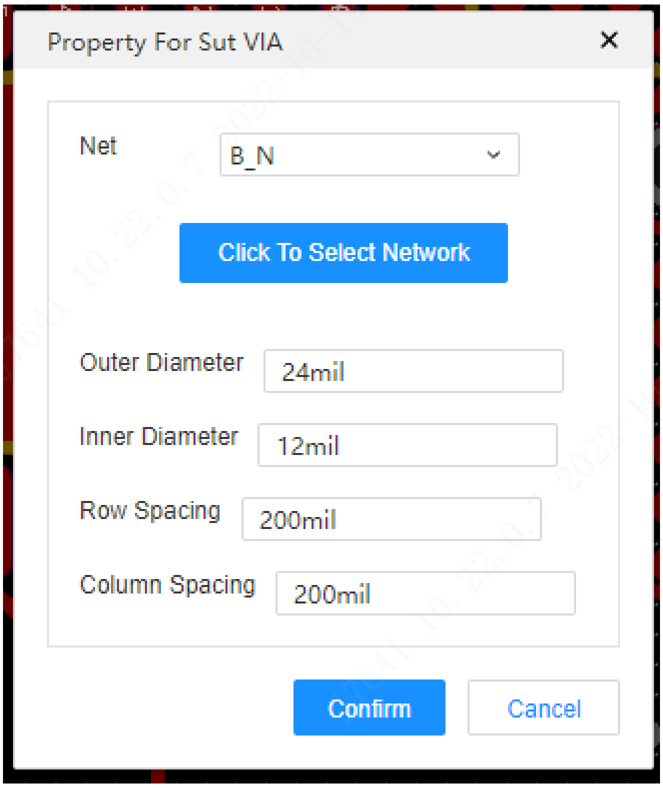
Suture Via technology can connect larger copper foils in different layers together, making vertical connections in the board structure while maintaining low impedance and short return paths. In RF design, suture holes are used with grommets to create a via wall to create an electromagnetically shielded PCB. Stitching holes can also be used to connect copper foils that are separate from the net to connect it to the net.

Place method:

- Top Menu - Place - Suture Via

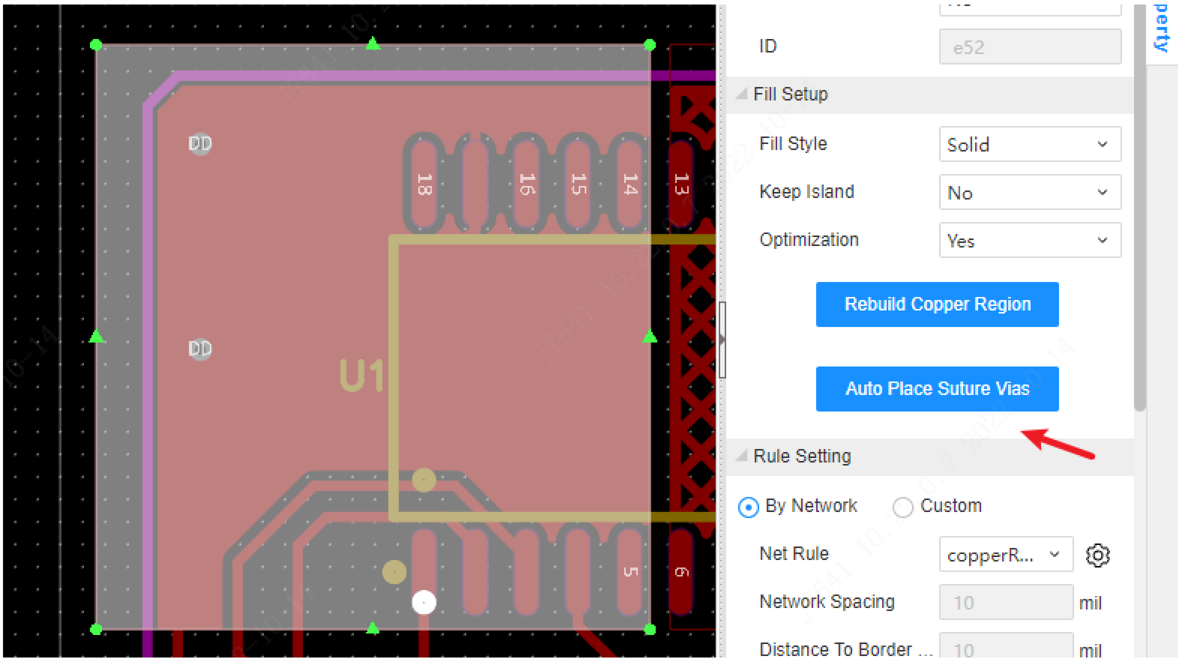


Set the properties of the seam hole, click OK, and then draw a rectangle in the PCB to place the seam hole according to the drawn rectangle.



- **Network**: Set the network for the suture vias.
- **Outer Diameter**: The outer diameter of the suture via.
- **Inner Diameter**: The inner diameter of the suture via.
- **Line Spacing**: Set the line spacing of the suture via.
- **Column Spacing**: Set the column spacing for suture via.

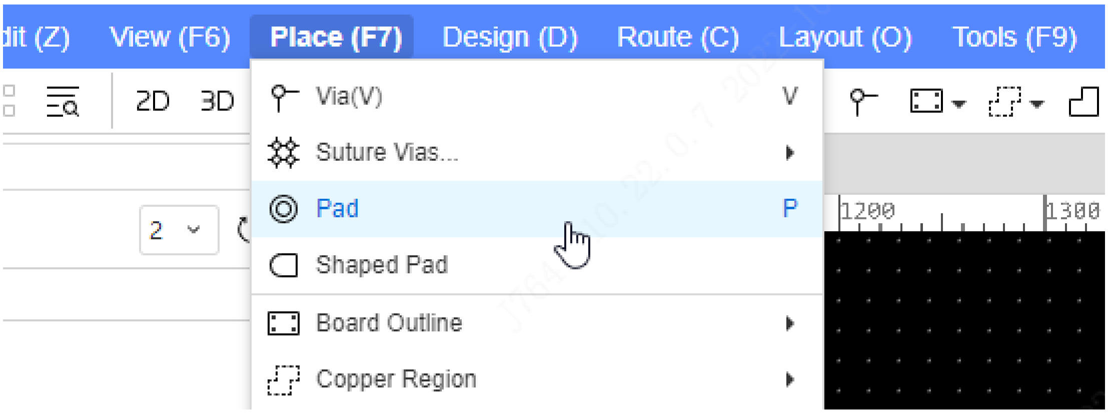
If you select a copper region, you can add the suture via for it



pad

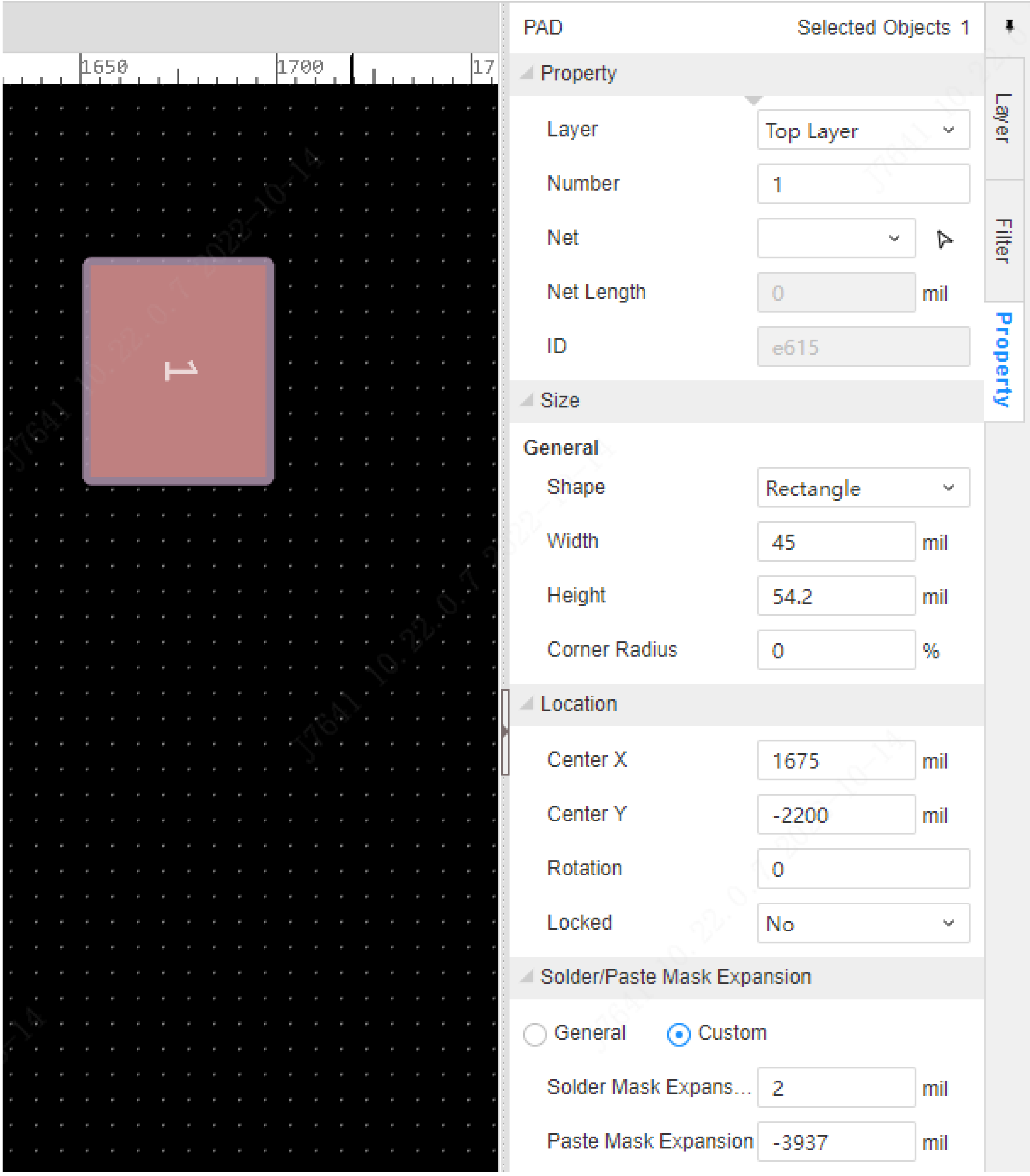
pads

Jialichuang EDA supports the placement of free pads, which is convenient for making simple metallized positioning holes. Shortcut key P

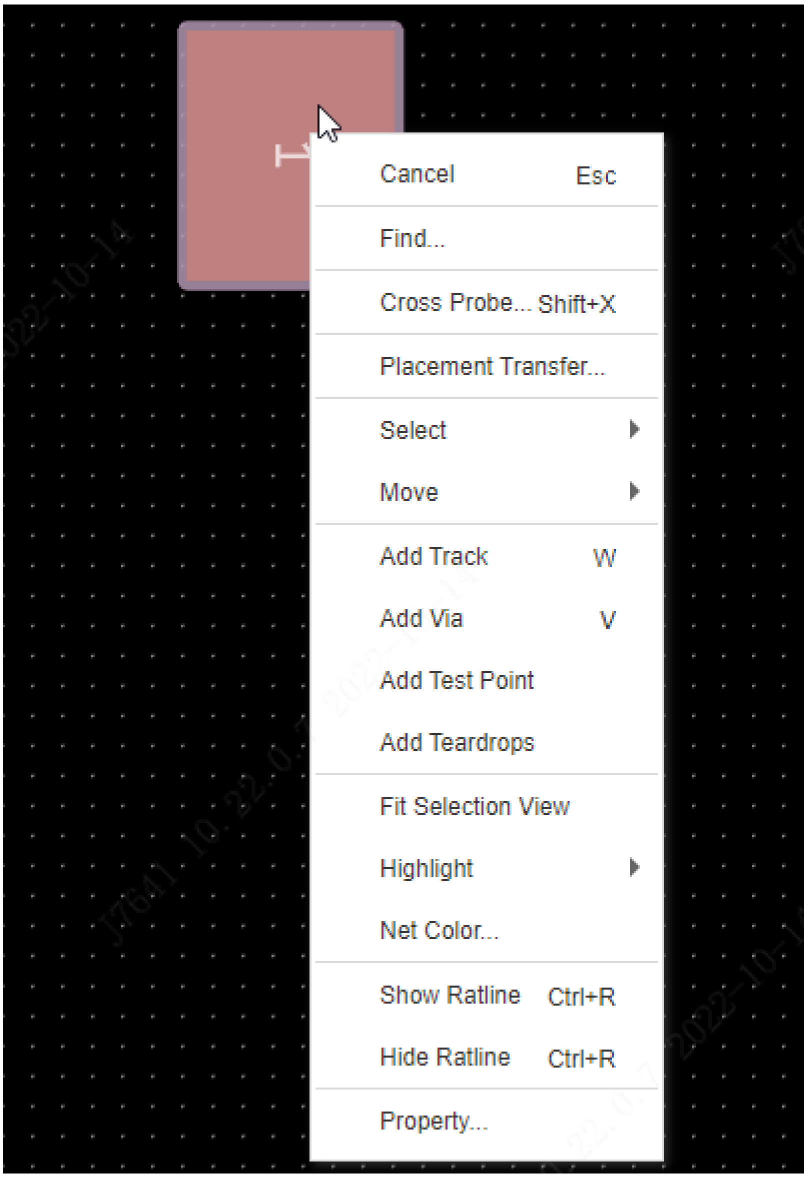


Non-metallized pilot holes can be achieved by placing circular pocket areas.

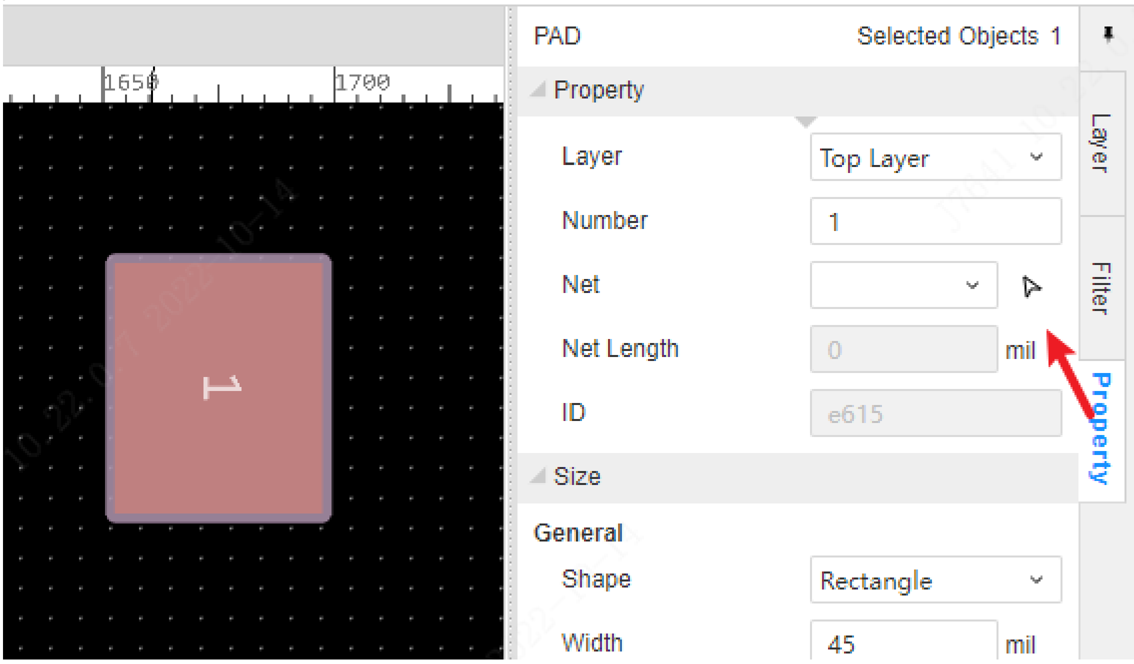
You can modify the layer and shape of the pad by setting the properties of the pad, and you can also modify the shape of each layer, etc.



The right-click of the free pad and the pad of the package supports a variety of functions, which is convenient to use.



In the property panel, you can quickly assign nets to pads by picking primitives with the mouse.

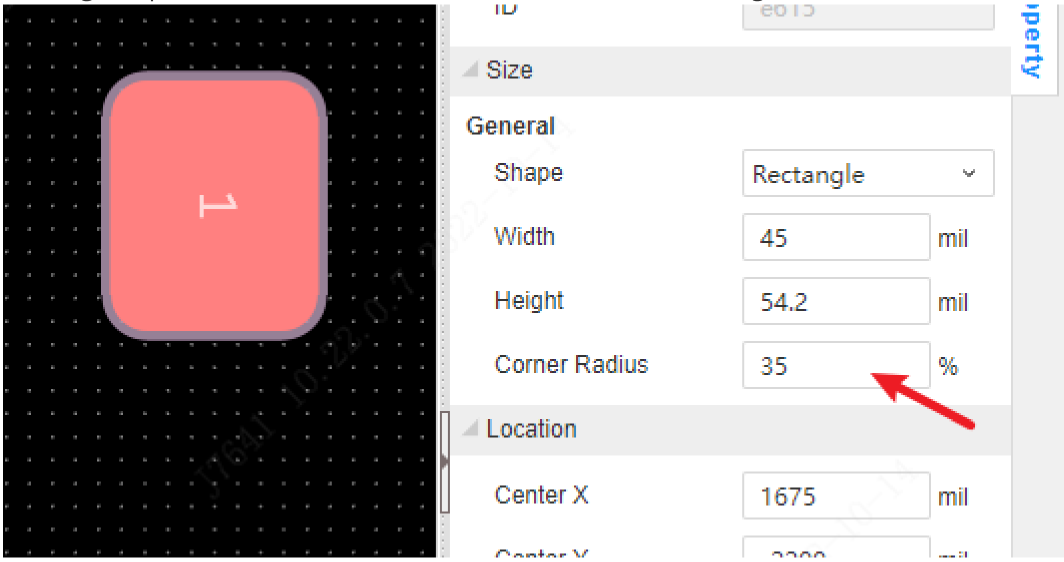


When pasting or placing pads directly onto primitives on a network，Is automatically assigned to the network

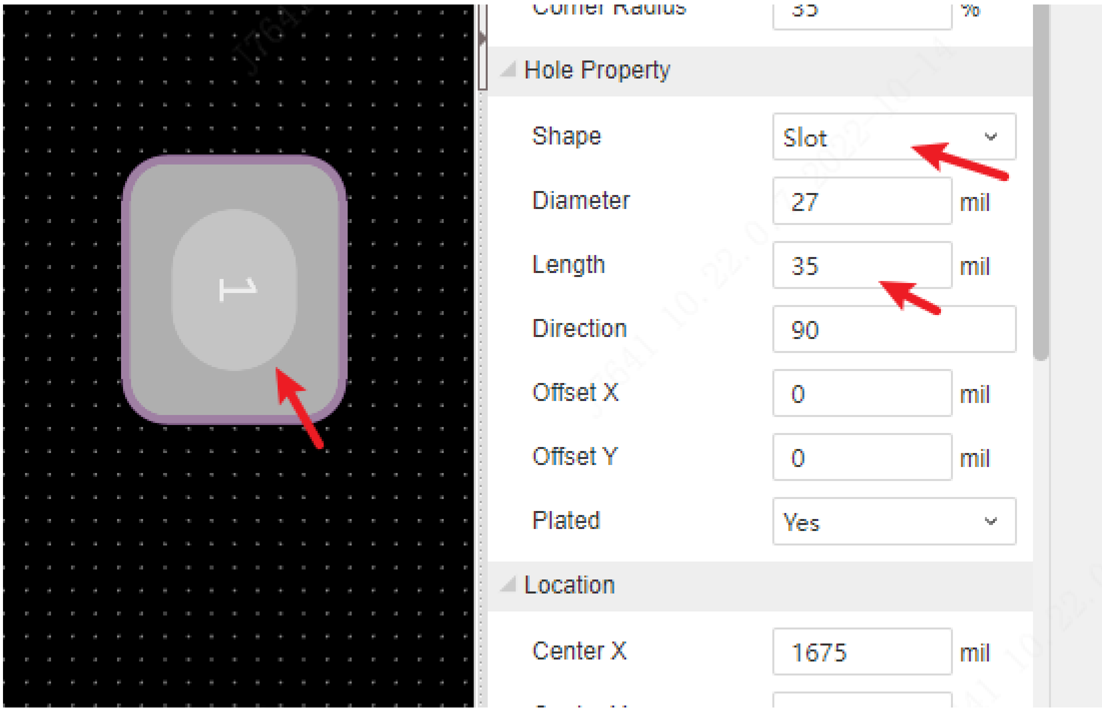
Pad shape

Pad shapes support circle, rectangle, oval, and polygon. Polygons need to be generated by drawing shaped pads.

When using rectangular pads, it is supported to set the corner radius to realize rounded rectangular pads. Set the corner radius ratio with the short edge as the 100% base.

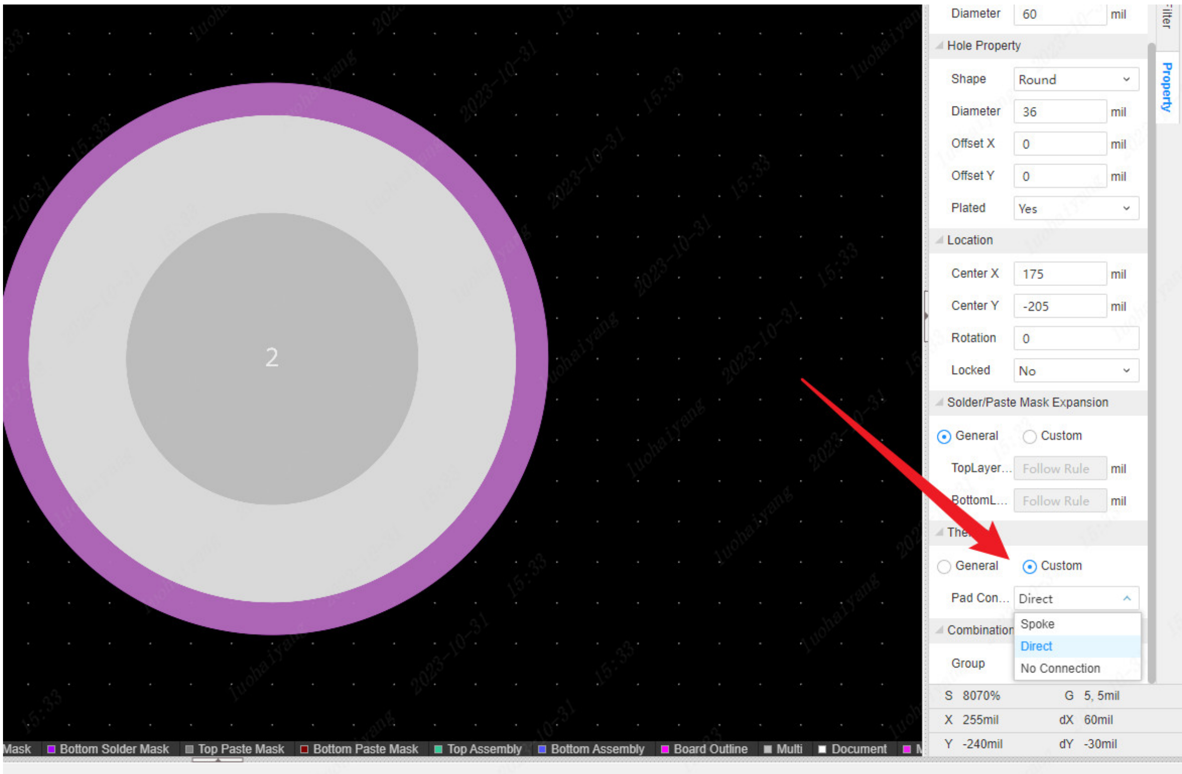


When different pad shapes are on different layers, different properties can be set, such as slotted hole pads for multi-layer pads.



Thermals

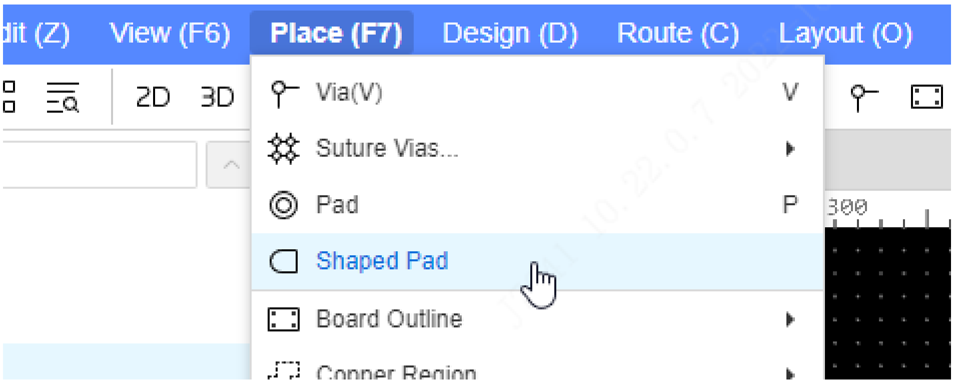
You can directly adjust the Thermals of this pad. When selected as universal, this pad follows the rules to generate Thermals. When selected as custom, you can directly change the connection mode of the pad here to divergent, direct or no connection



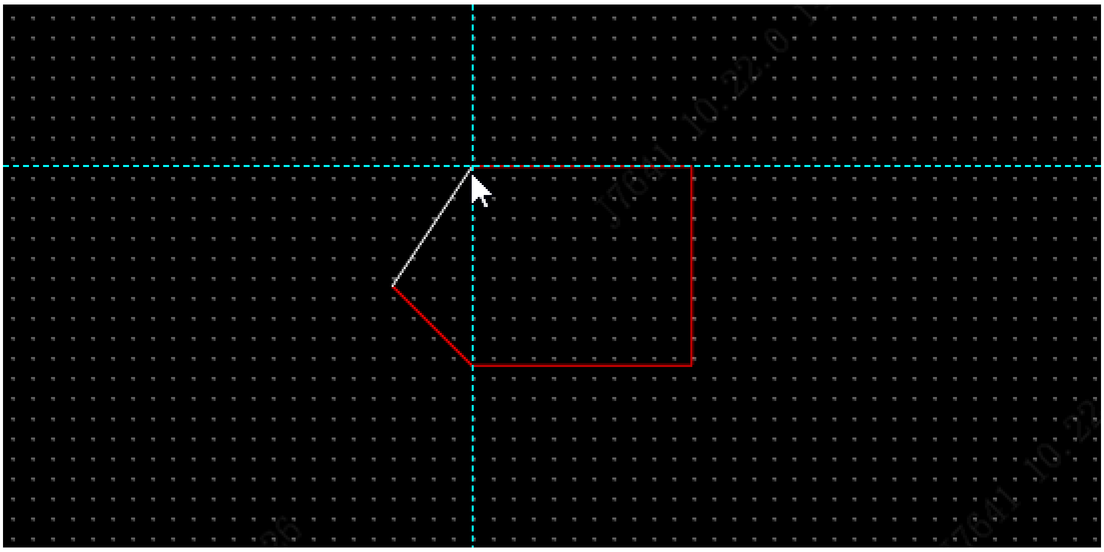
Shaped Pad

Shaped pads

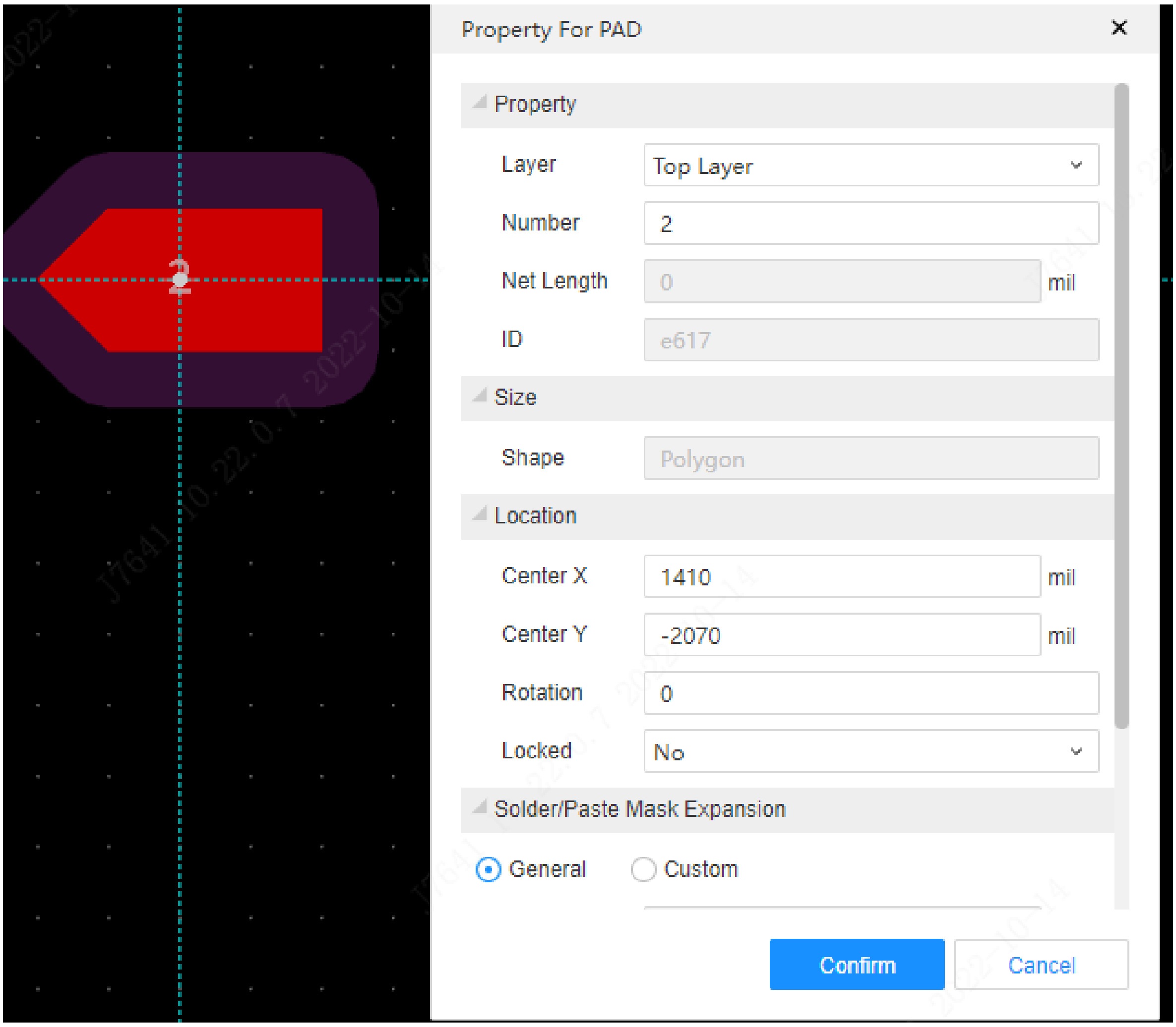
Special-shaped pads, also known as polygonal pads, many irregular-shaped pads can be drawn through this function. For example, some antennas can also be drawn with special-shaped pads.



Click the menu to enter the drawing mode. Similar to drawing the filled area, by modifying the corners of the wiring (shortcut key L), you can switch between different corners and draw different shapes.



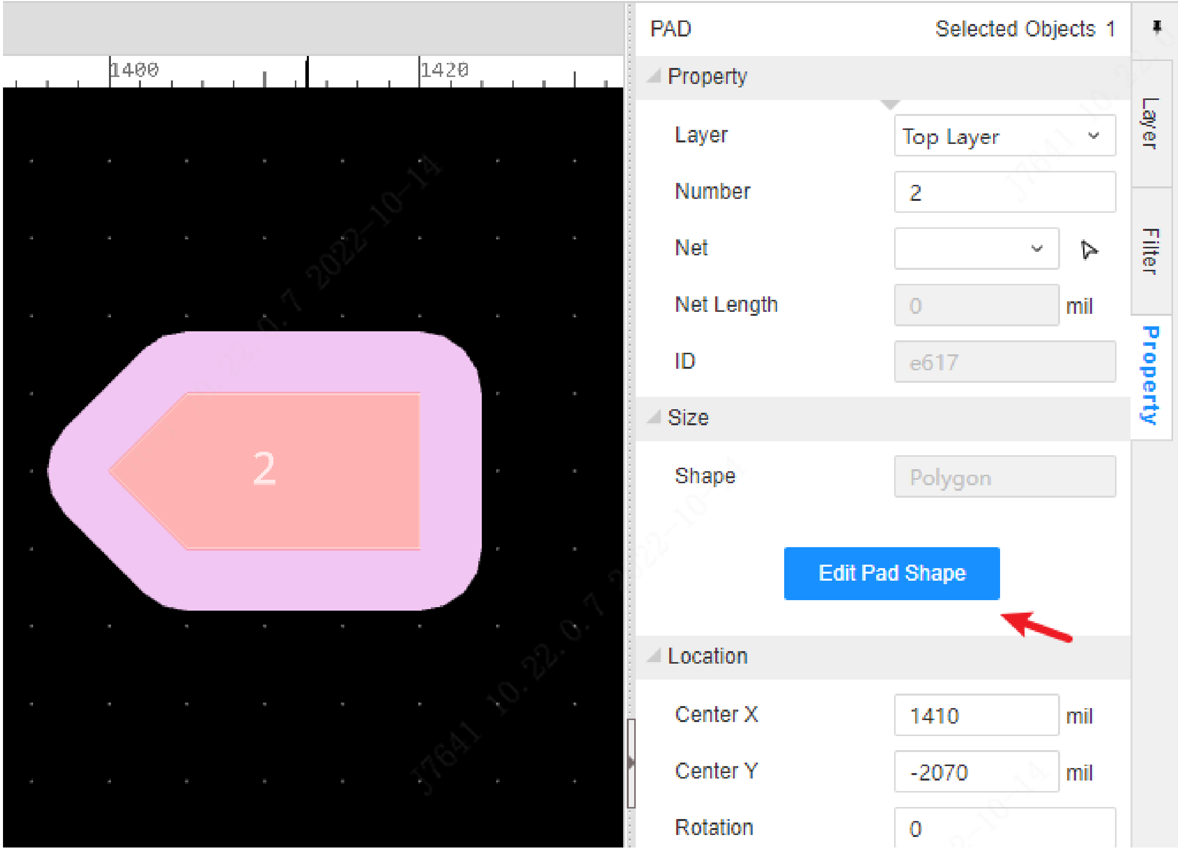
Right-click to complete the drawing and set the property parameters of the pad



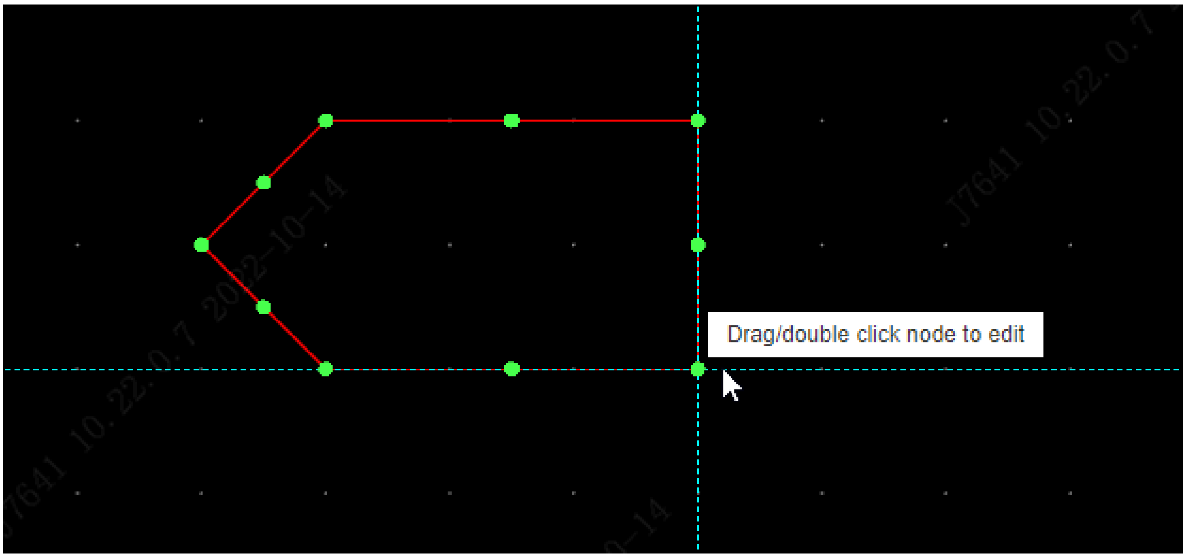
OK to finish drawing.

Edit Shape

Click on the shaped pad, there is a button "Edit pad shape" in the property panel on the right.



After clicking, the cursor enters the edit mode, and you can drag the node or double-click the node to edit the pad shape.



Board Outline

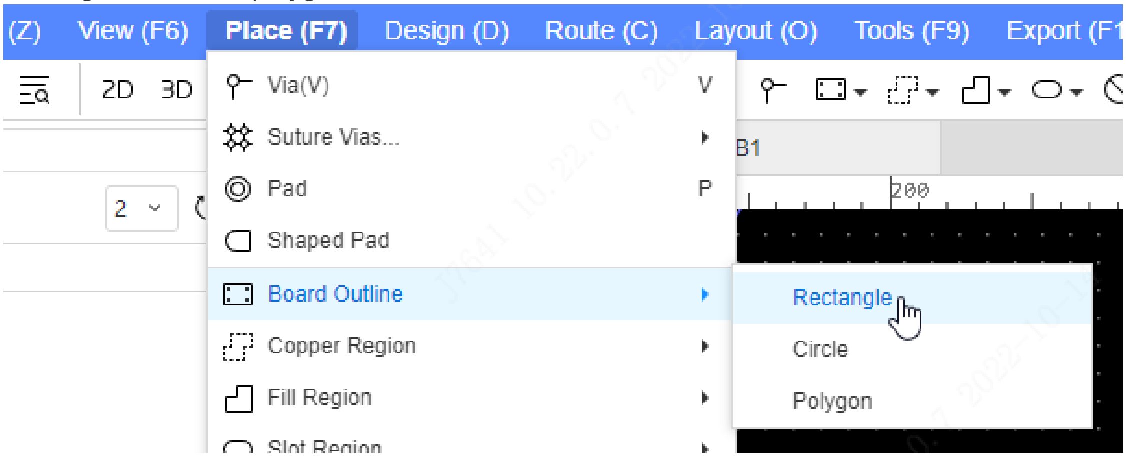
Before starting the PCB design, you first need to create a frame for the board. The board frame can be created by direct drawing and by importing DXF.

illustrate:

- Only one plate frame can be placed in a plate frame, more than one plate frame will be turned into a grooved area

draw directly

- Top Menu Place - Board Frame
- The top toolbar board frame enters the drawing mode, providing three ways to draw rectangle, circle and polygon



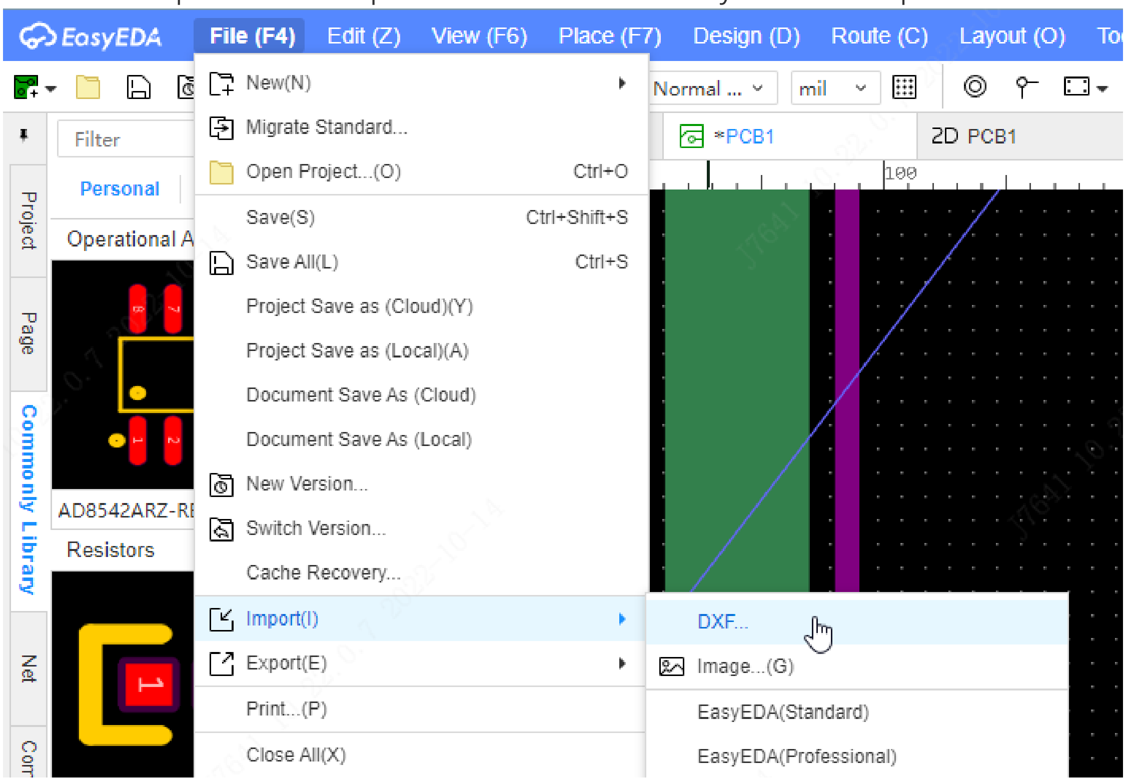
The drawing methods of the board frame, copper area, filling area, slotting area and forbidden area are exactly the same.

place board frame

Import DXF

You can switch to a board frame through any type of contour object. If the line is not closed, it will prompt to automatically close the contour. This function is mostly used to convert the imported lines to the desired type (plate frame, cutout area, forbidden area, etc.) after importing DXF.

- Click on the top menu File - Import - DXF Select the DXF file you want to import



the more import dxf information please refer at Import DXF section.

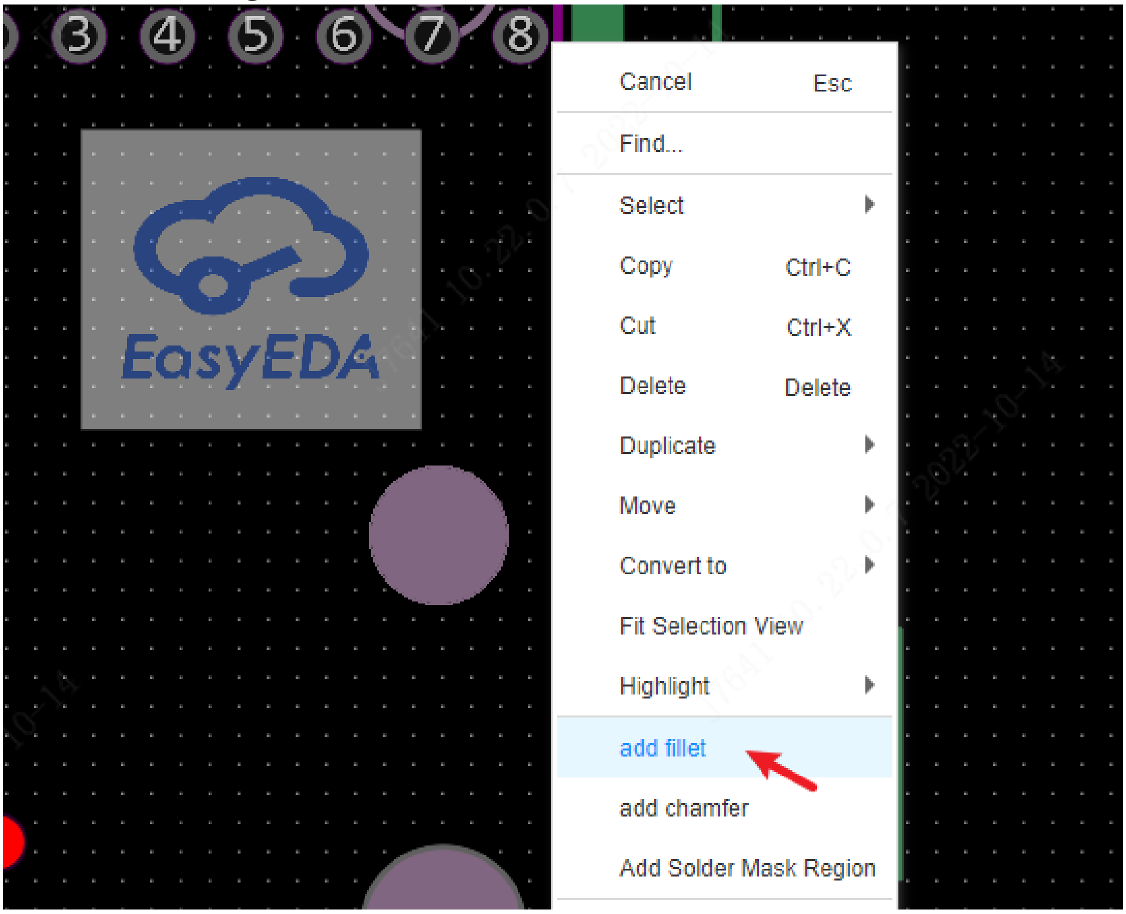
Notice:

- The plate frame and grooving area must be closed. If the line you select is not closed, you need to select multiple lines and merge them into a whole through the right-click menu **Merge as closed object**, and then convert the plate frame
- The non-closed lines of the board frame layer will also be exported as the board frame layer, which does not affect you to draw the board frame with lines.

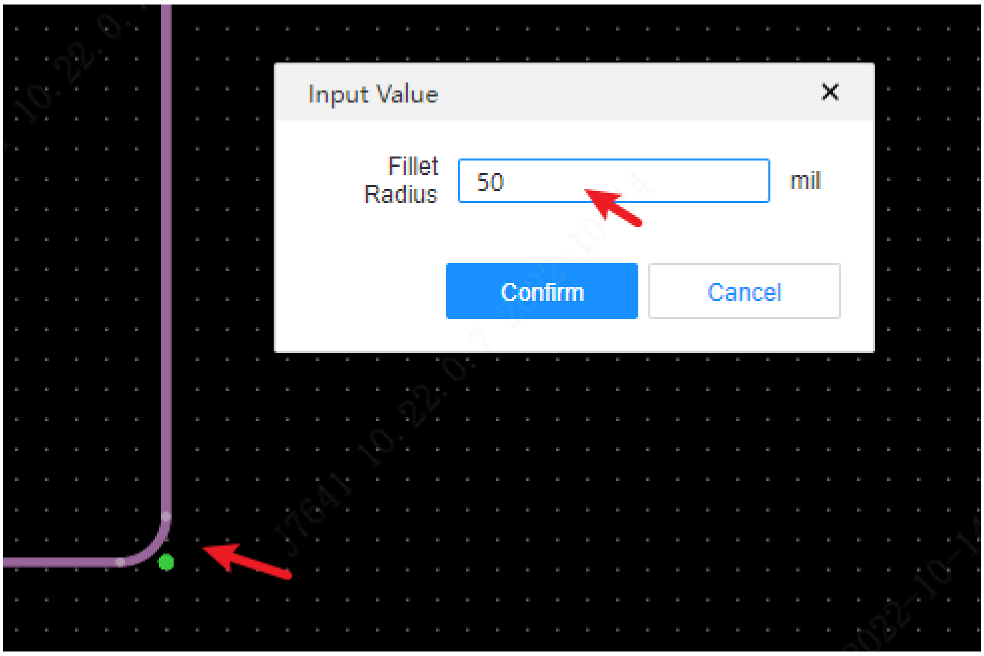
add rounded corners

The method of drawing a rounded rectangle border in the professional version is different from that in the standard version. It is split into another function. The specific operation steps are as follows:

- Select the board frame, the board frame needs to be a complete closed loop to generate rounded corners - right mouse button - add rounded corners

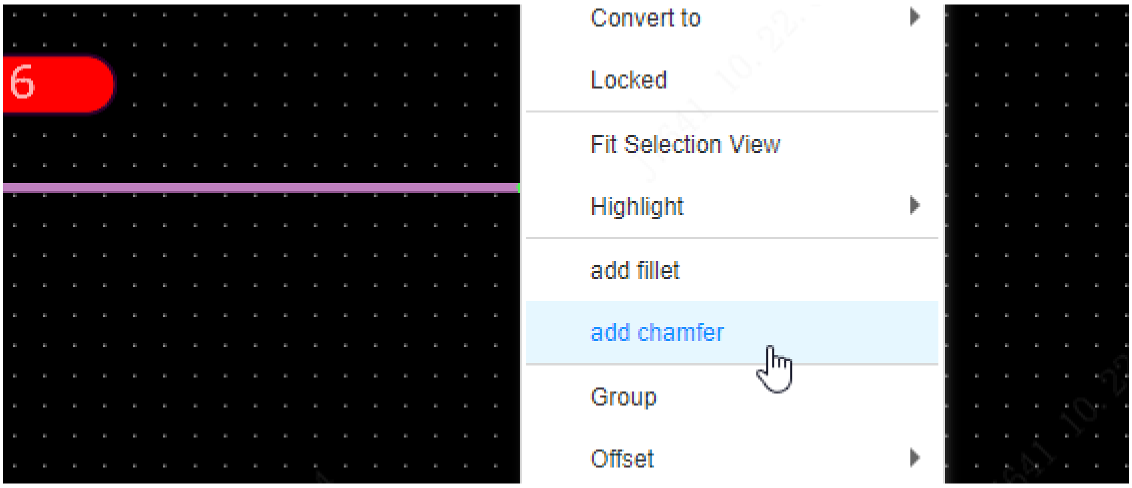


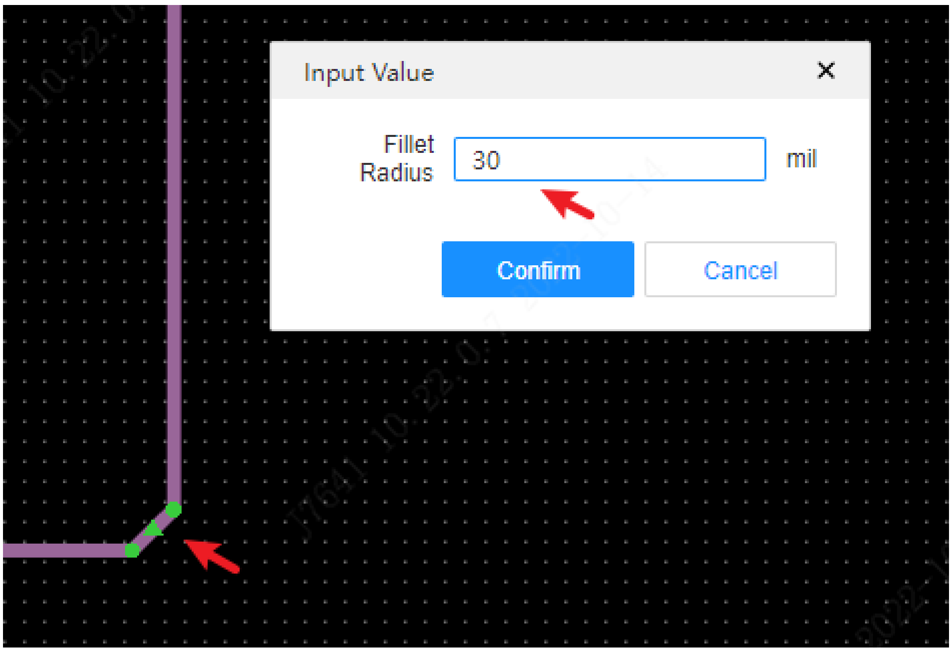
- Enter the width of the rounded corners in the pop-up window - click OK to generate rounded corners for the board frame



Add Chamfer

The operation steps are the same as adding rounded corners, and will not be explained here.



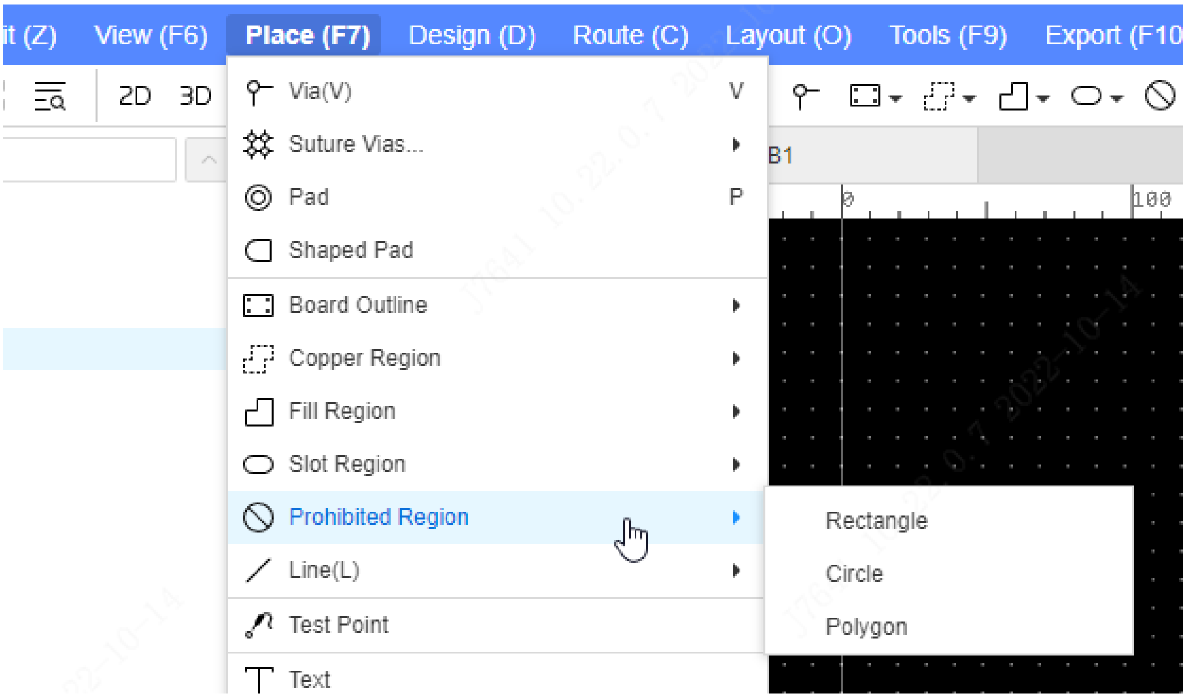


Prohibited Region

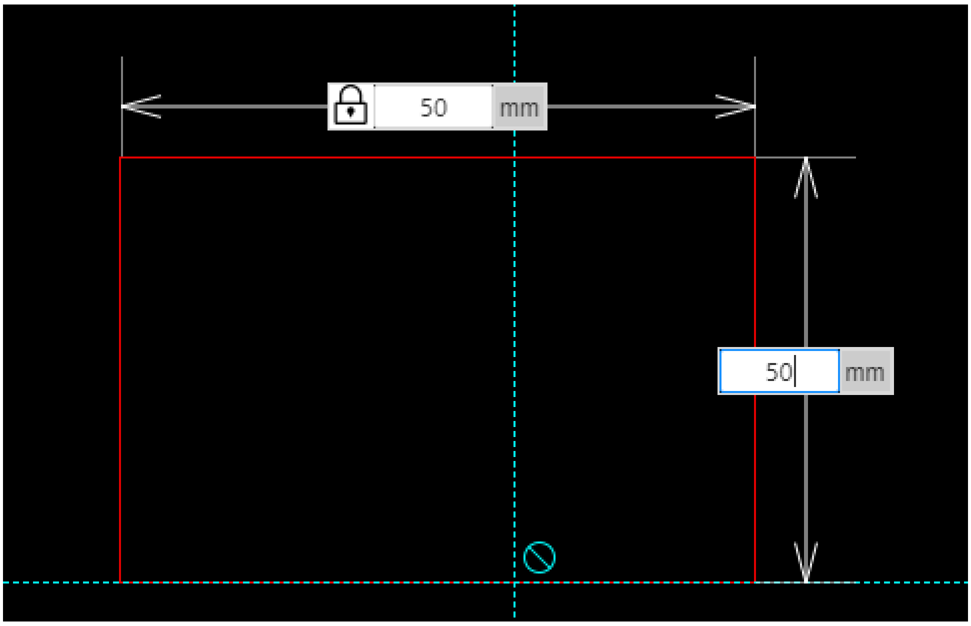
In designing a PCB, some circuits are sensitive to signals, and the signals are easily interfered. Usually, a prohibited region is set up. The prohibited region includes forbidden wiring, copper coating, and devices.

Steps:

- Top Menu - Place - Prohibited region - Rectangle

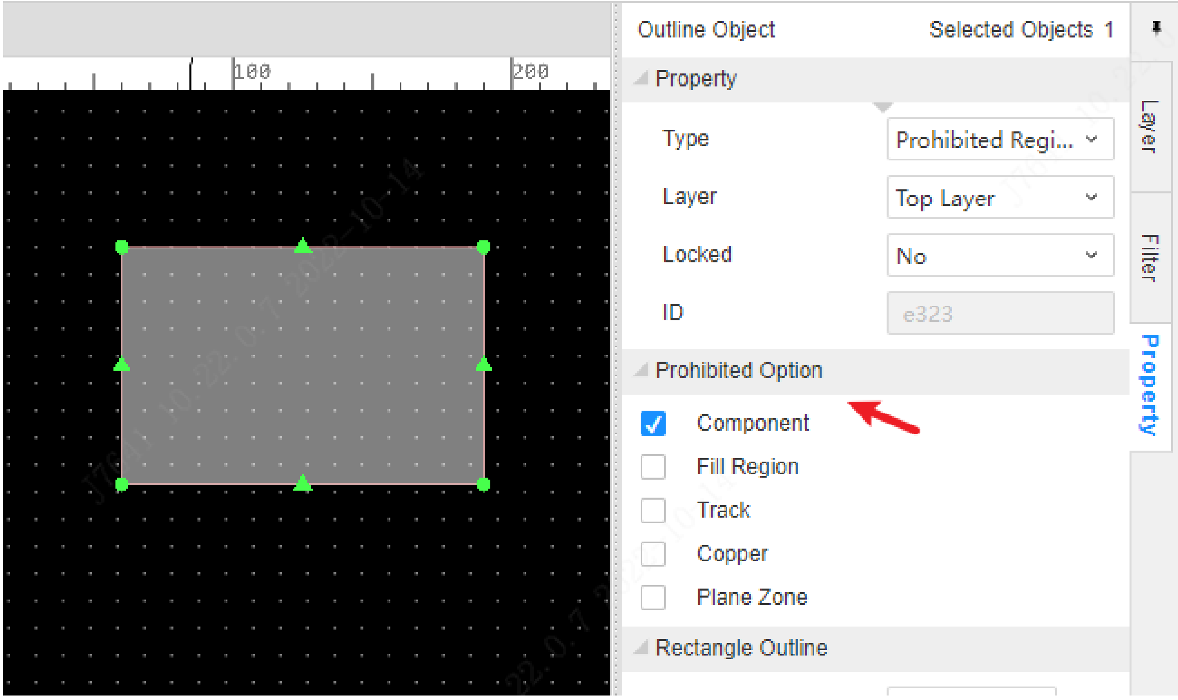


After the cross cursor appears, you can draw a rectangular prohibited region in the PCB, or you can enter the size value to be drawn in the dynamic input box



After drawing, the property pop-up window will pop up, where you can set the property of the prohibited region.

prohibited option



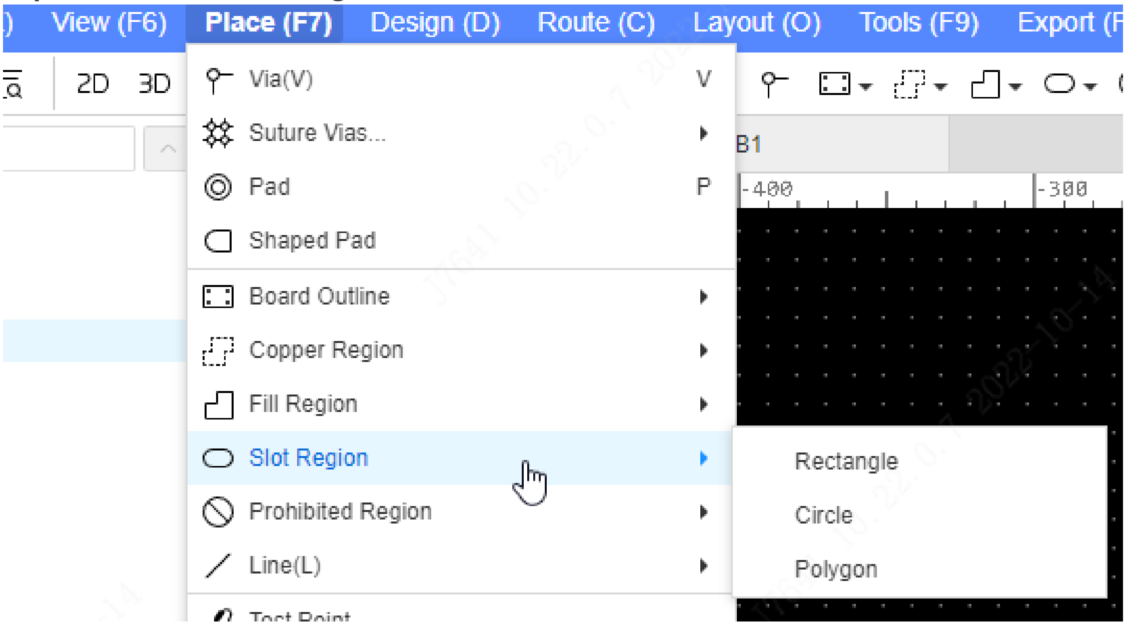
- **Component**: When checked, the currently drawn prohibited region cannot place components in it.
- **Fill region**: When checked, the currently drawn prohibited region cannot draw filled regions inside, nor can it be drawn into the prohibited region from outside. This option only for DRC checking, it doesn't like the copper option to cutout the fill region.
- **Track**: When checked, the currently drawn prohibited region cannot draw track inside
- **Copper**: After checking, the currently drawn prohibited region cannot draw the copper region in it, and the global copper-laying will cut the copper skin of the prohibited region.
- **Plane Zone**: After checking, the plane zone will be coutout the copper.

Slot Region

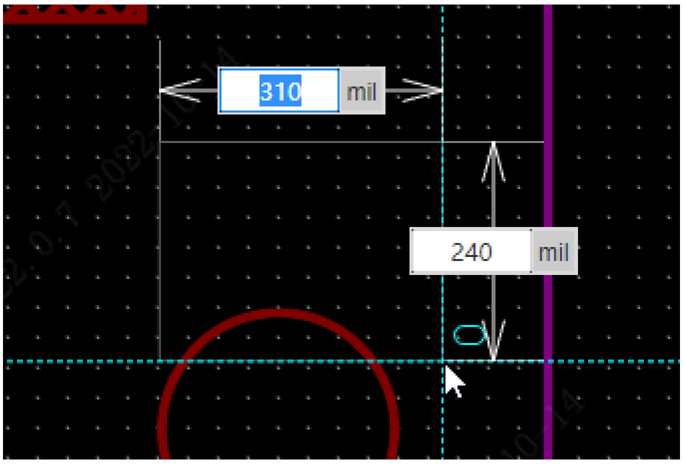
In PCB design, whether it is the creepage spacing of high-voltage boards or the requirements of the board structure, it is often encountered that the board needs to be grooved. The groove is to hollow out the designed PCB.

The specific operation steps are as follows:

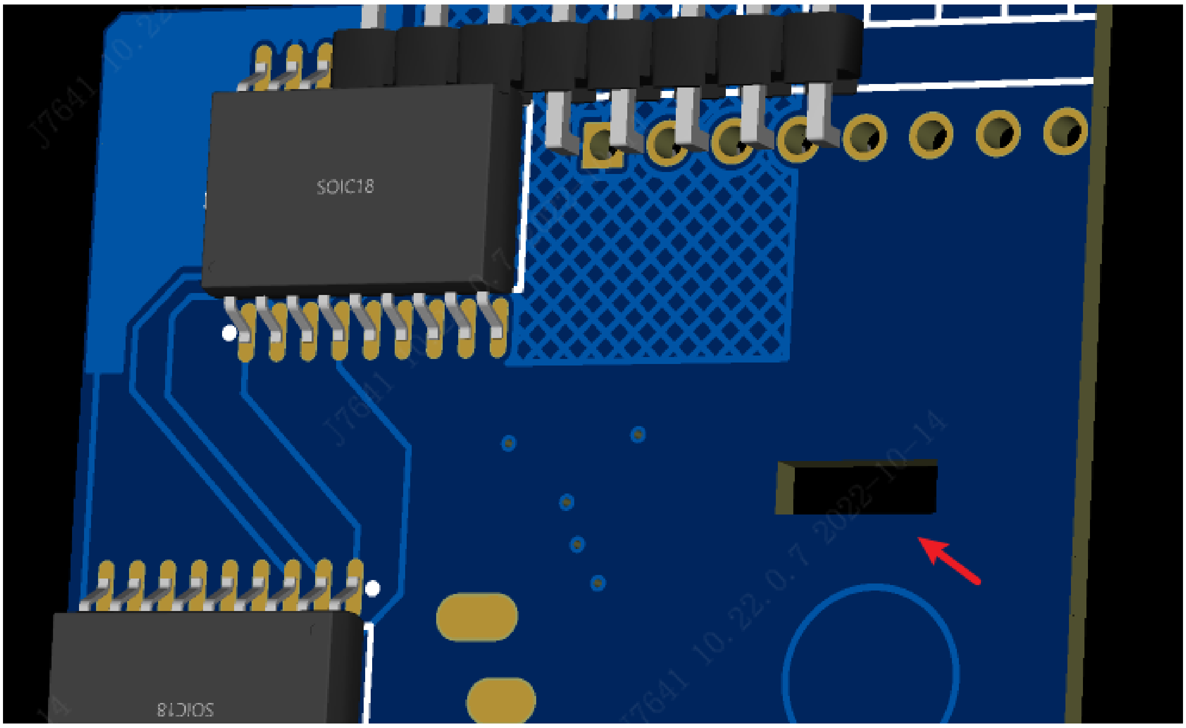
- **Top Menu - Place - Slot Region**



When the cursor turns into a cross, you can draw in the PCB



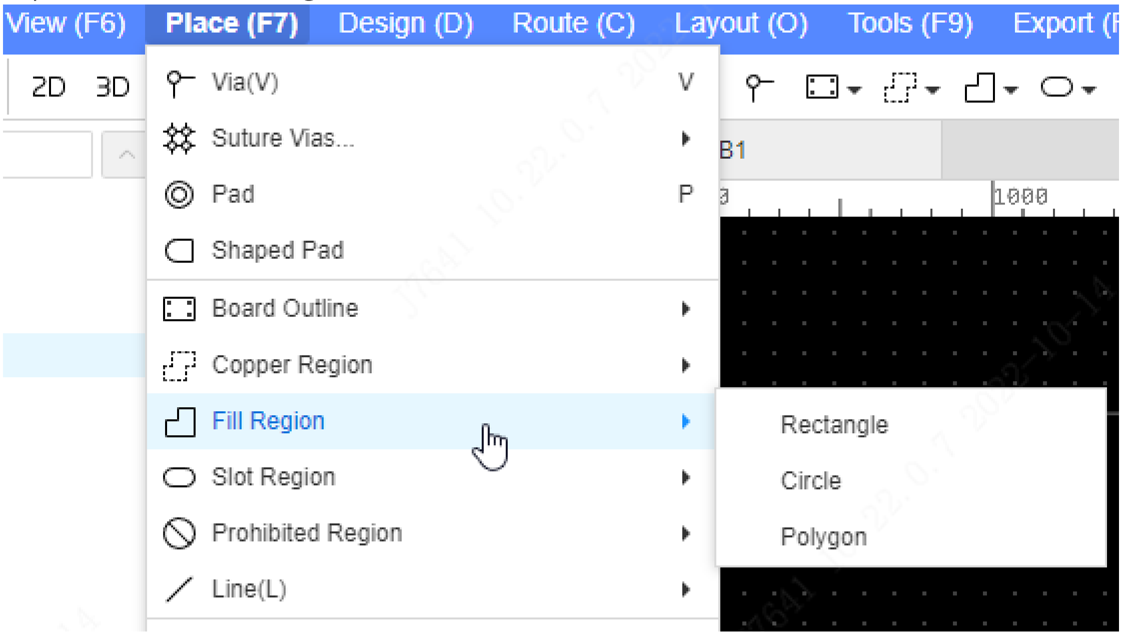
After drawing, the PCB has a slot region



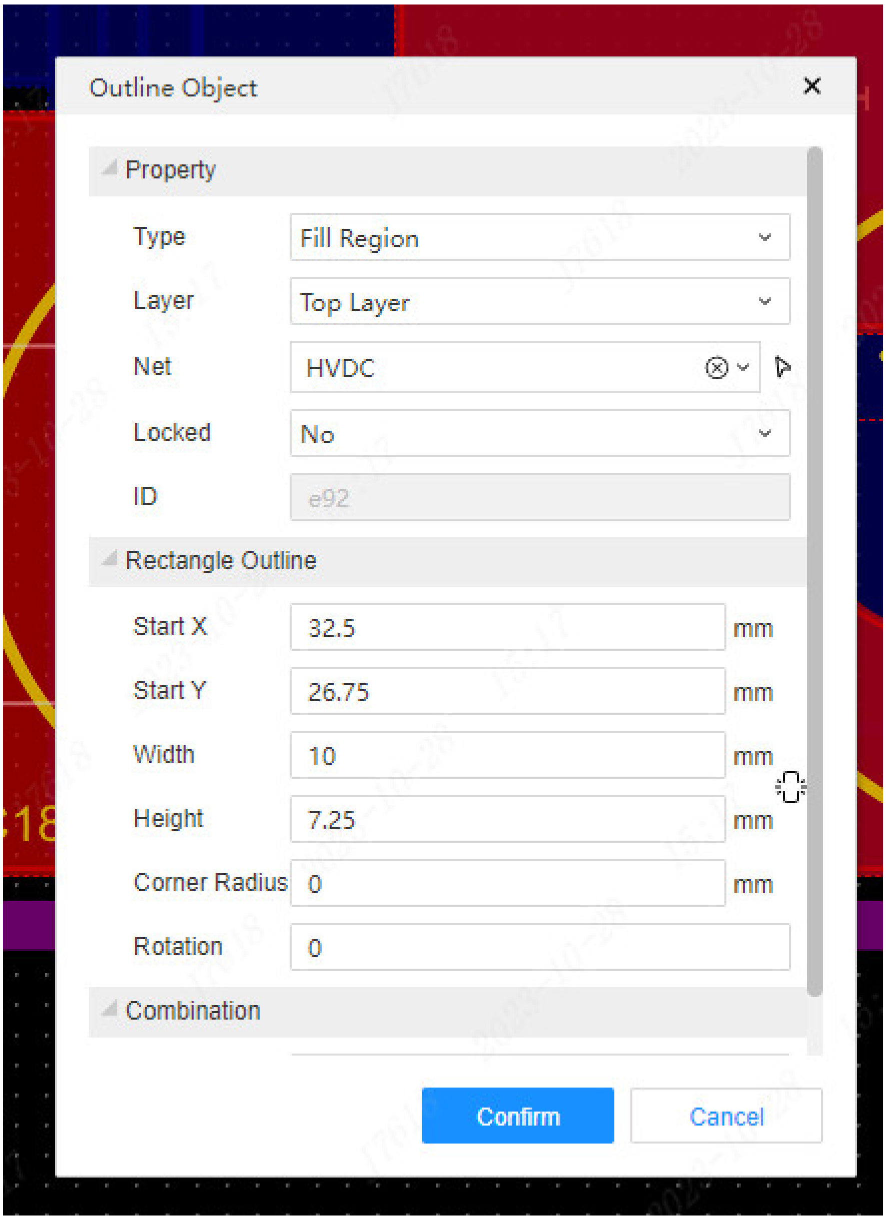
Fill Region

EasyEDA Pro provides a padding area. You can draw the required fill information, which is similar to copper, but solid fills cannot create gaps with elements of different nets.

- Top Menu - Place - Fill Region

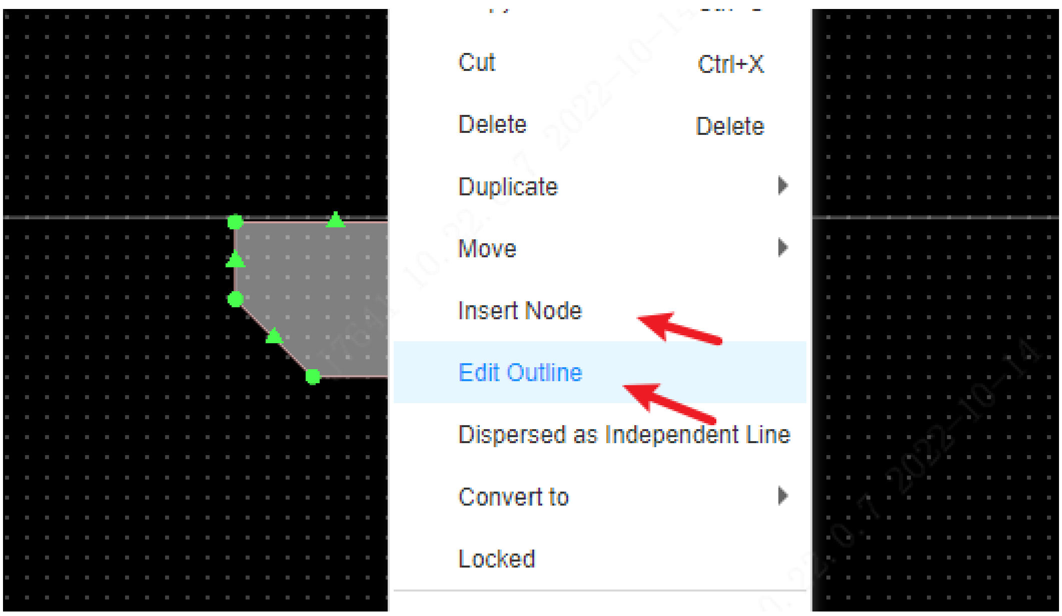


After drawing, the net is automatically set up for the filling area, and the layers and networks of the fill region can be modified by pop -up windows.



Attributes

- **Layer**: Supports switching solid fill to other layers: top layer, bottom layer, top silkscreen, bottom silkscreen, document, multi-layer, etc. These layers will only be displayed after the layer tool is enabled.
- **Net**: On top and bottom layers, or other inner signal layers, nets can be set to have electrical properties. If you use a solid fill to connect two pads directly, you need to set their nets to be the same, the solid fill needs to cover the center of the pad, and it needs to be connected with a single solid fill, otherwise the flying leads will not disappear.
- **Type**: Modify the filled area, the default is the filled region layer, and it is not recommended to change to other layers. The default fill area layer is sufficient.
- **Edit Outline**: Support solid filled coordinate point editing at right-click menu, edit the shape you need, including the shape of the arc.



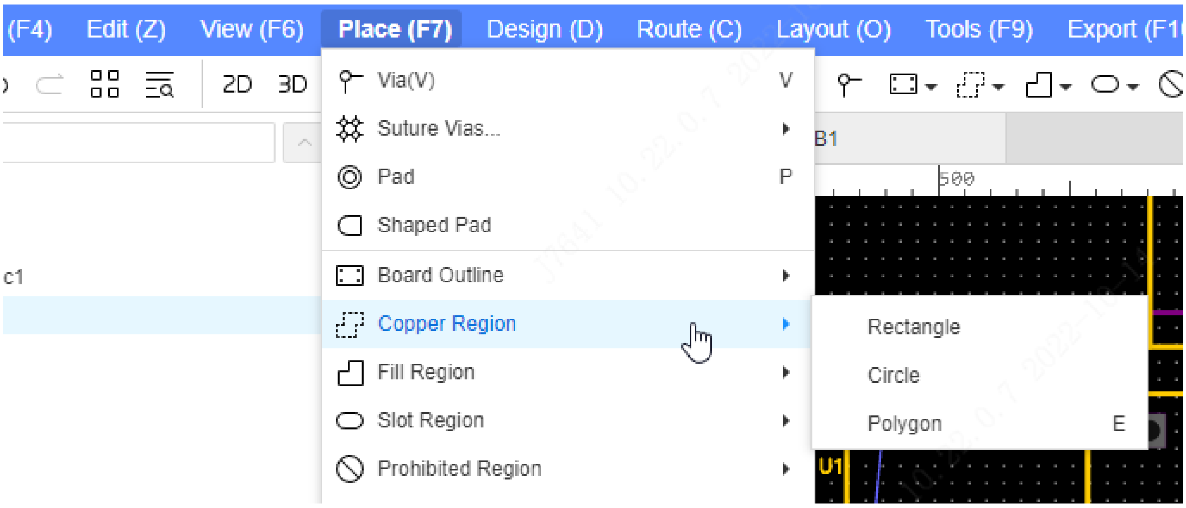
Copper Region

If you want to keep the entire copper area for grounding or power, you can use the "Copper" function.

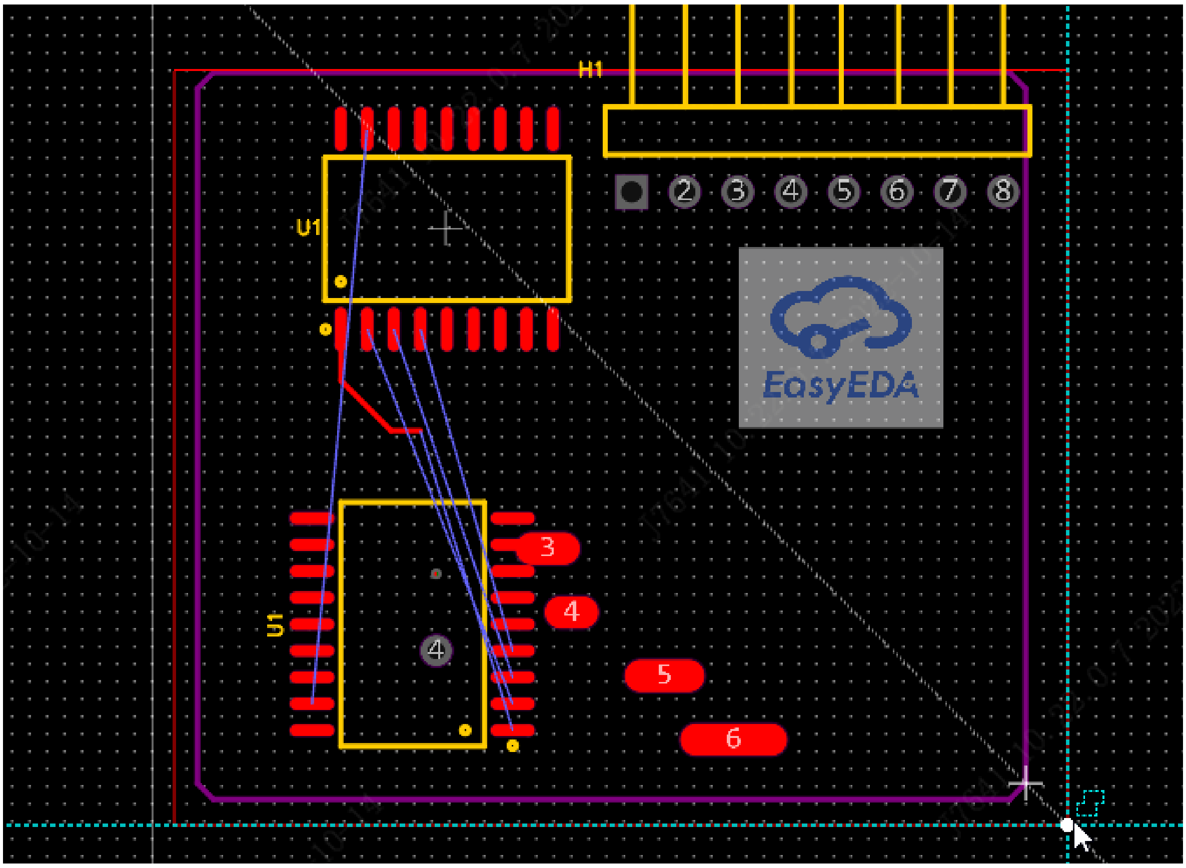
After clicking, you can draw the copper area around the area you want to copper. You can draw directly outside the board border, and you don't need to follow the board border. EasyEDA Pro will automatically cut the excess copper foil.

The top and bottom layers need to be drawn separately. A board can draw multiple copper areas and set them separately.

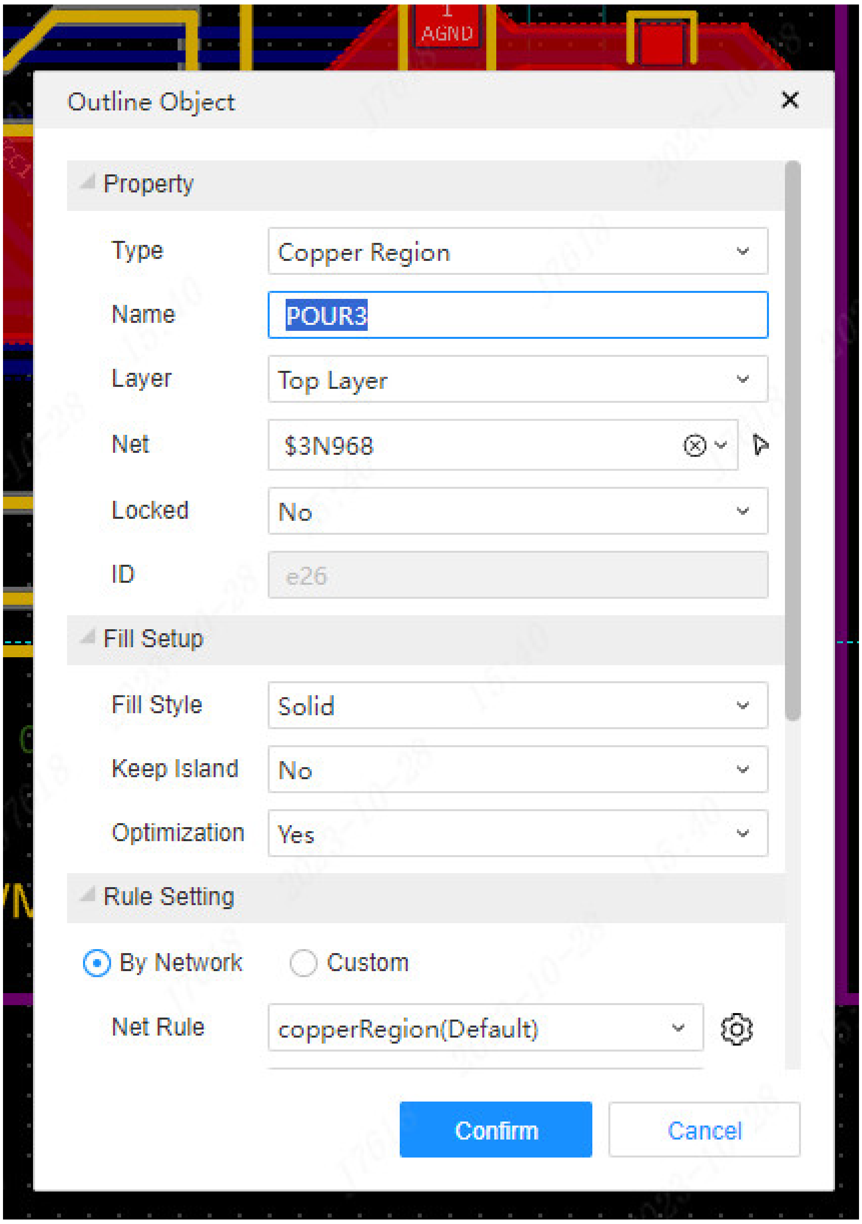
- **Top Menu - Place - Copper Region - Rectangle** (choose according to personal needs)



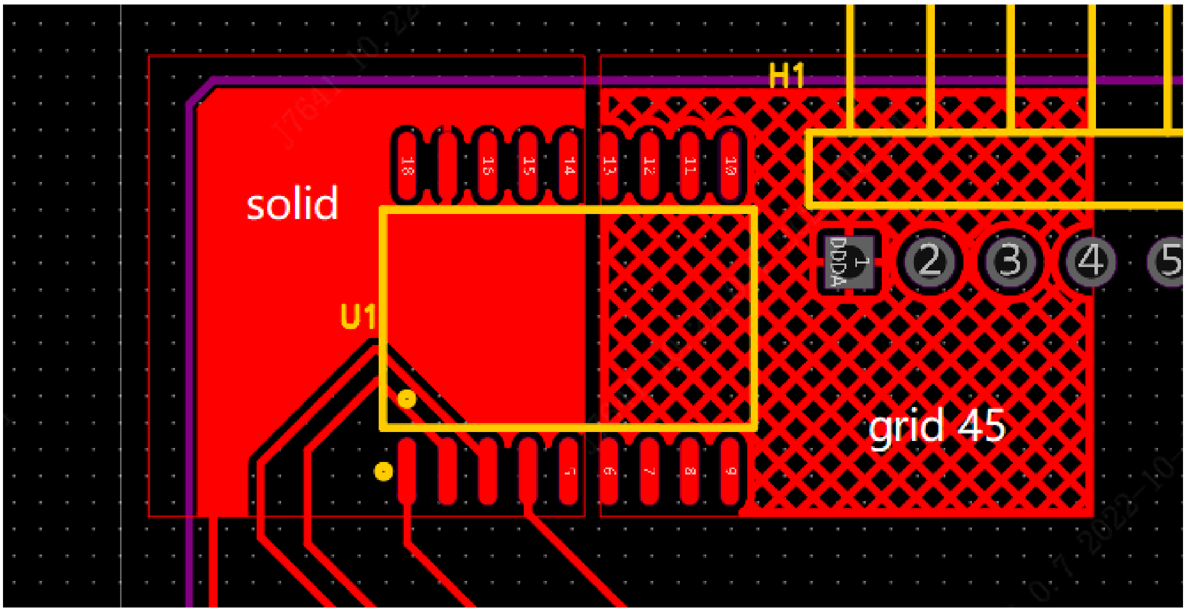
Draw rectangular copper skin,



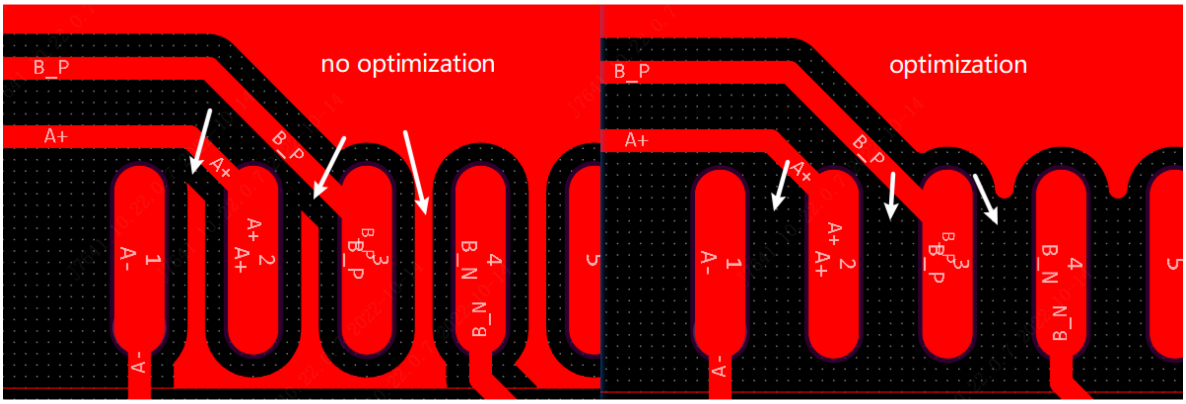
After drawing, the net that is connected to the copper region will be automatically set up,You can modify the layers, networks and other attributes of the copper area through pop -up windows.



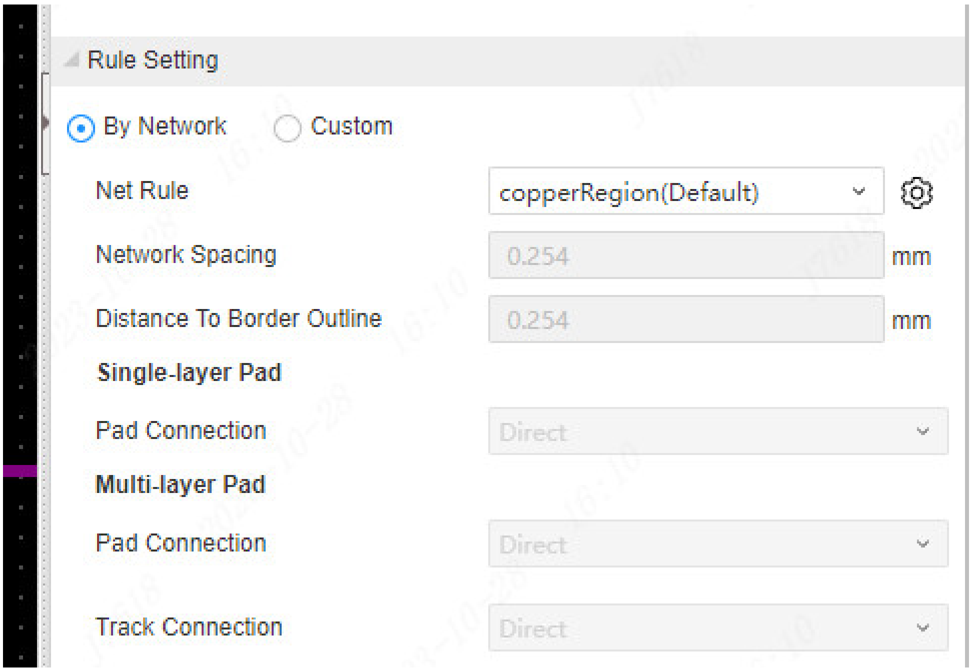
- **Type**: Outlie object defaults to copper region type;
- **Name**: Different names can be set for the copper region.
- **Layer**: The layer of the copper area can be modified: top layer. bottom layer. inner layer 1. inner layer 2. inner layer 3. inner layer 4. When the type of the inner layer is the inner electric layer, the copper layer cannot be drawn;
- **Net**: Set the network the copper foil is connected to. When the network is the same as the element network on the canvas, the copper can be connected to the element and it will be displayed, otherwise the copper will be considered as an island and be removed.
- **Locked**: Only lock the position of the copper. After locking, the size and position of copper laying cannot be modified through the canvas;
- **Fill Style**: Full Fill: Normal copper fill style; 45 degree net: The area is filled with a 45 degree grid fill. 90 degree net: The copper fill in this area is a 90 degree net type.



- **Keep islands**: yes or no. That is, whether to remove dead copper. If there is no network set for copper region, then the entire copper region will be regarded as dead copper and removed. If you want to keep the copper region, you can choose to keep the island or set up an existing network on the PCB for the copper region, and rebuild the copper region;
- **Optimization**: Appears only when the fill style is full fill, and the mesh copper region enables manufacturing optimization by default. The default is yes, the sharp corners and thin copper wires less than 8mil will be removed, which is convenient for production; set to otherwise, the sharp corners and thin copper wires will be displayed;

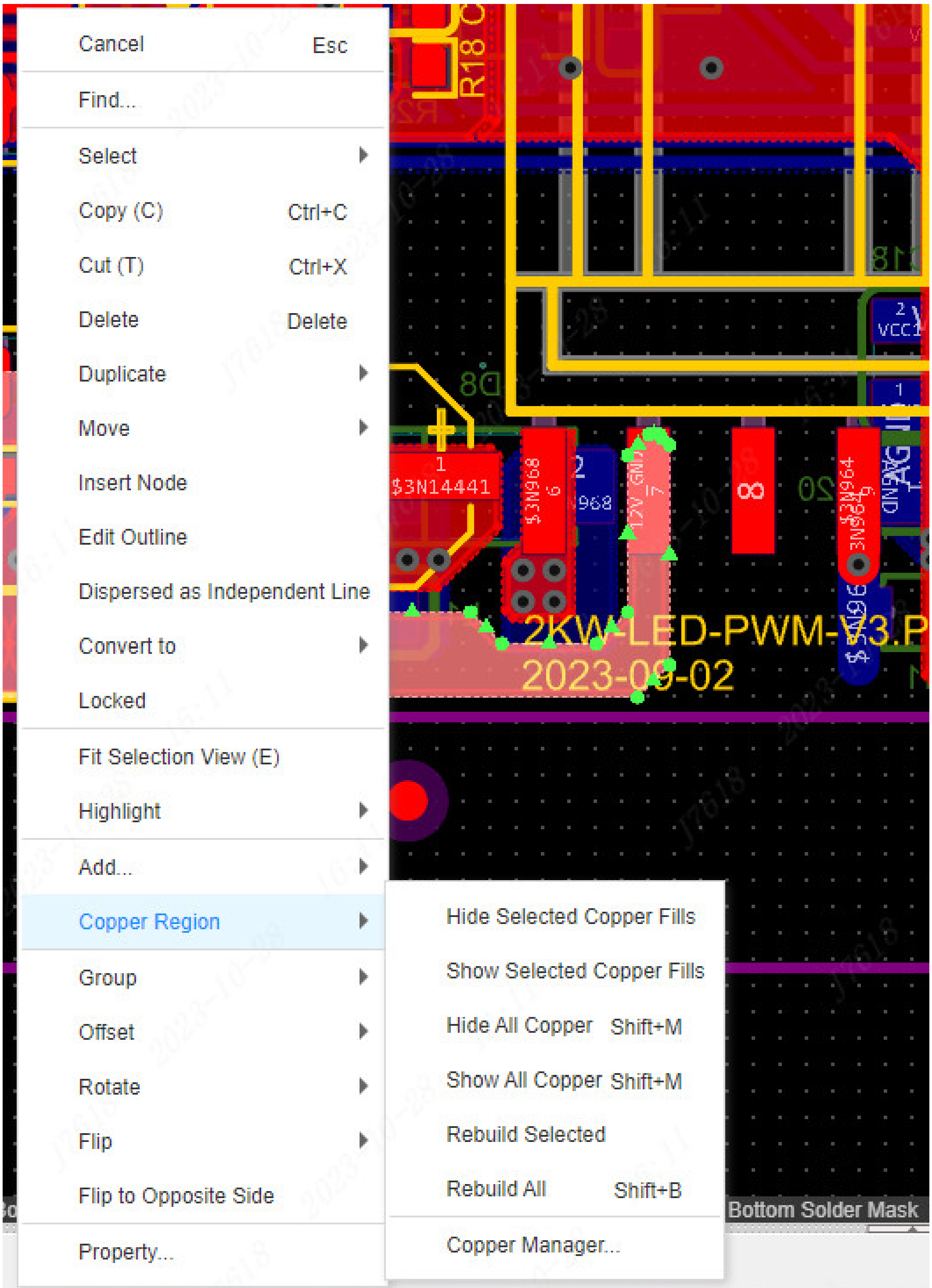


- **Rebuild copper region**: Rebuild according to the selected copper.The shortcut key Shift+B will rebuild all copper (including the internal electrical layer).
- **Place/remove suture via**:Automatically placed suture holes (batch holes) according to the selected copper.
- **Rule setting**: You can switch the copper paving rules and custom copper paving rules according to the network.It does not support the attributes of copper paving directly.



Right -click menu

The right -click menu after selecting the copper area provides a lot of fast functions.

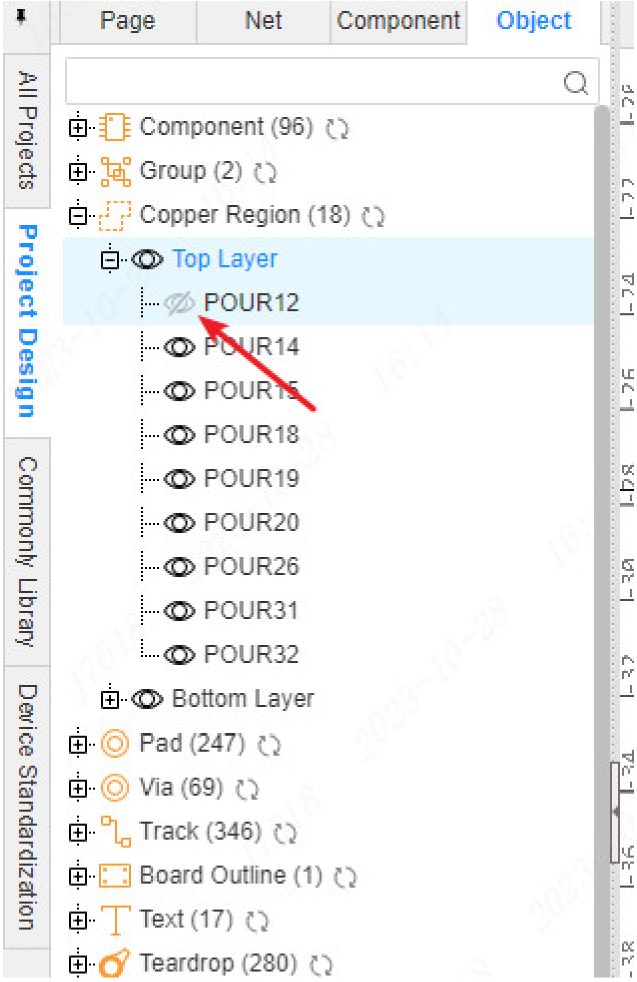


- Edit contour: Modify the outline shape of the copper.
- Turn to: Turn the copper -paving area to a fill area, prohibit the area, etc.
- Bald copper area: can display hidden copper -paving areas; rebuilding copper -paving areas.

Hidden display copper

EasyEDA supports a variety of functional entrances to hidden copper paving.

- Click the copper right -click menu or the top view menu for hidden display
- Click on the right attribute panel on the right side of the canvas blank, hide the copper area
- Click on the left object tree, click the copper -paved eye icon to hide



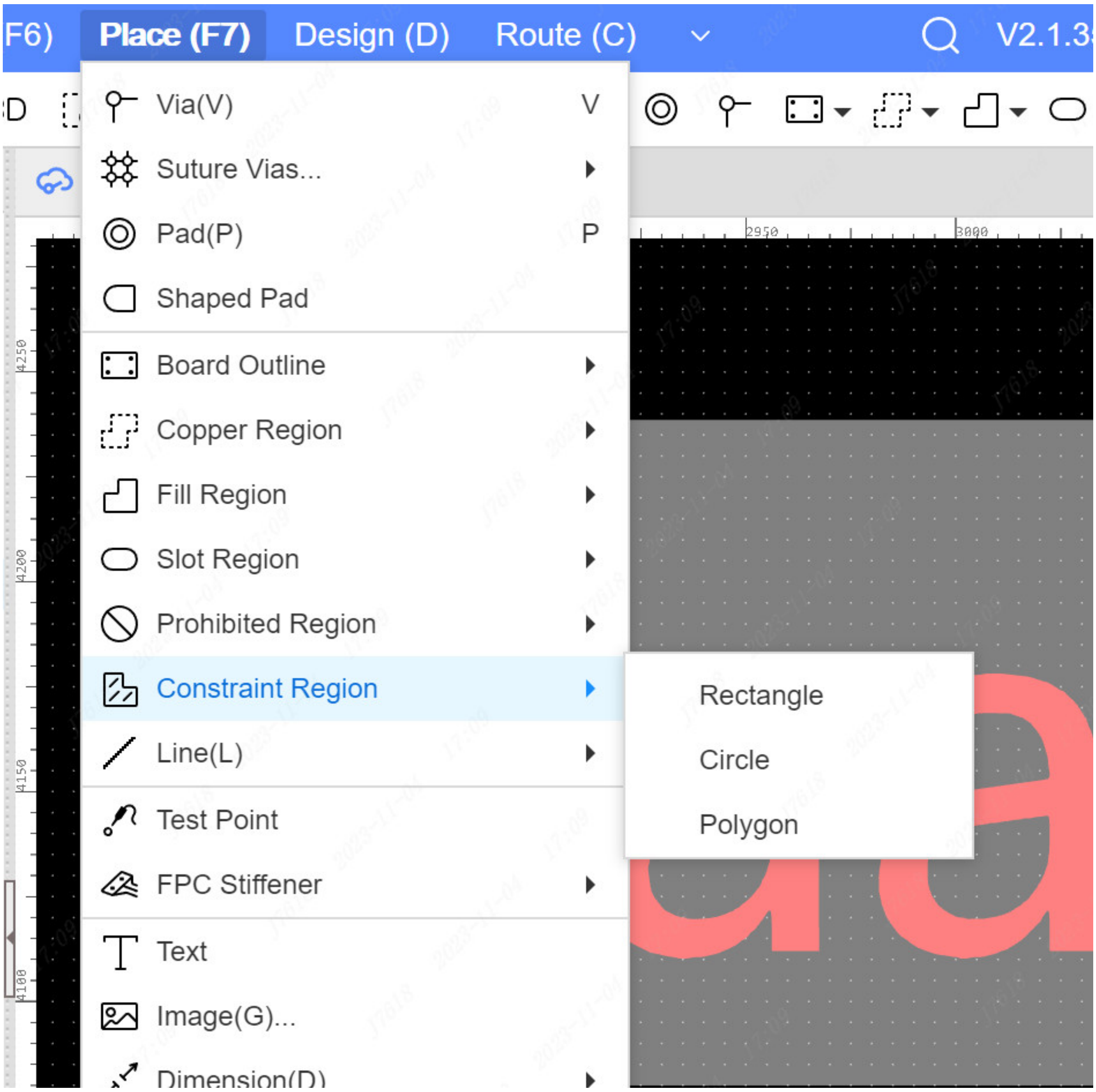
- Click on the right filter panel and click the copper -paved eye icon to hide

Constraint Region

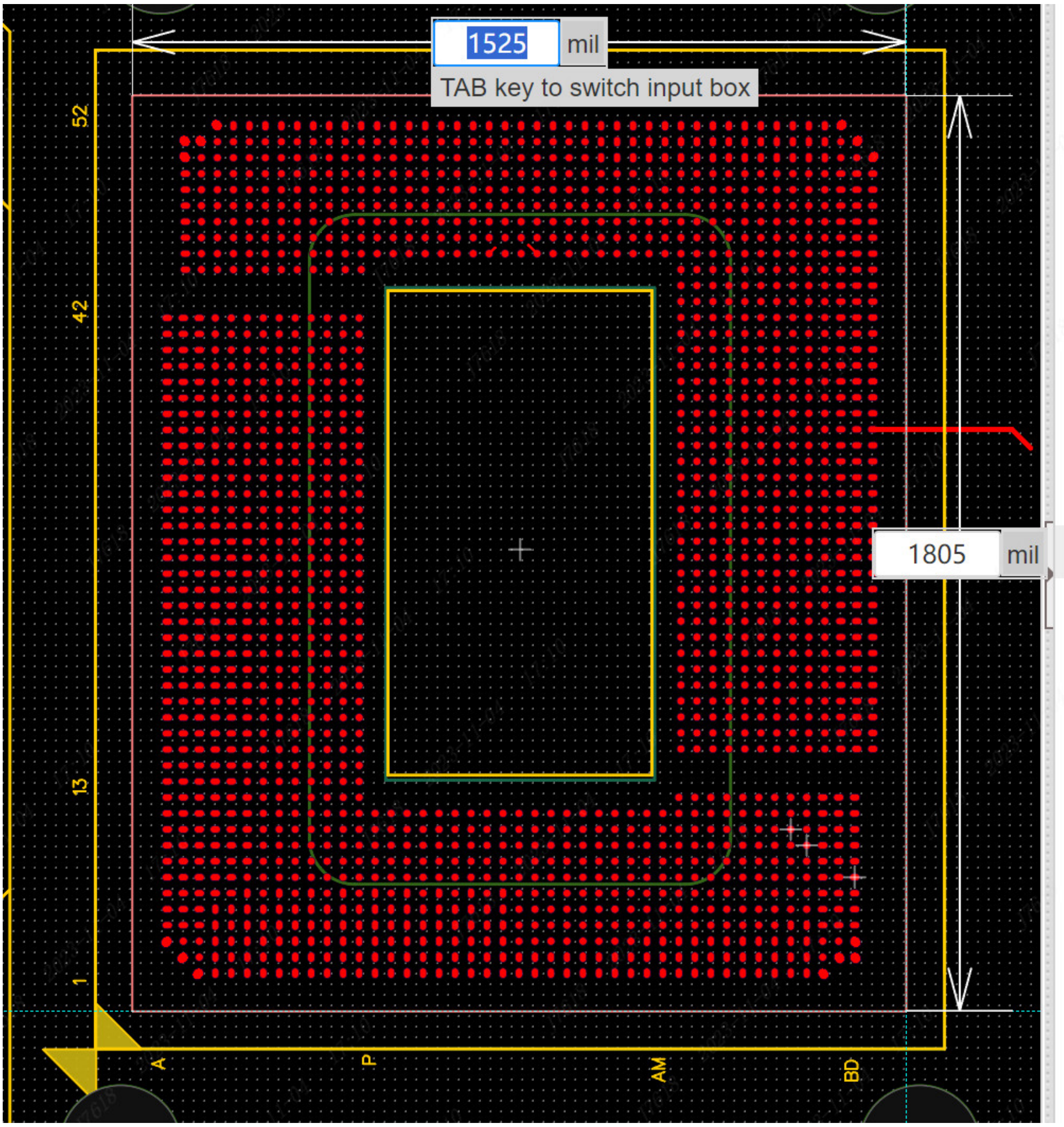
Constraint region is mostly used in PCB design containing BGA. After drawing a constraint area, special rules can be set on the area.When wiring, punching, and copper -laying operations in the area, you will interact in priority in accordance with the width, over -hole size, and spacing of the rules. When operating outside the area, according to the normal rules,。

Place the constraint area

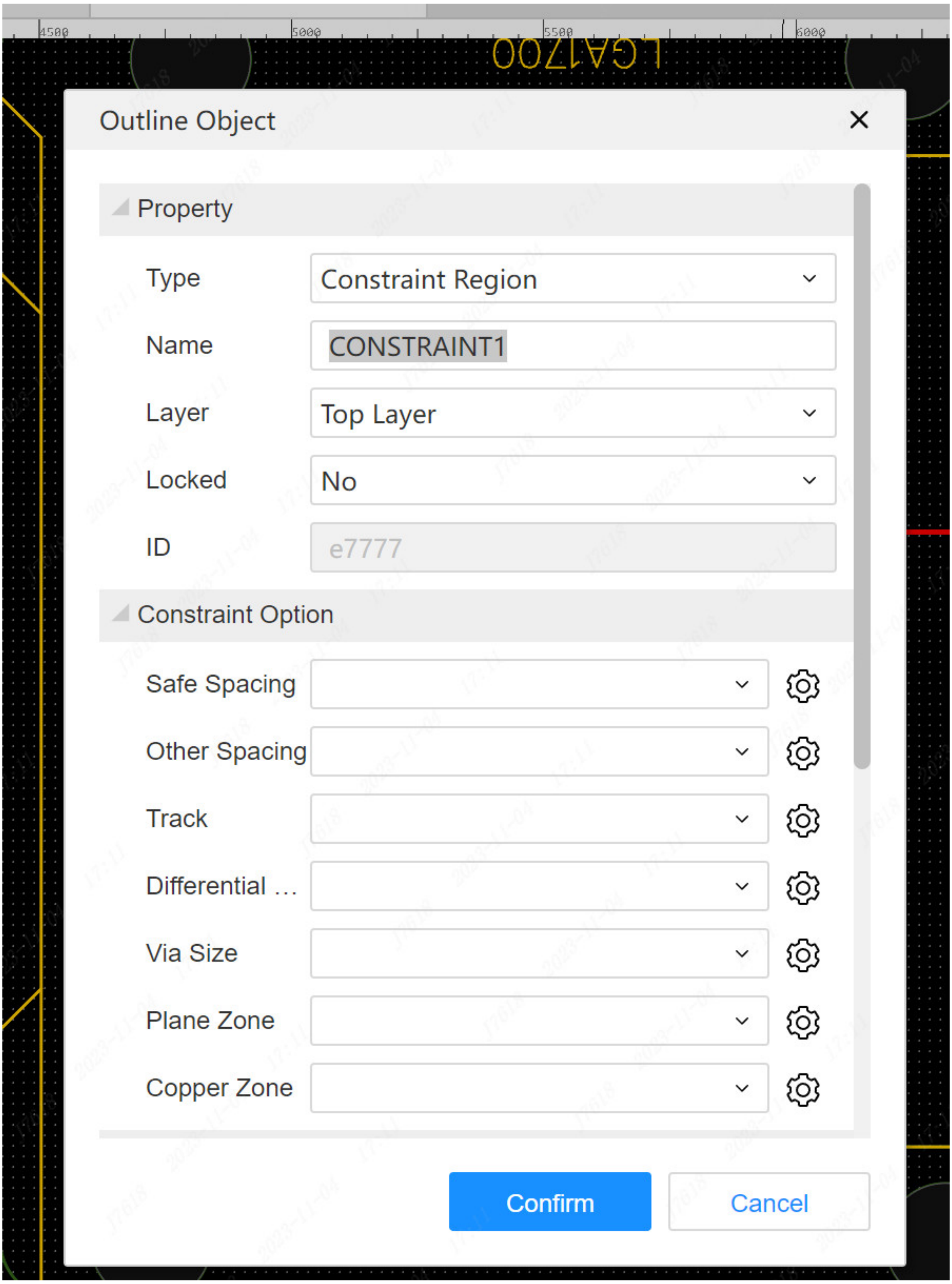
Steps: Top menu-place-constraint region



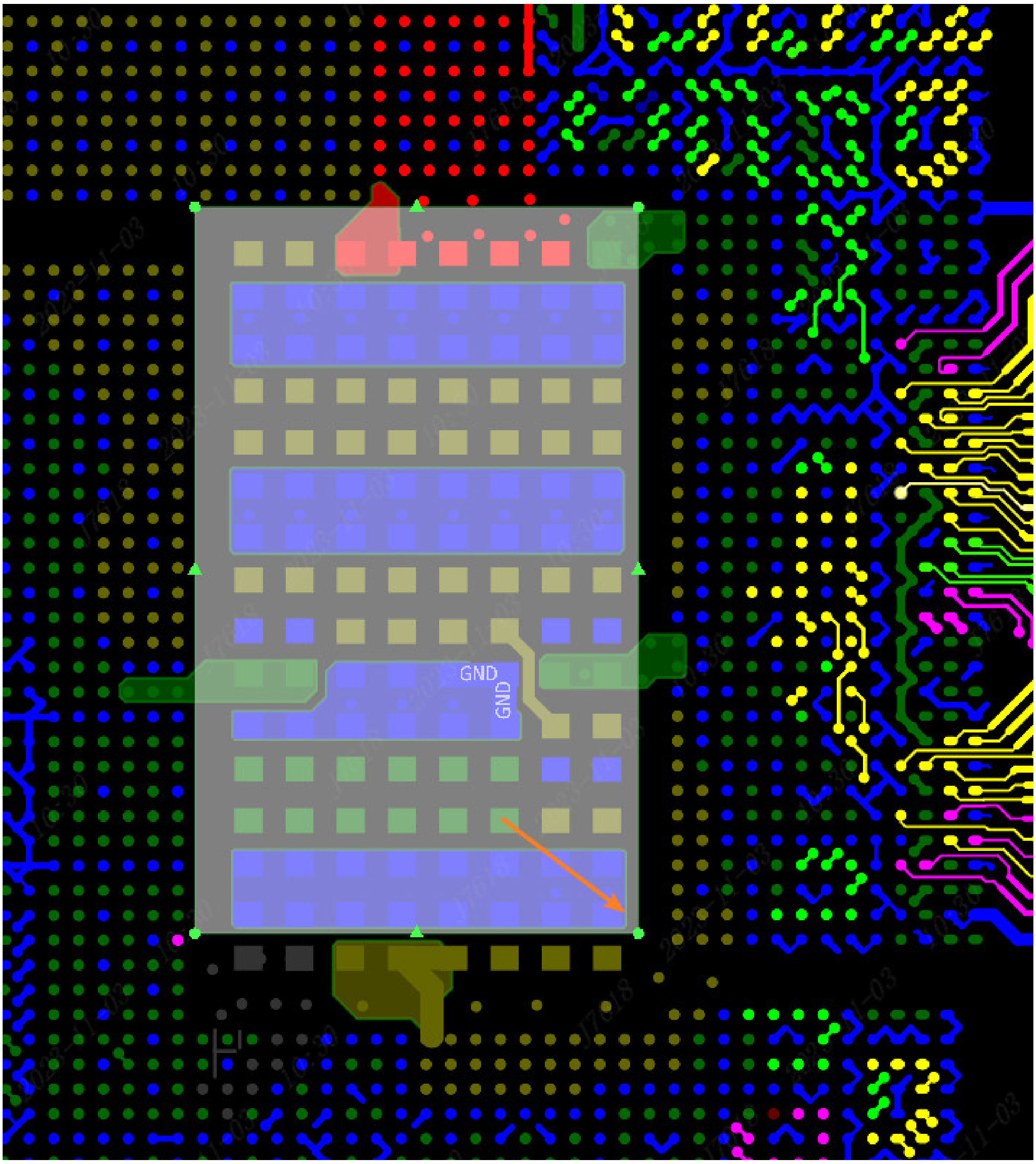
1. After selecting the rectangular, circular, and polygonal shape through the top menu, the canvas enter the cursor mode, and start to draw regional shape.



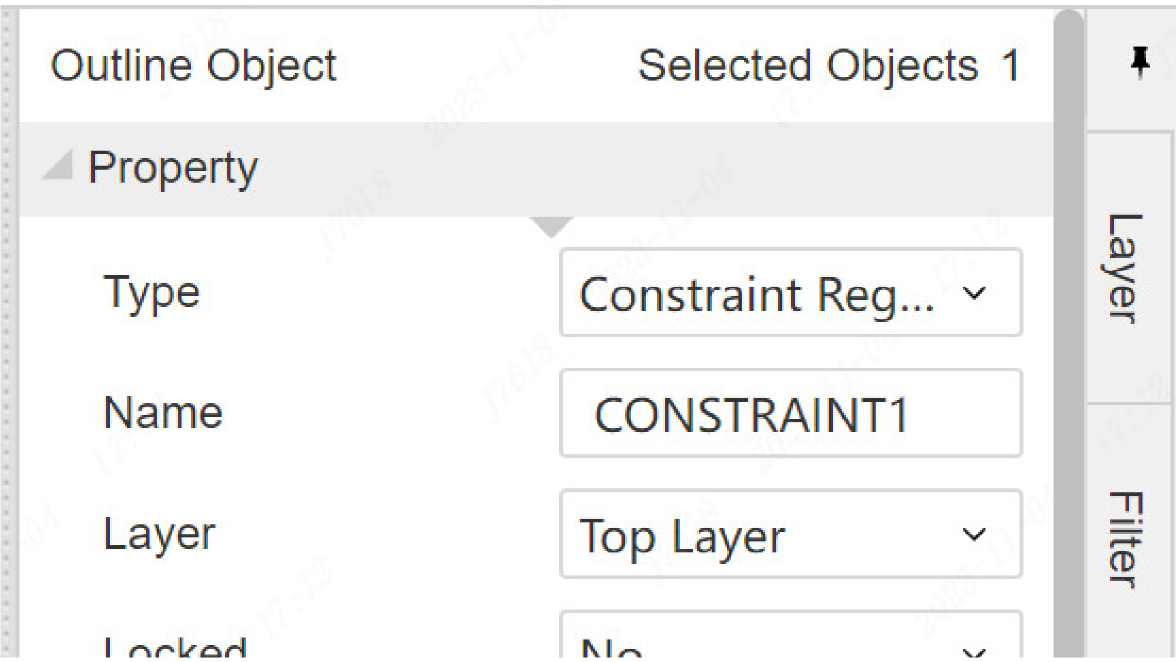
2. After drawing a shape, the attribute pop -up window will appear in the constraint area. You can set the relevant attributes in advance. You can also click to confirm first, and modify it on the attribute panel.

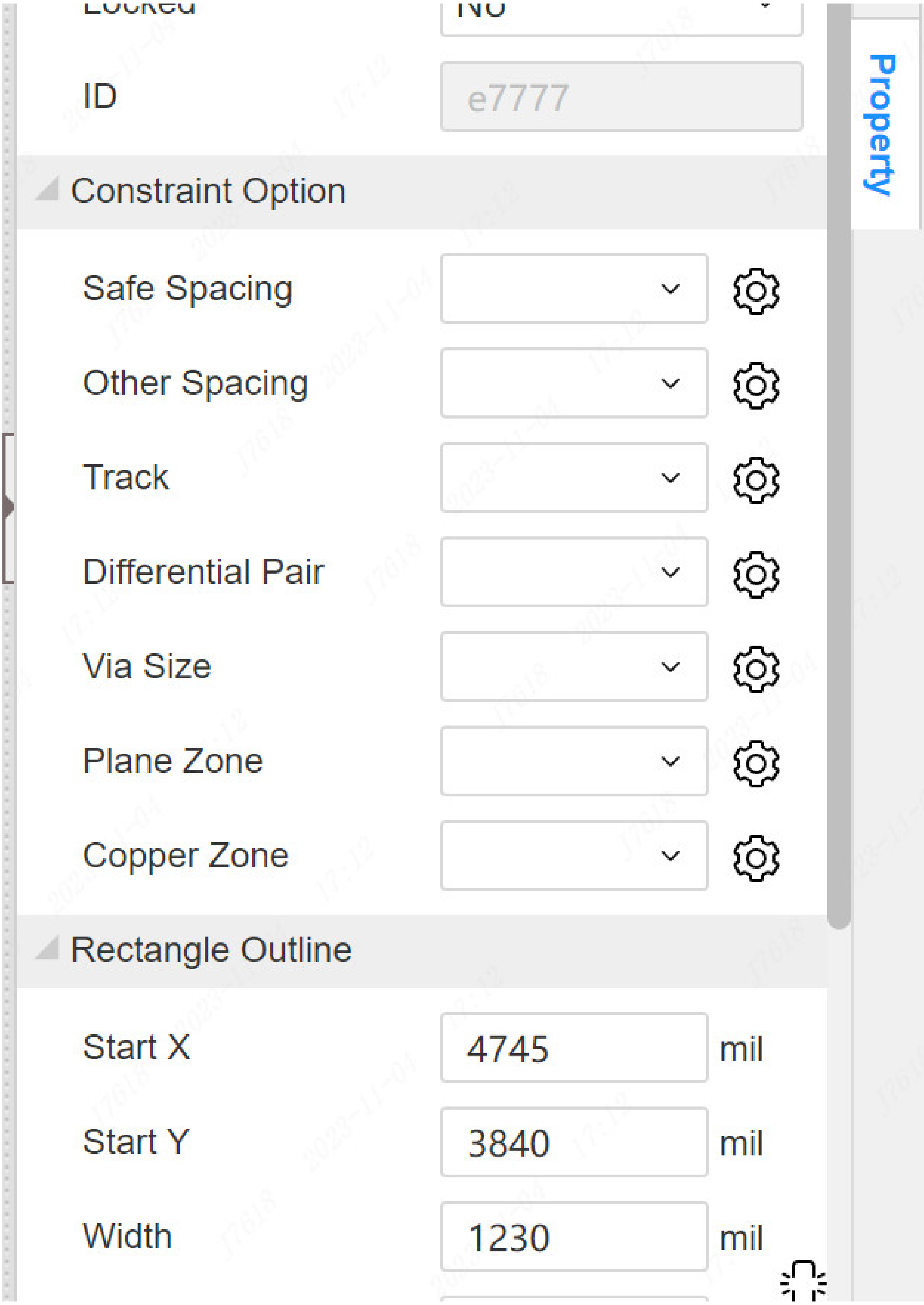


3. After the placement is completed, if the shape of the constraint area needs to be fine - tuned, you can drag the control point to adjust the control point, which is the same as the operation of the outline object such as the adjustment of the board frame.

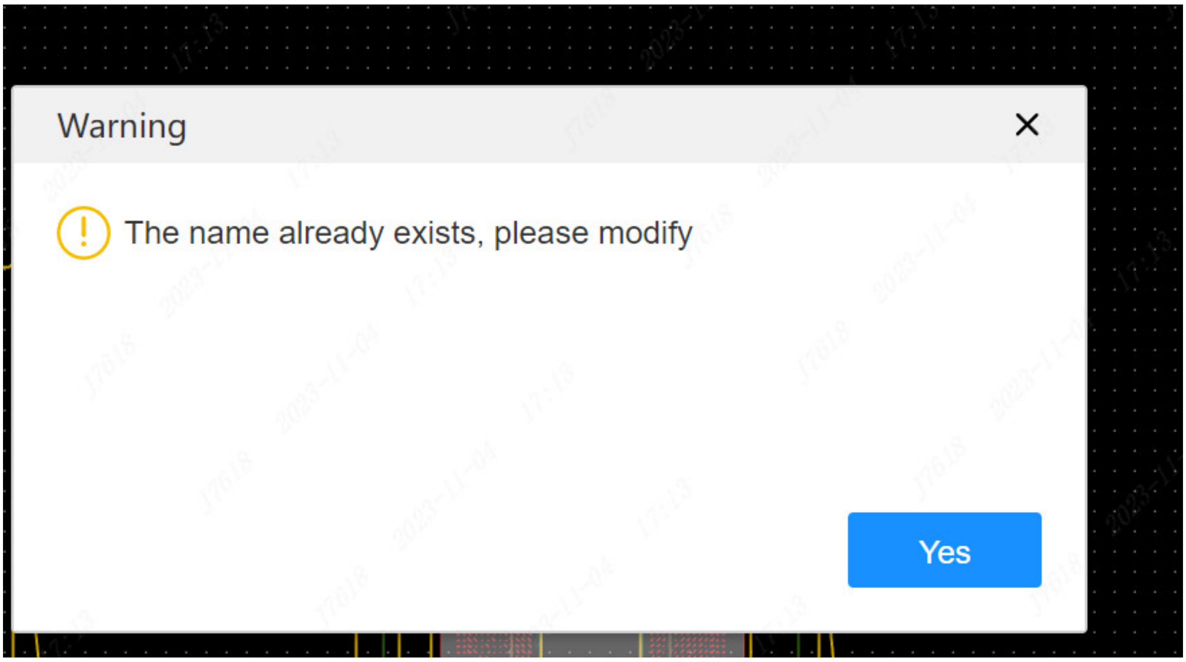


constraint region property

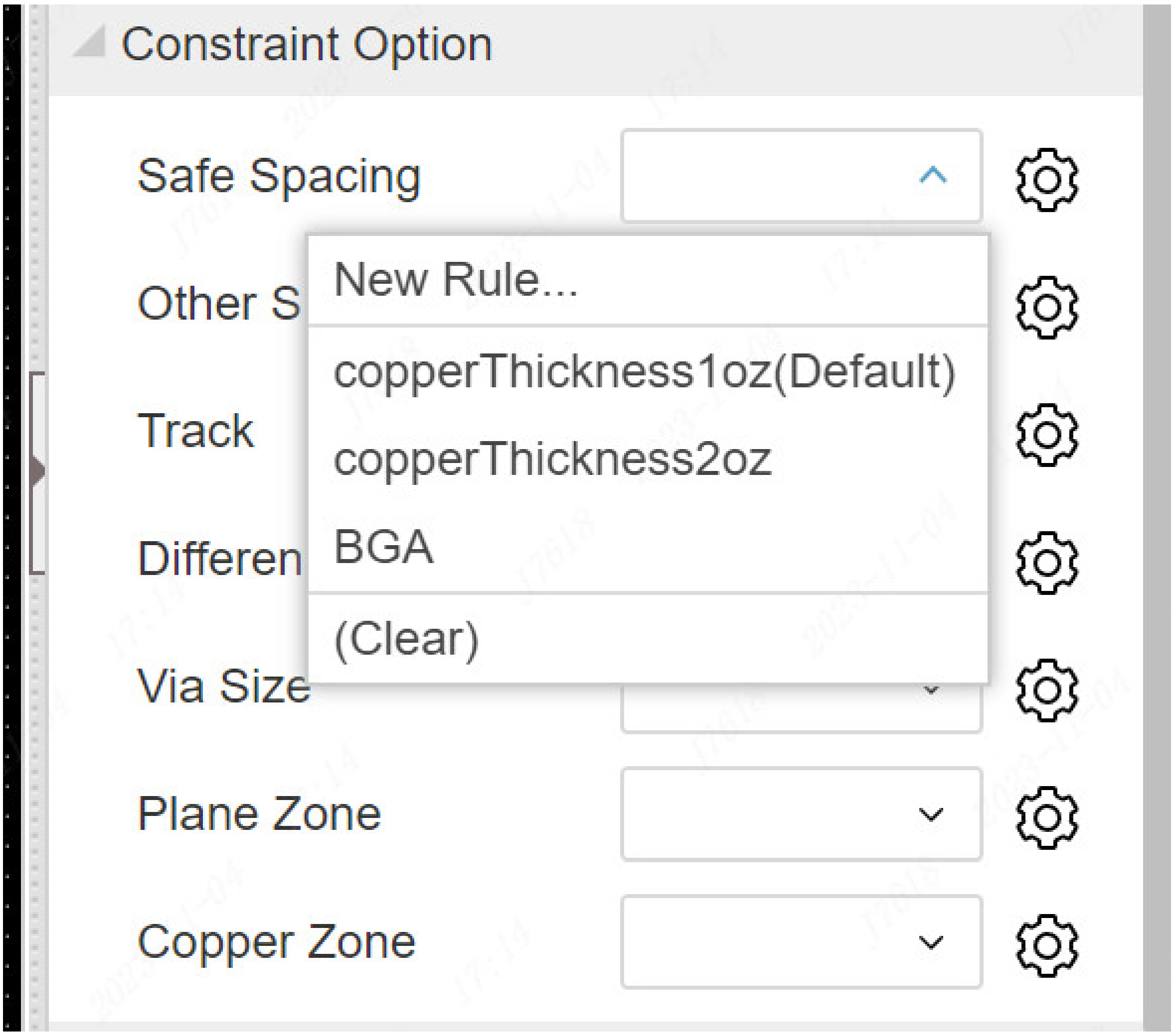




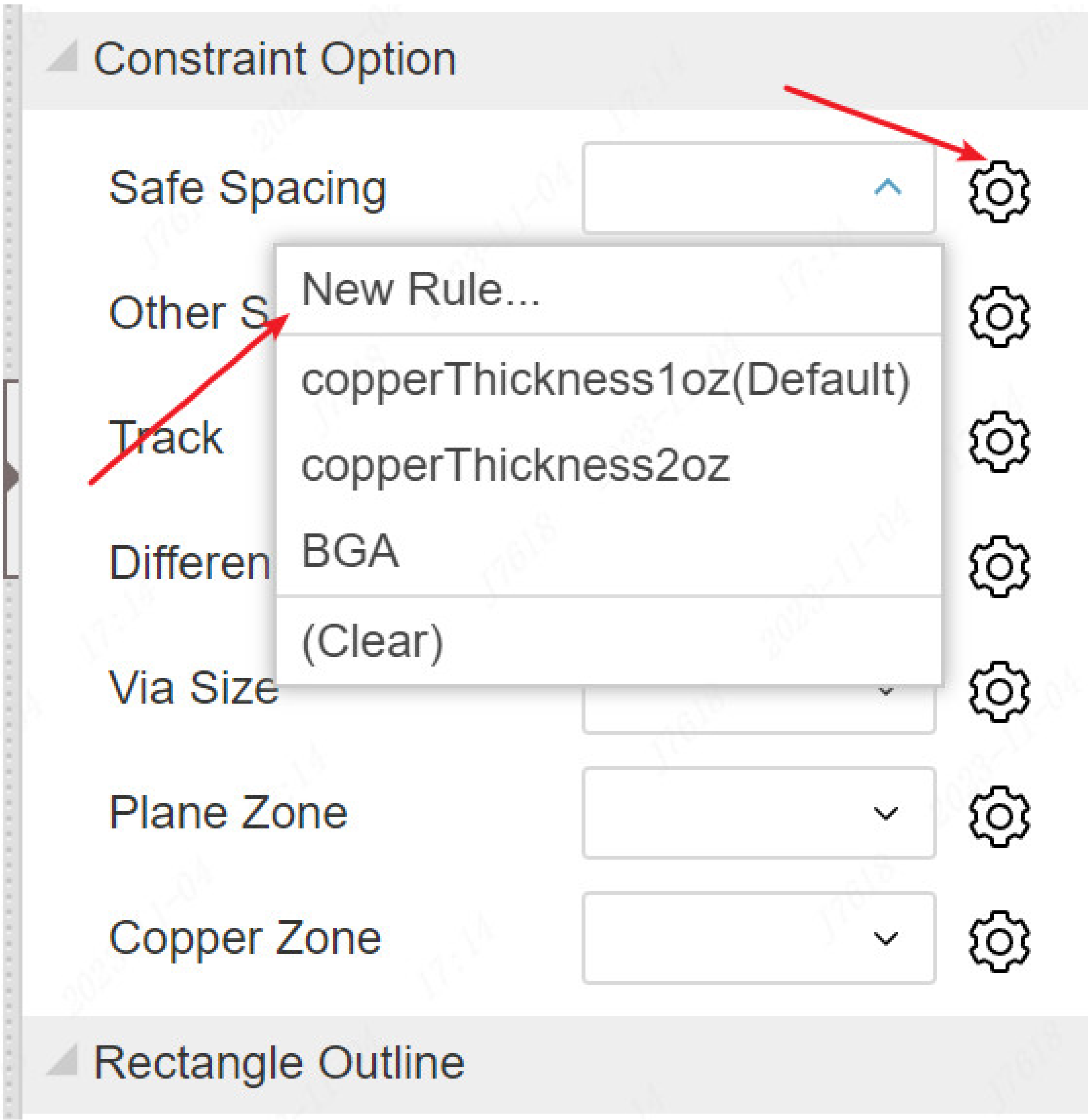
- **Type**: The constraint area is a contour object system. Pulling can be switched to other types, such as filling areas and copper -paving areas.
- **Name**: The name of each constraint area is unique. This name will be used when the regional rules are allocated in the design rules.If the modified name is a already existed name, it will not be successful.

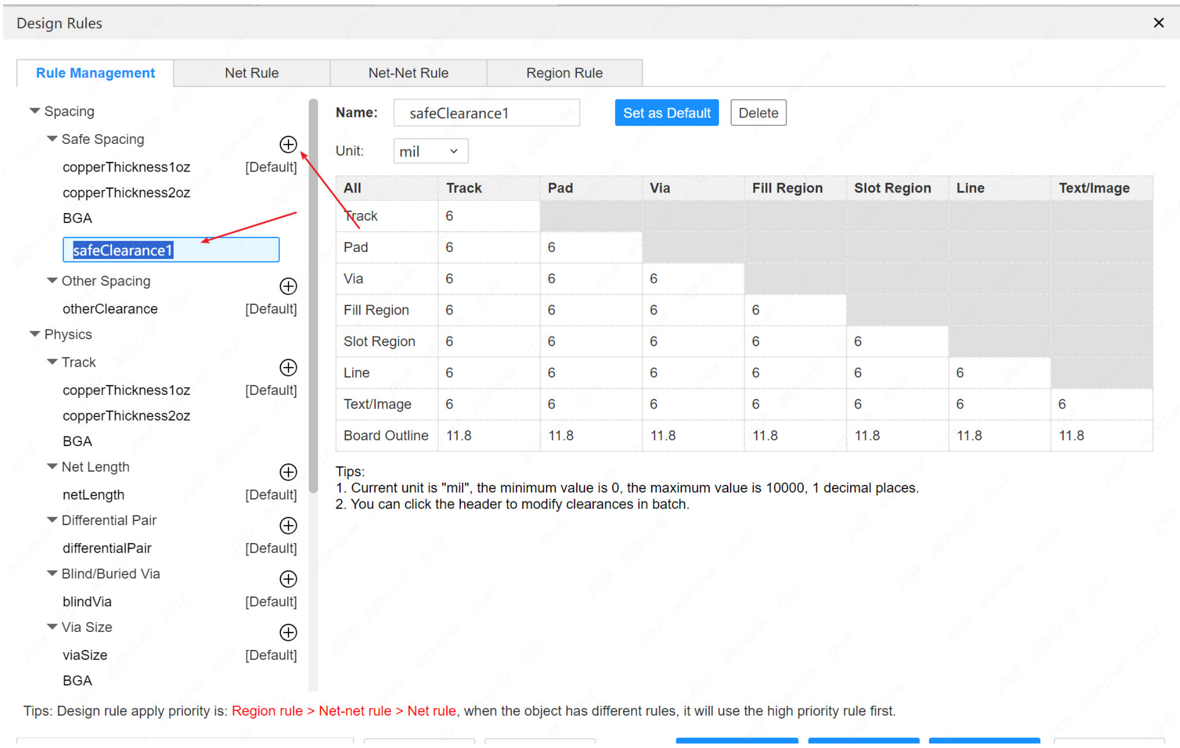


- **Layer**: The constraint area can be set in any copper foil layer or multi -layer.When the constraint area is set in a single layer, the DRC examination will only have the rules in the restraint area of this layer; when the constraint area is set in a multi -layer, the part of all copper foil layers in the constraint area will apply the rules in the constraint area.
- **Constraint Option**: In the attribute panel, you can directly pull down and switch the corresponding rules in the area. Another way is to enter the design rules dialog box for allocation.All rules are empty default. For the air, it means that there is no special rules allocated in this area, and DRC calculation is performed according to normal without rules.When a special rule needs to be assigned to the area, if a smaller rule needs to be set up, it is necessary to set up a smaller rule, and directly pull down and switch to the corresponding rules.



If the rules in the area have not been set in advance, you need to enter the rules management of the design rules first, and you can directly click the setting icon or drop -down option of the corresponding rules.

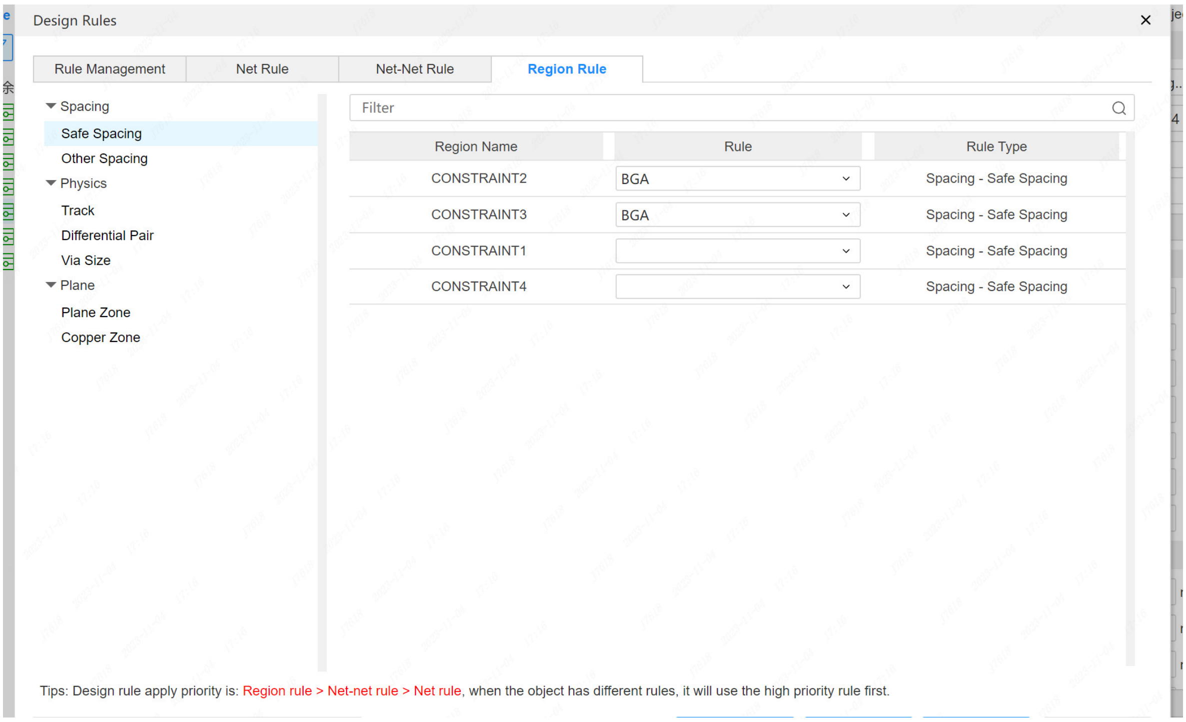




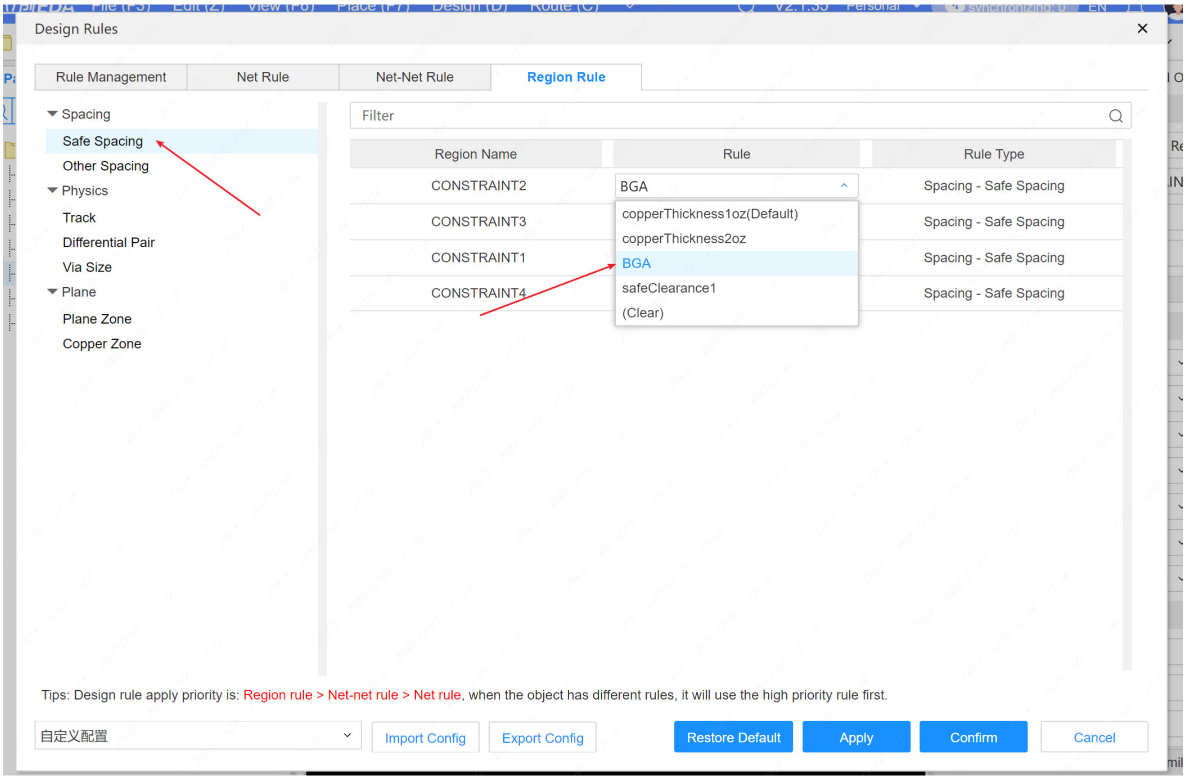
Constraint Region Rules

In addition to allocating the attributes of the constraint area to allocate rules in the region, you can also enter the design rules dialog box for the distribution of regional rules.

Steps: Top Menu-Design-Design Rules, the top TAB switch to region rule.



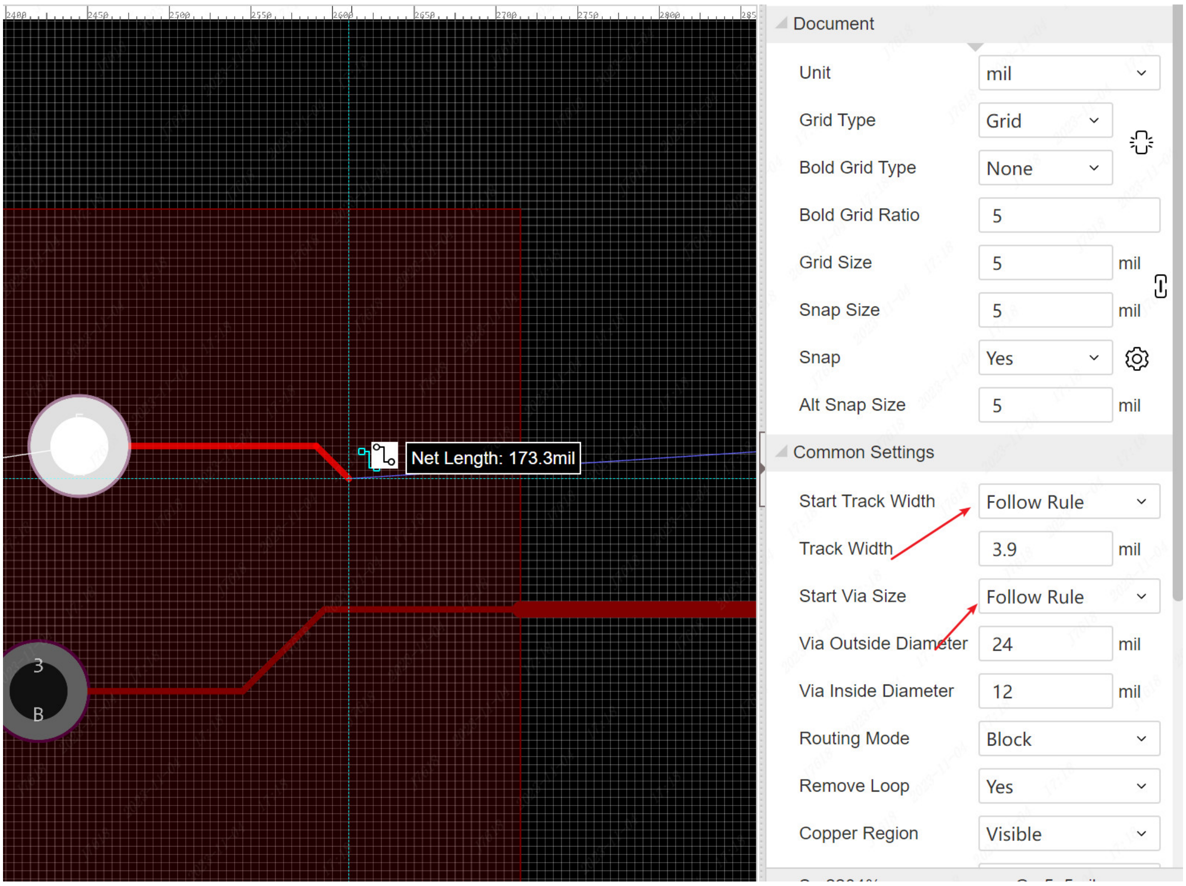
The left tree is selected by the left tree under the page, and the right side of the right side is pulled down and switching the rules for lowering different areas.



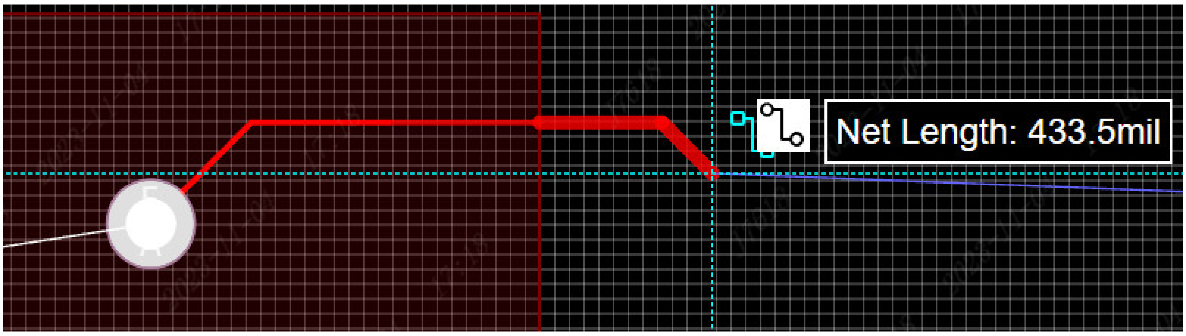
Constraint Region Interaction

Set up the rules' constraint areas, and different effects will be produced when the wiring interaction is performed.

When the start width and starting size are set to follow the rules, the wiring starts from the interior of the constraint area, and will be based on the width rules, differential rules, spacing rules, over -hole size, etc. in the region, etc.The rules are qualified.



When wiring from the region to the region, or from outside the area to the region, the width rules of the inner and outer wires are different, and the width of the wire will be automatically changed in the inner and outer boundary.



Rules allocate priority: regional rules> network-network rules> network rules.

Real -time DRC and DRC inspections, paving copper, and special rules in the constraint area, will be calculated in accordance with regional rules.

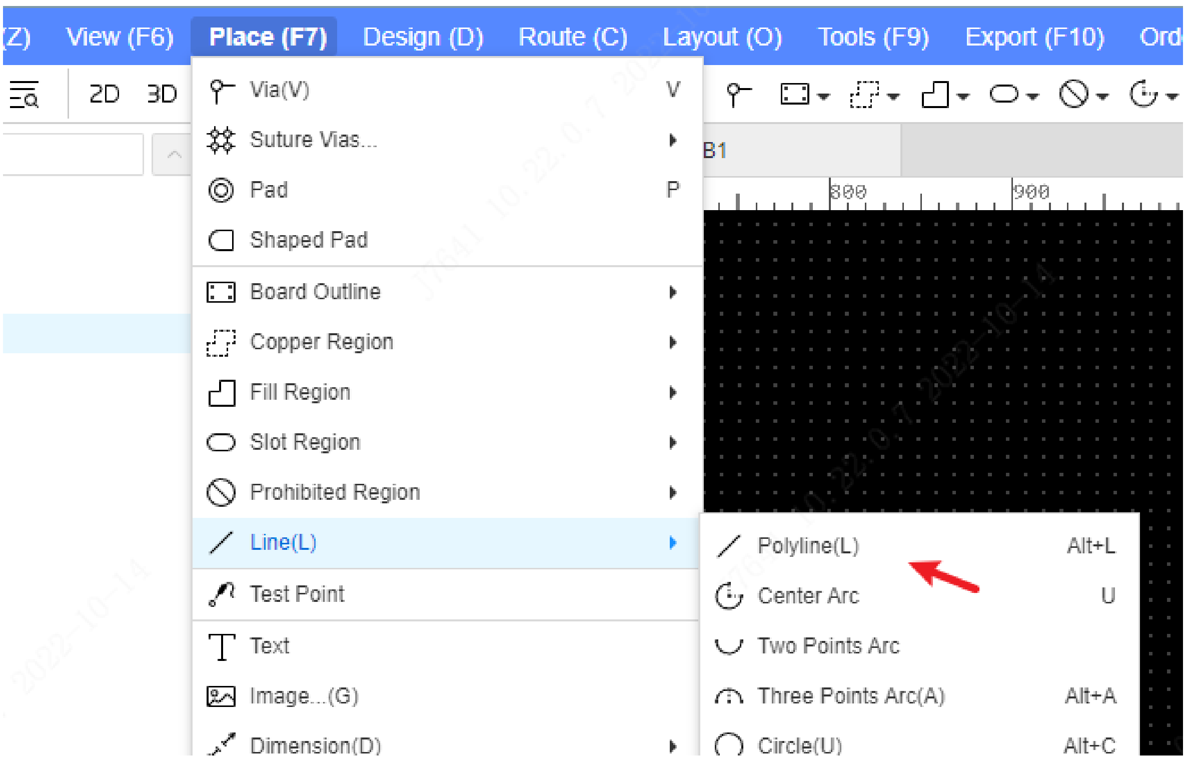
Line

The line tool is different from the wire in that it has no electrical properties and can only be used as a drawing tool

Polyline

Polylines are different from wires. Polylines do not have any electrical properties. Polylines can be used to draw board frames. Segmentation planning cannot be used for wires.

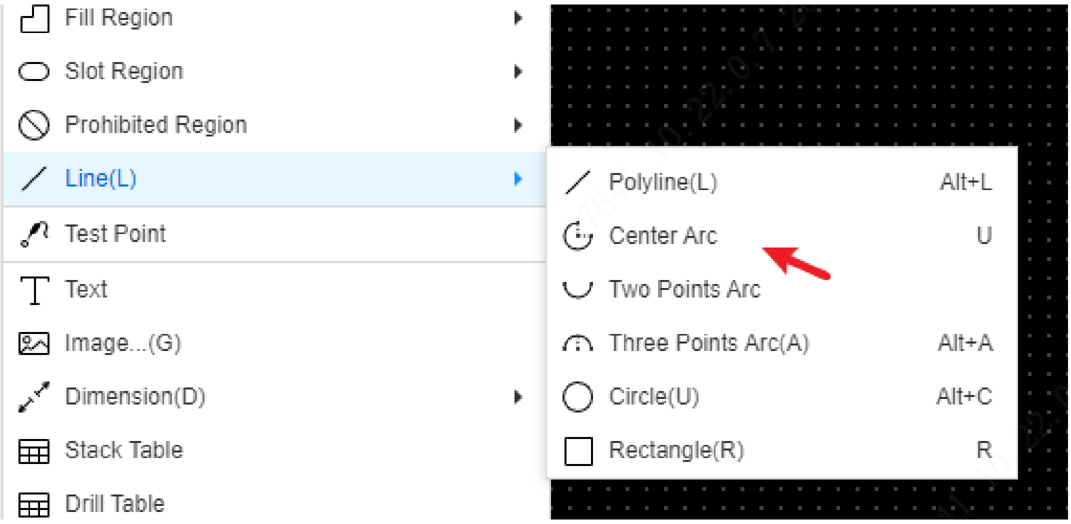
- Top Menu - Place - Line - Polyline
- Use shortcut keys `Alt` + `L`



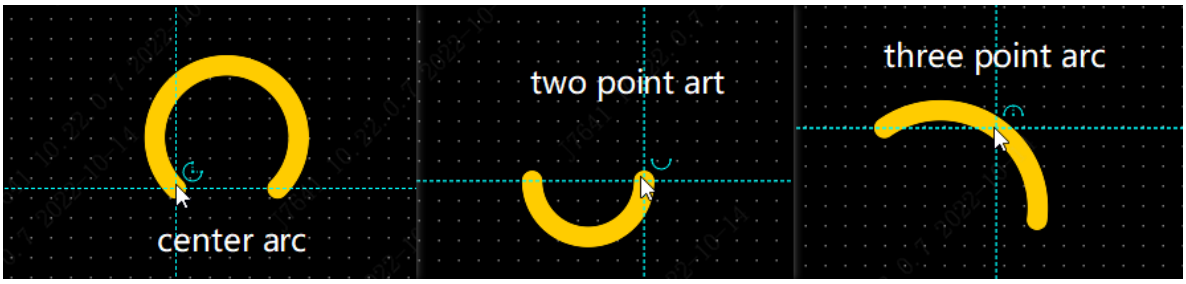
Arc

Used to draw an arc based on the center point in the PCB interface

- Top Menu - Place - Line - Center Arc

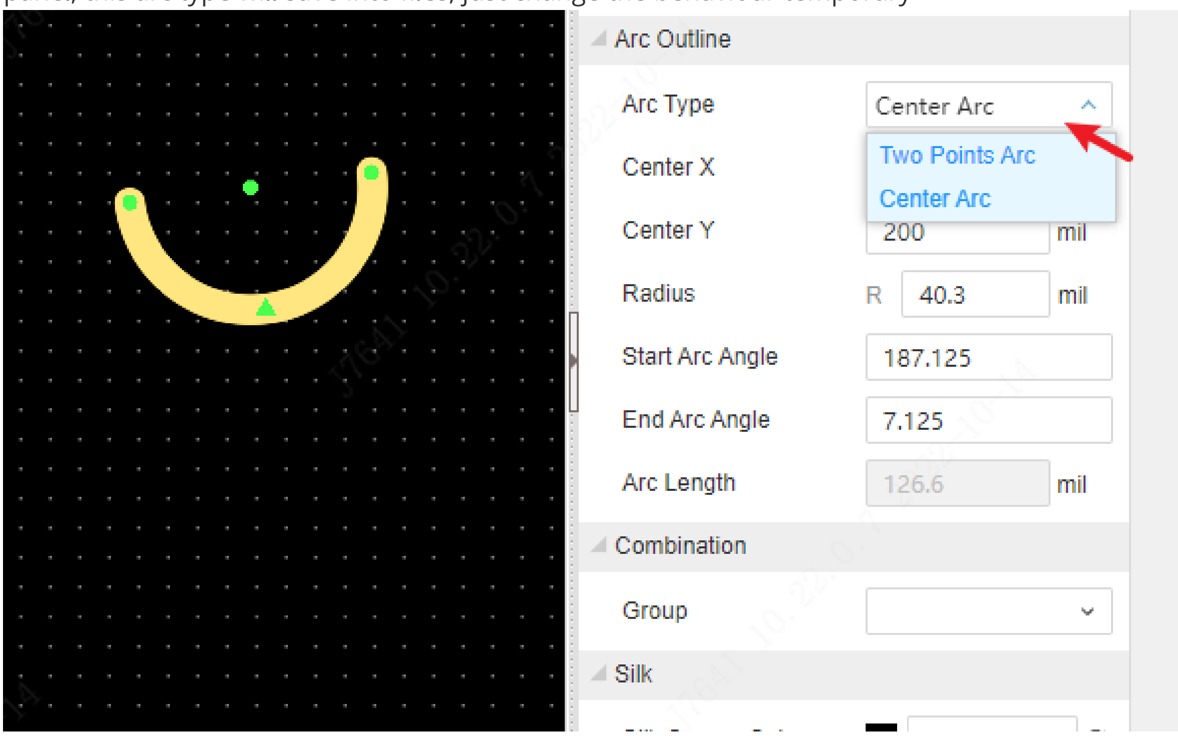


Center arc, two point arc, three point arc they are just different draw behaviour, not the different object type.



Center Arc: set the arc center first, then move the cursor set the radius, and set the start point and end point
Two point arc: Click the start point and then set the end point, it only 180 degrees
Three point arc: set the start point and then set the end point, and then set the radius

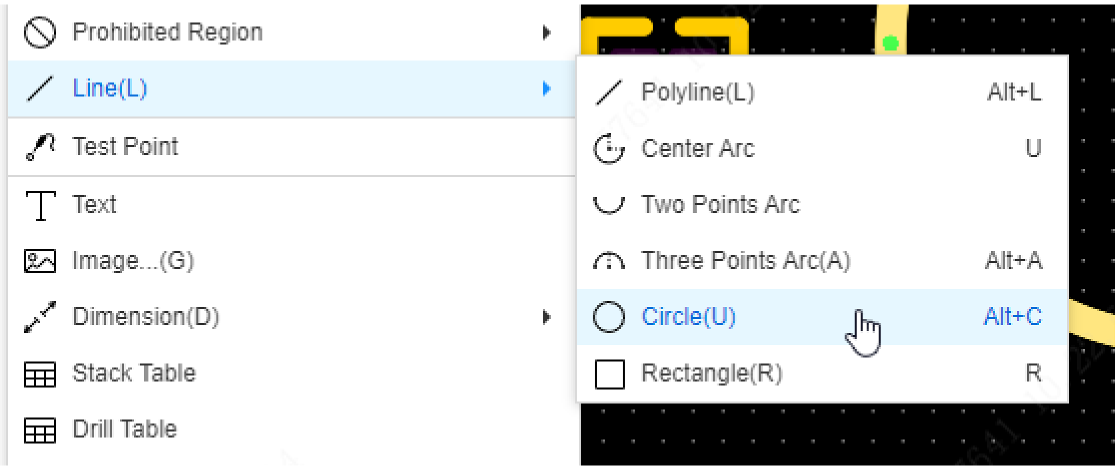
when you want to change the Arc adjust behaviour, you can change the Arc Type at property panel, this arc type will save into files, just change the behaviour temporary



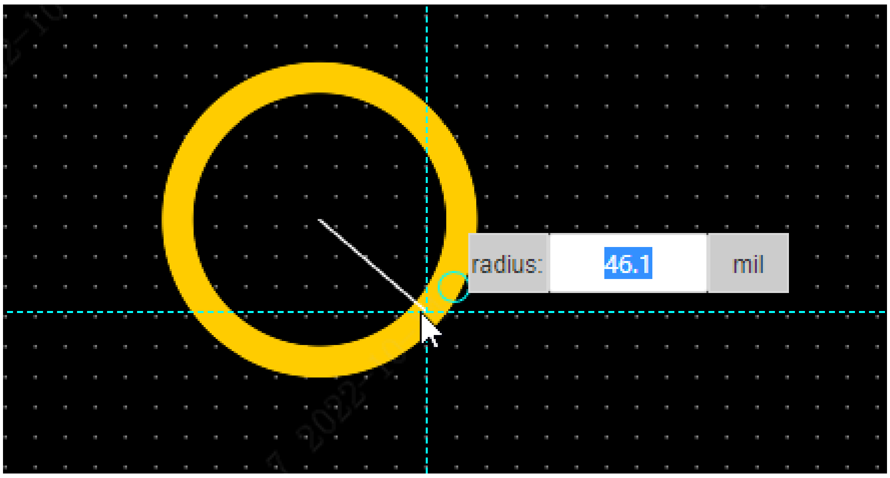
round

Used to draw a circle in the PCB interface.

- Top Menu - Place - Line - Circle



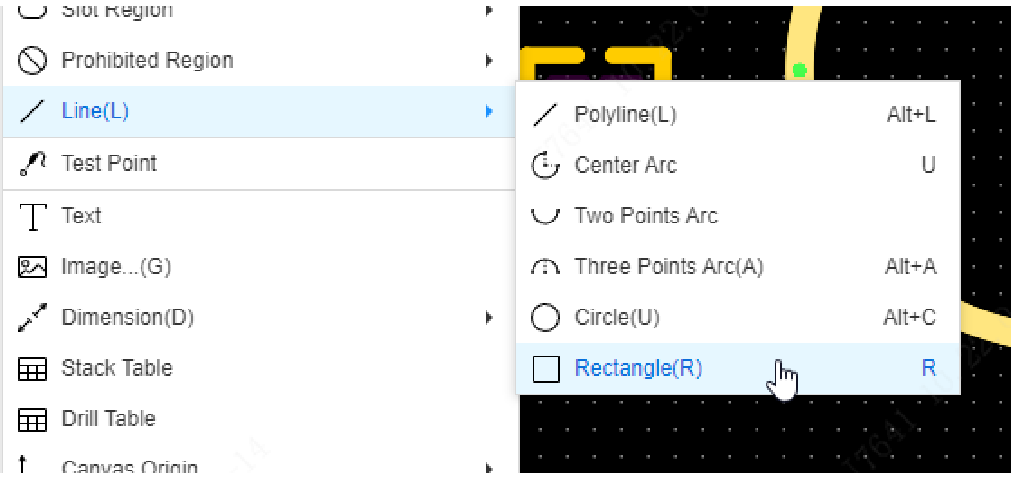
- Use shortcut keys `Alt` + `C`



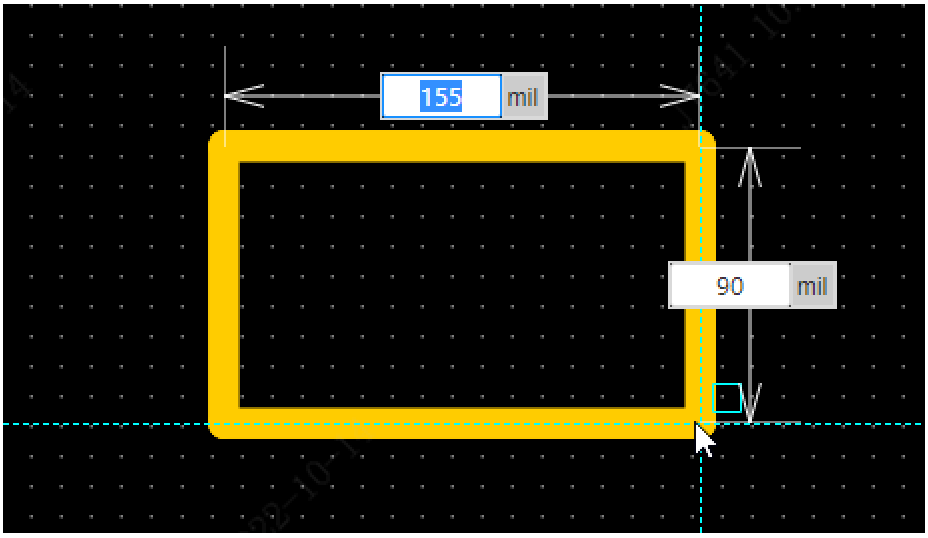
rectangle

It is used to draw a rectangle in the PCB interface. After drawing the figure, you can click the figure to modify the corresponding properties in the right panel.

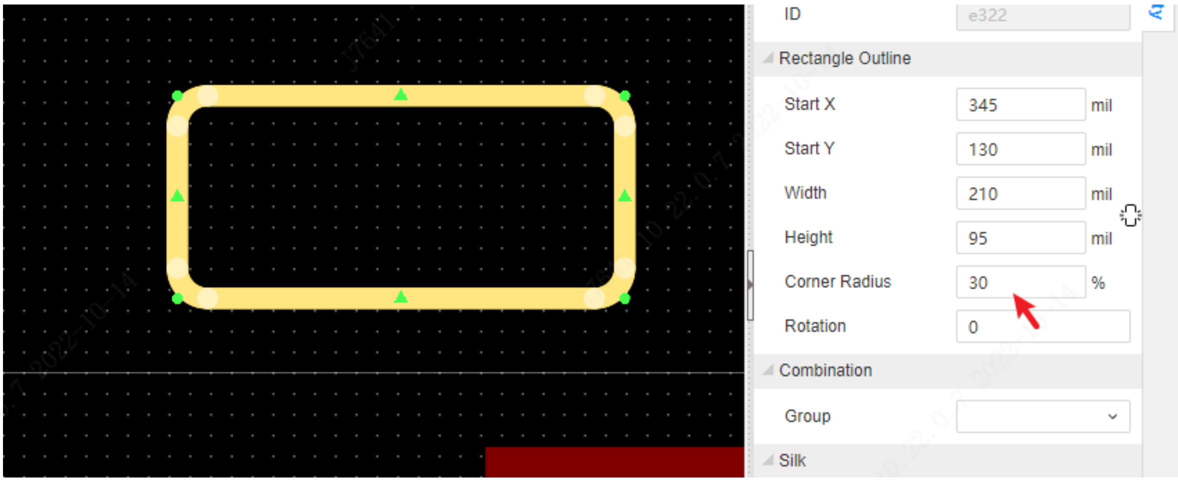
- Top Menu - Place - Line - rectangle



- Use shortcut keys `Alt` + `R`



If you want to set a round rectangle, you can set the corner radius for it

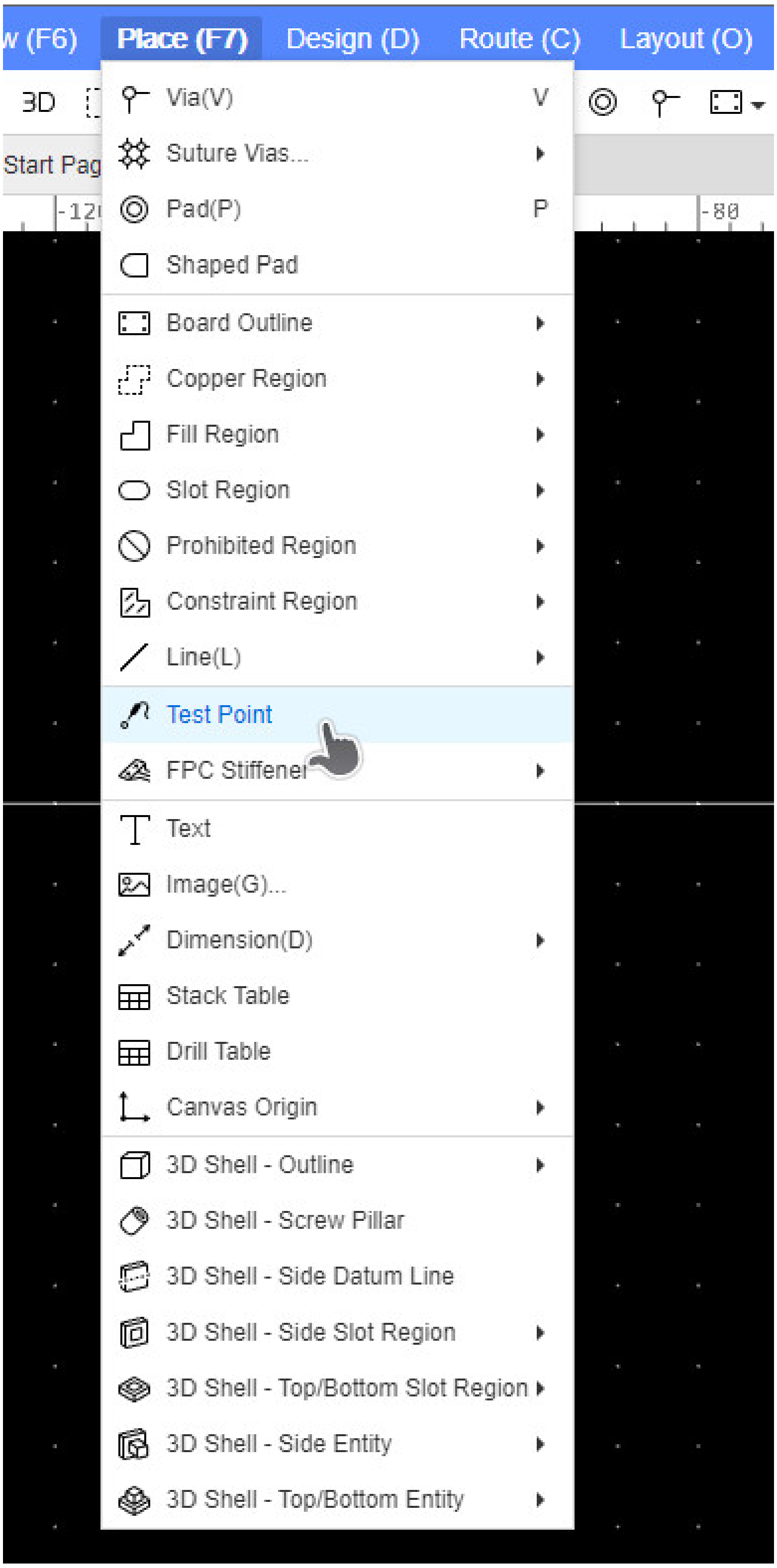


Test Point

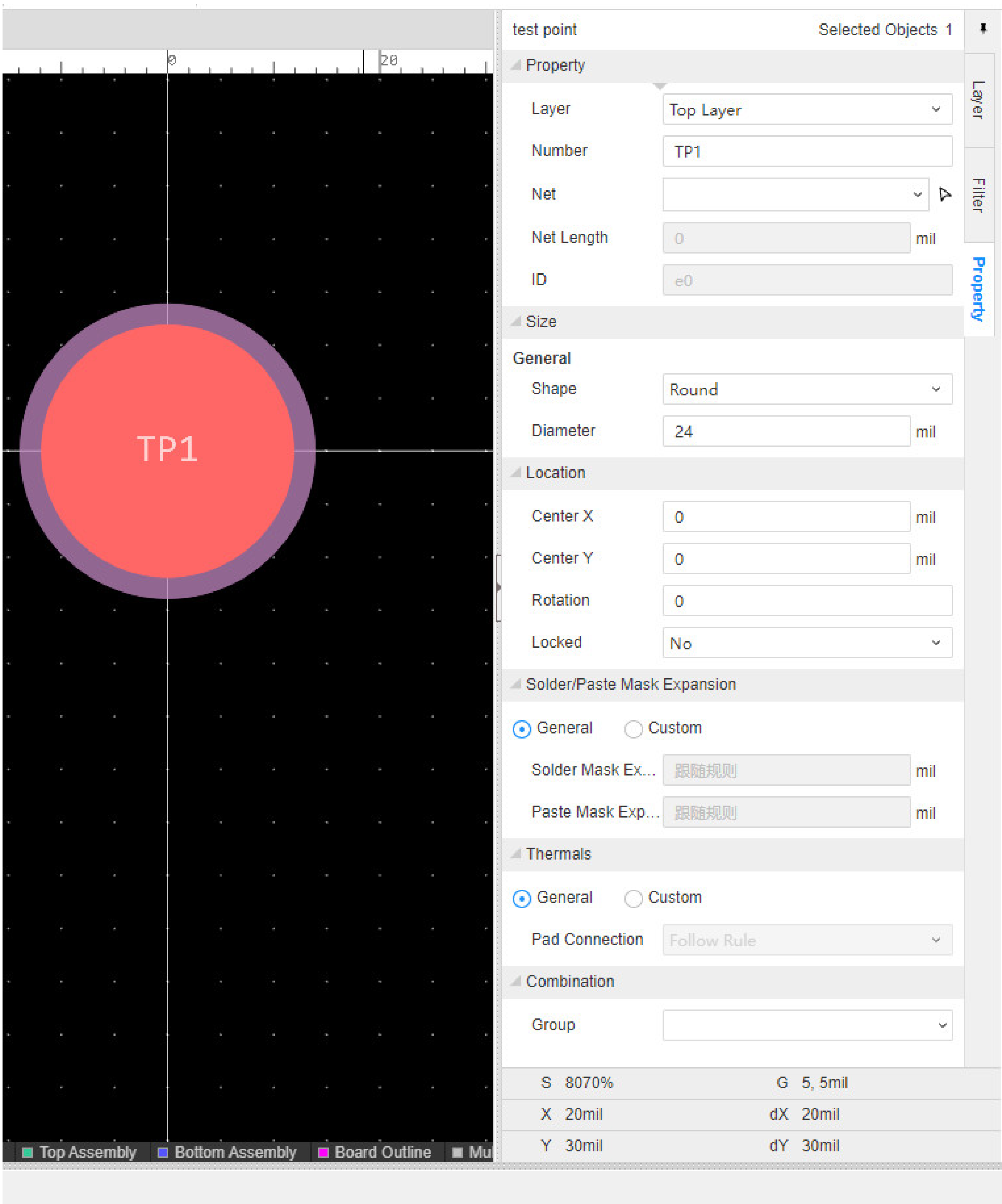
EDA Pro supports placing test points directly on the PCB for connecting test instruments for circuit testing and fault diagnosis. Test points are usually located at critical locations on the circuit board, such as connectors, chip pins, critical signal lines, etc.

Place method:

- Top Menu - Place - Test Point,



After placement, you can set parameters such as layer, network, size and shape of the test point in the right property panel

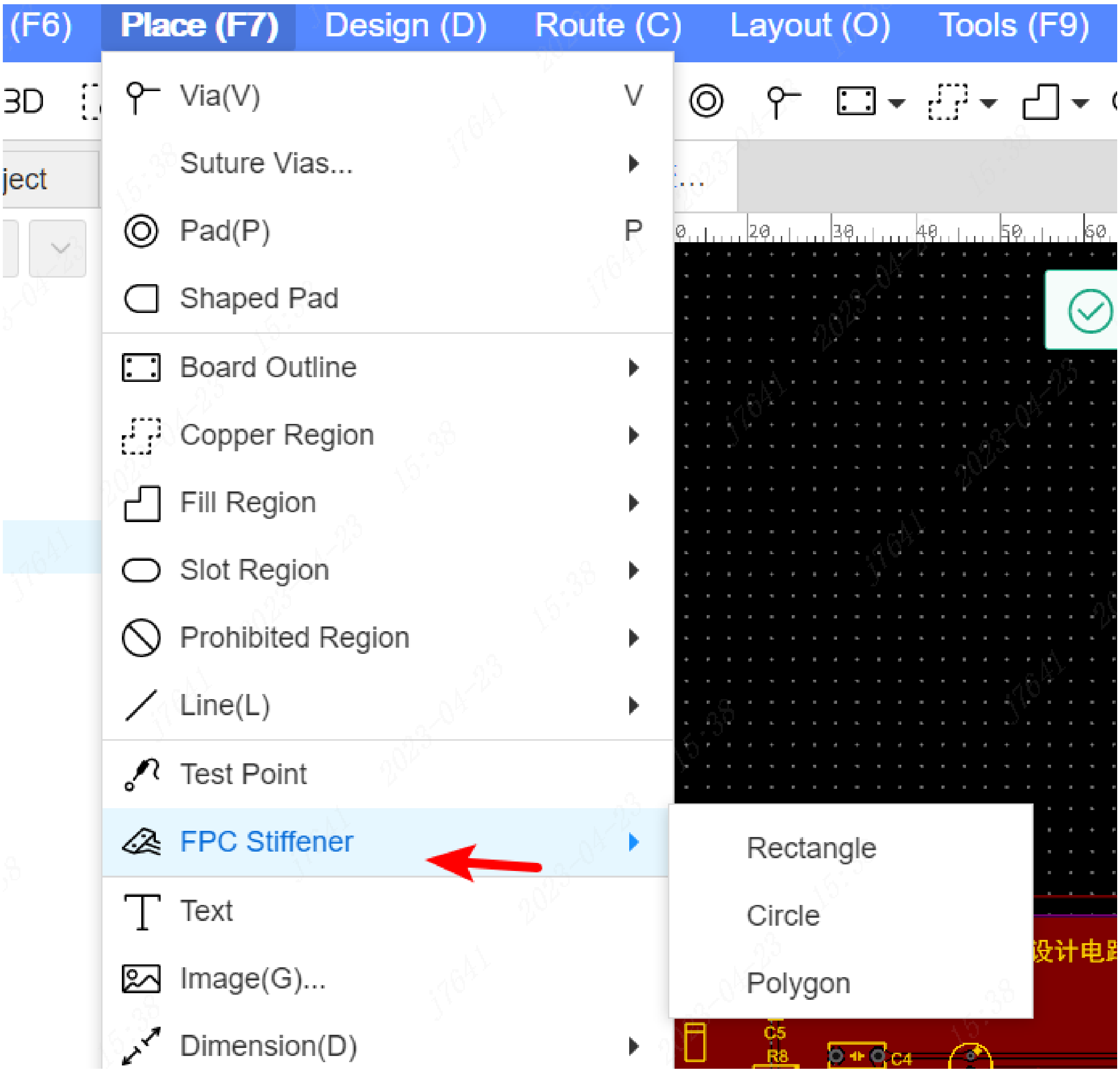


FPC Stiffener

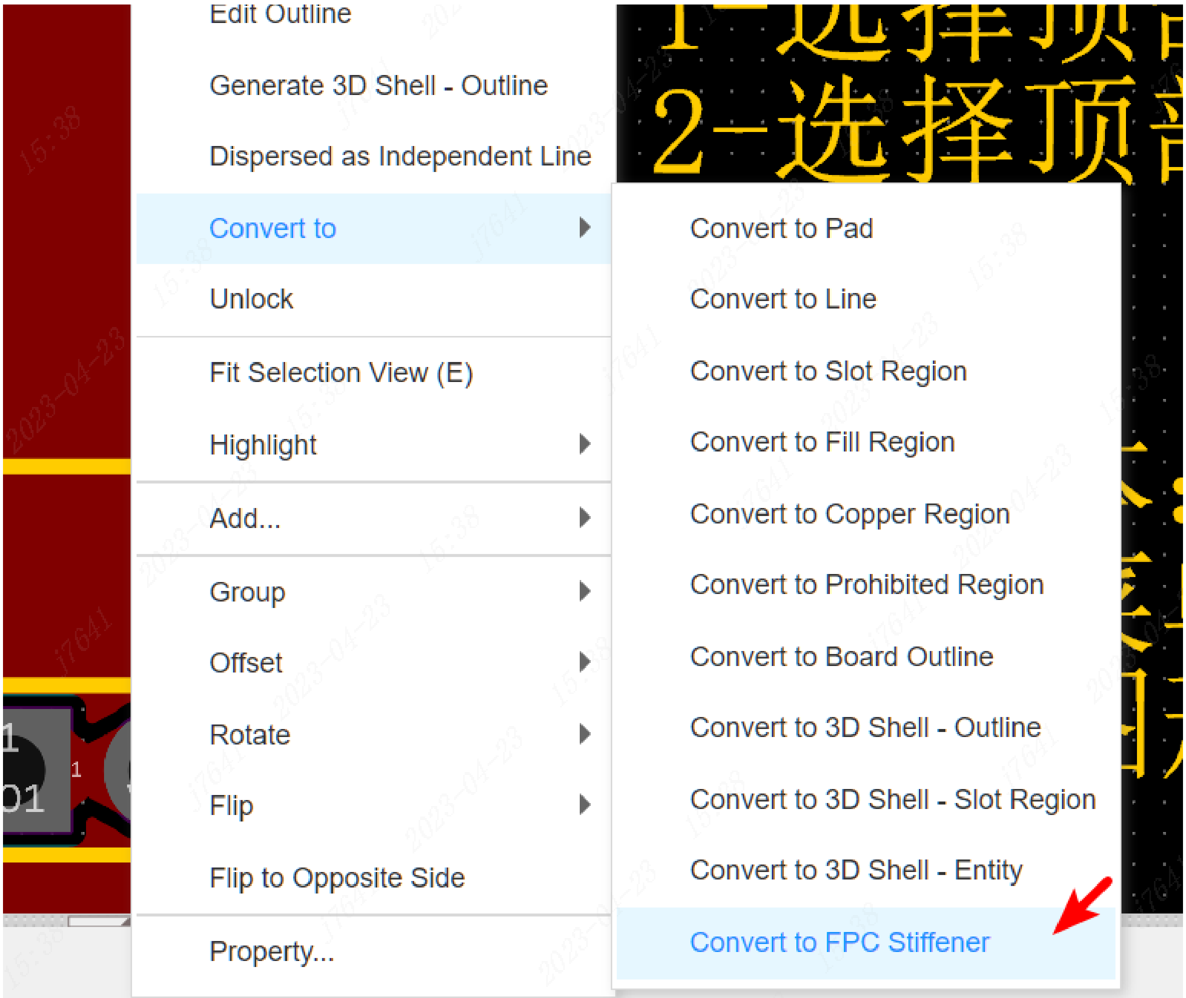
EasyEDA Pro began to support the drawing of FPC stiffener boards in v2.0. Combined with JLPCB PCB order system, it can easily access PFC design files, reduce the process of manual mutual confirmation, reduce errors, and reduce time costs.

FPC stiffener board drawing steps

1. First, draw according to the usual PCB, and draw the board frame and other graphics elements
2. Place the FPC reinforcement board graphic element at the FPC cable interface. The interface is used to insert into the FPC segment, so it is necessary to draw a reinforcement board to increase the strength of the card FPC cable

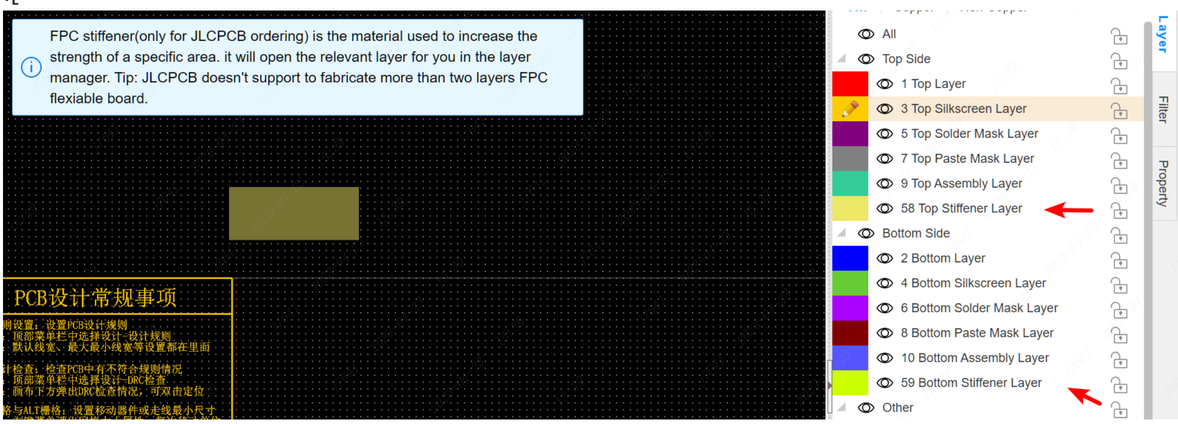


After importing DXF, it is also supported to convert to reinforcement board through the right-click menu

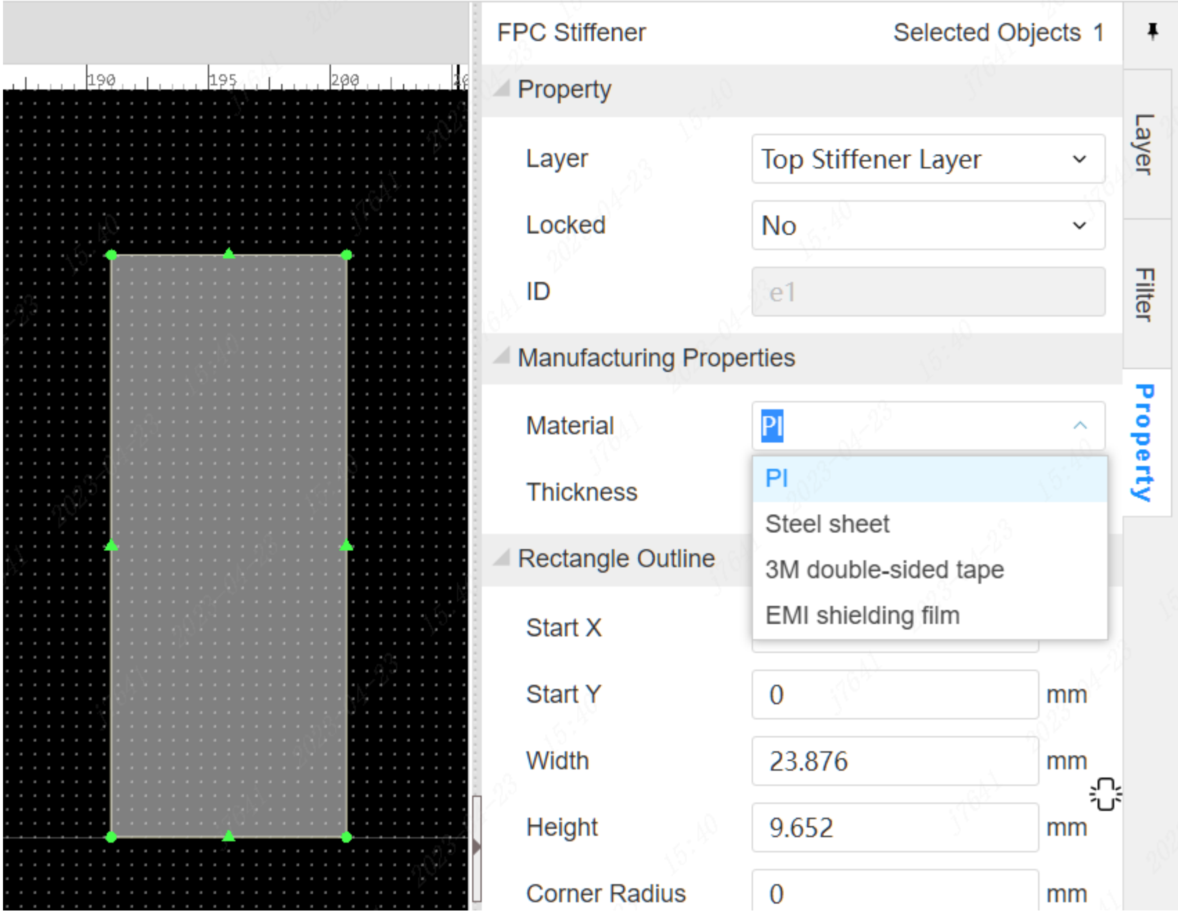


3. PCB will automatically switch the PCB type to FPC soft board for you in the layer manager, and open the two layers dedicated to reinforcement.

![



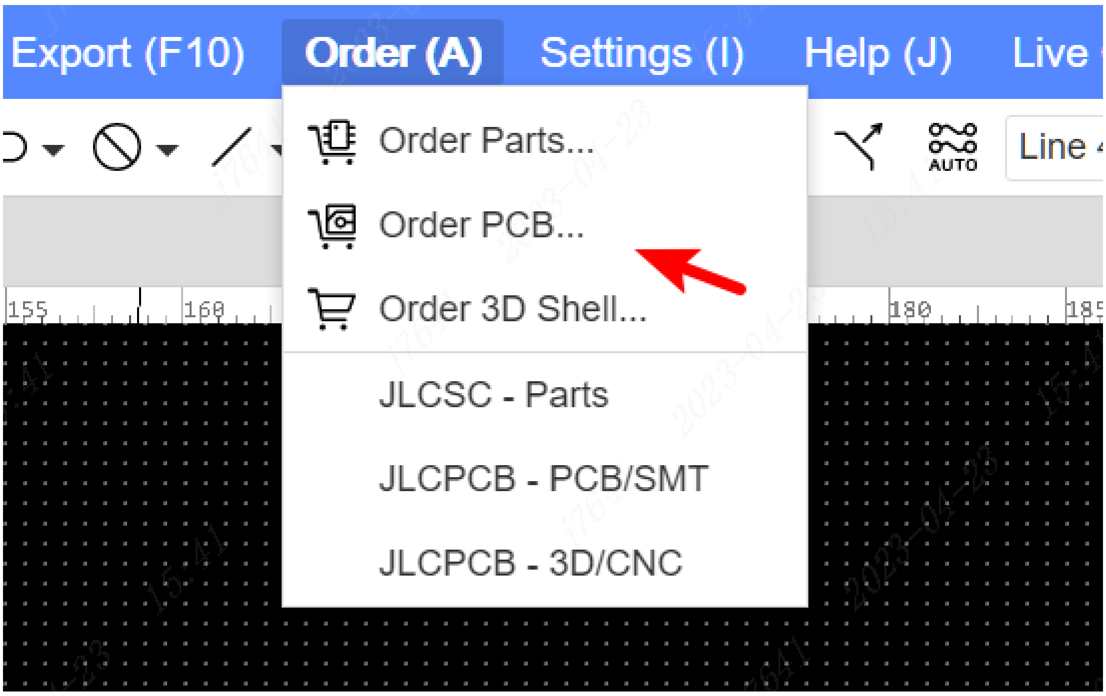
4. After placing, select the reinforcing plate, and set its material (PI, steel sheet, 3M double-sided adhesive, electromagnetic shielding film), thickness/model in the property panel, so that the engineer can according to your setting after placing the order Options are automatically priced and produced.



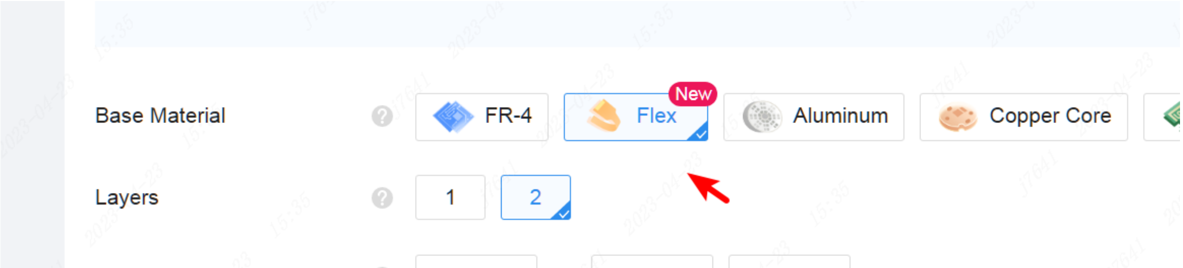
5. After the setting is completed, the drawing of the FPC reinforcement board can be completed.

FPC soft board order

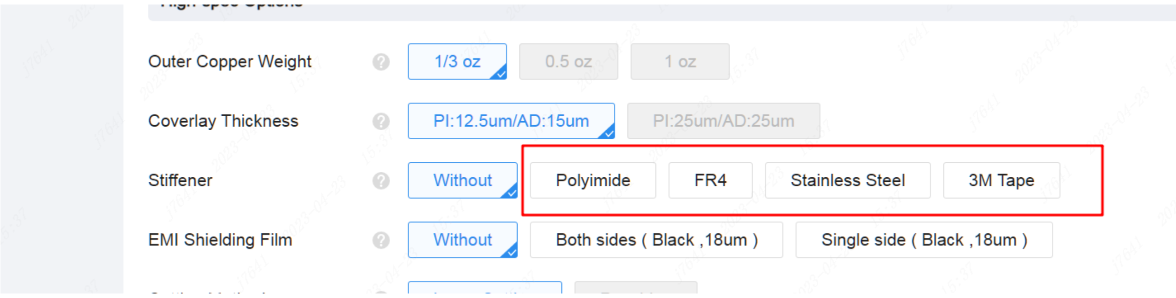
1. After drawing the FPC reinforcement board, export Gerber or place an order with one click



2. On the order page, select FPC soft board(flex) for the board type



3. The reinforcement method is selected according to the attributes selected during design



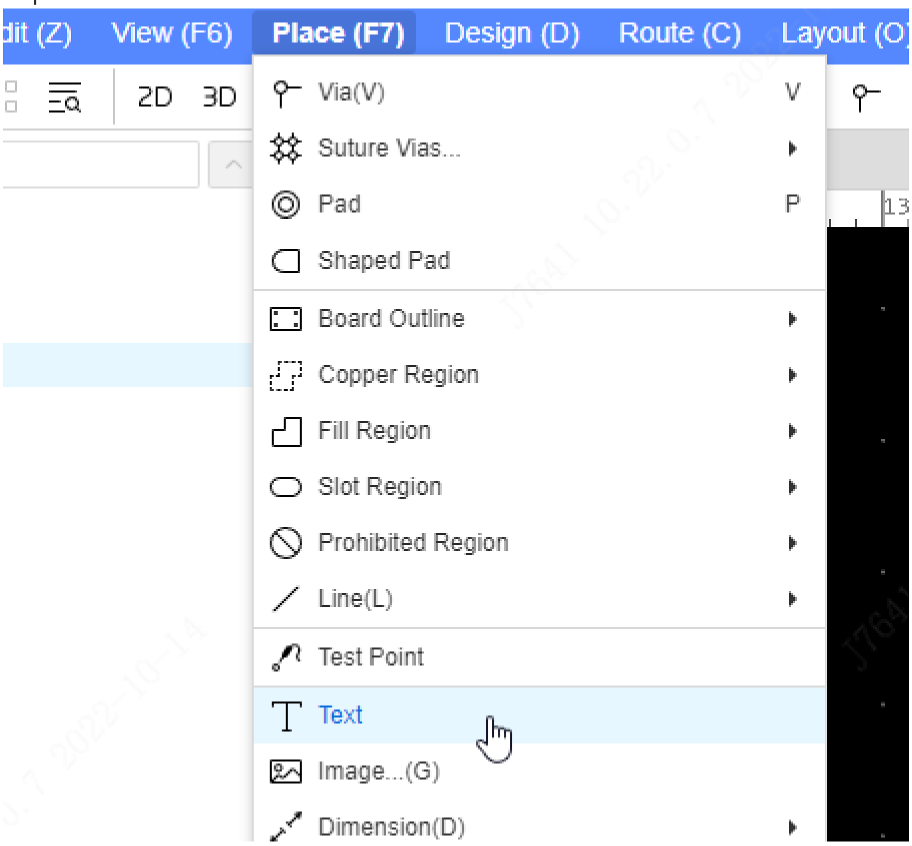
4. Submit the order after filling in other parameters.

Text

Placed in the PCB for instructions or identification.

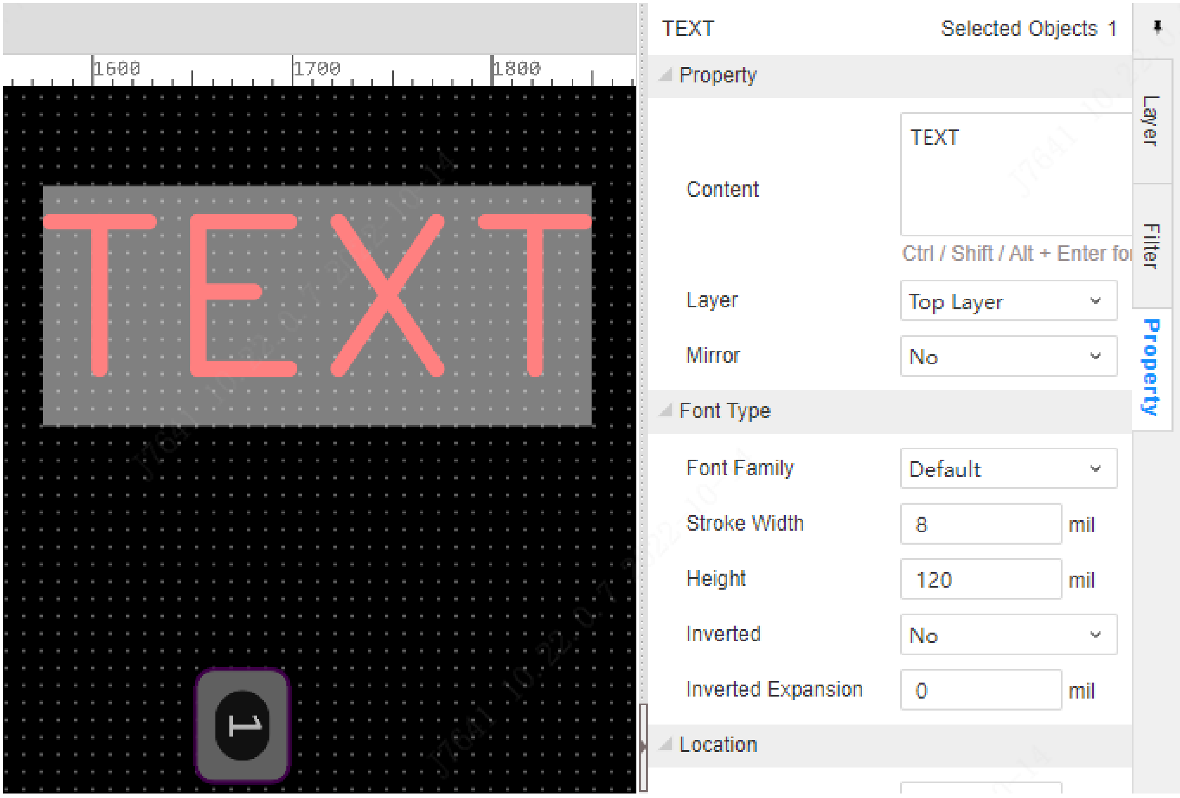
Place method:

- Top Menu - Place - Text

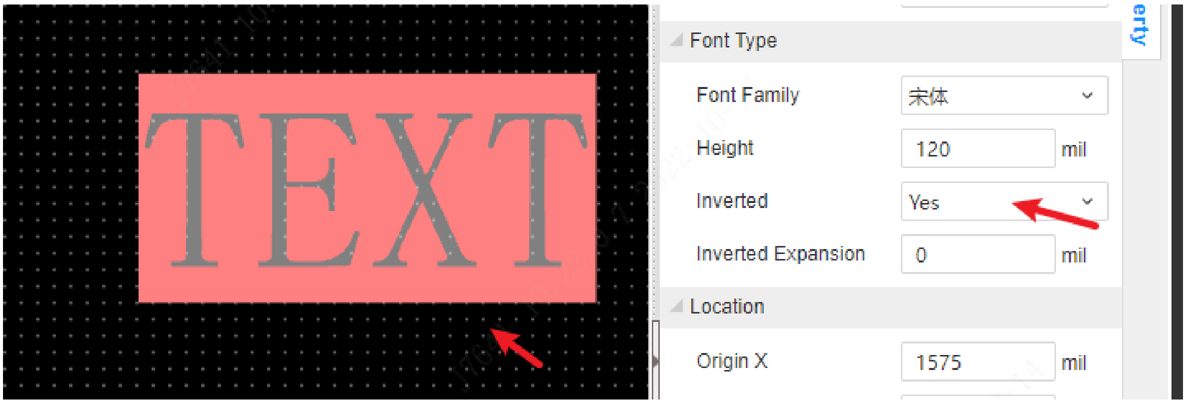


Enter the text content to be placed in the text box,After setting the font attribute,click OK to generate a preview, and click again to place it in the schematic.

Click the mouse to put it into the PCB, and select it to modify the properties of the text in the right panel.



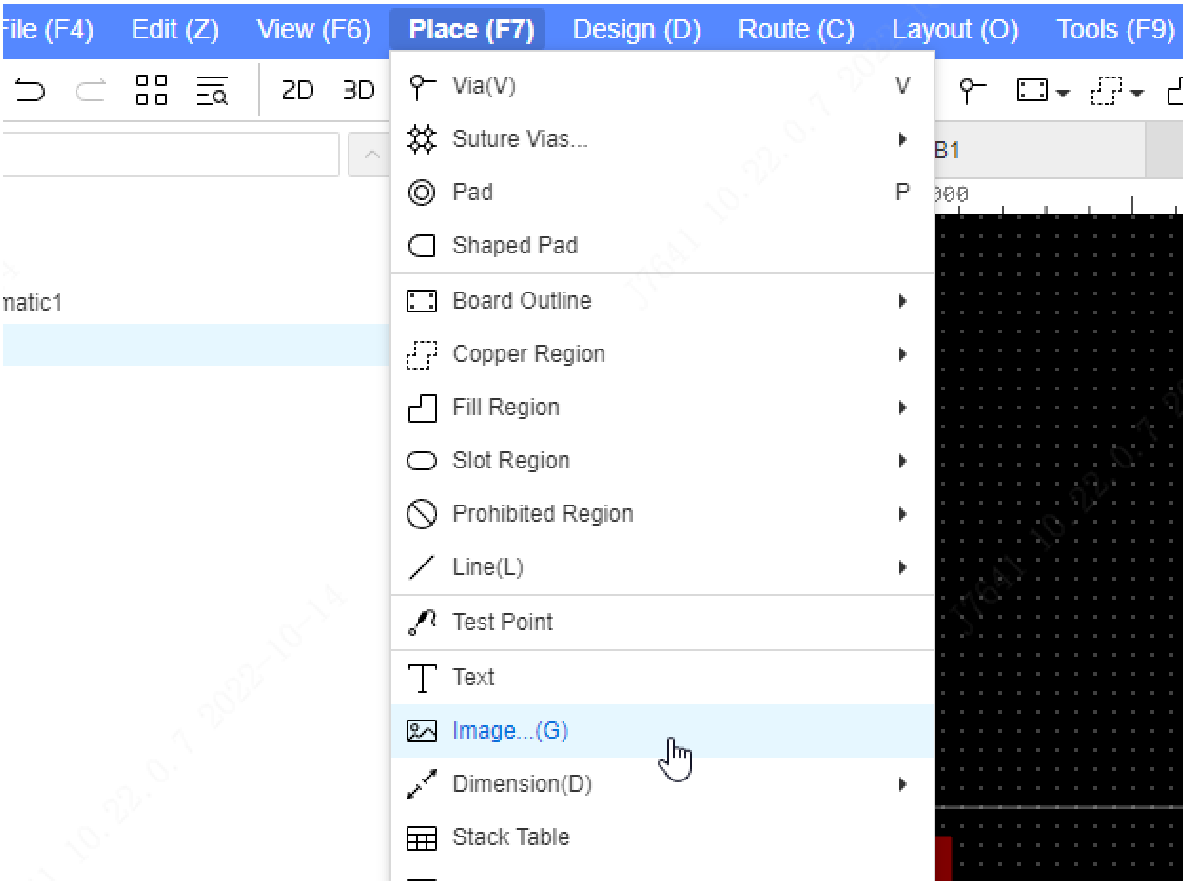
If you want inverted the text background, you can set the inverted as Yes.



Image

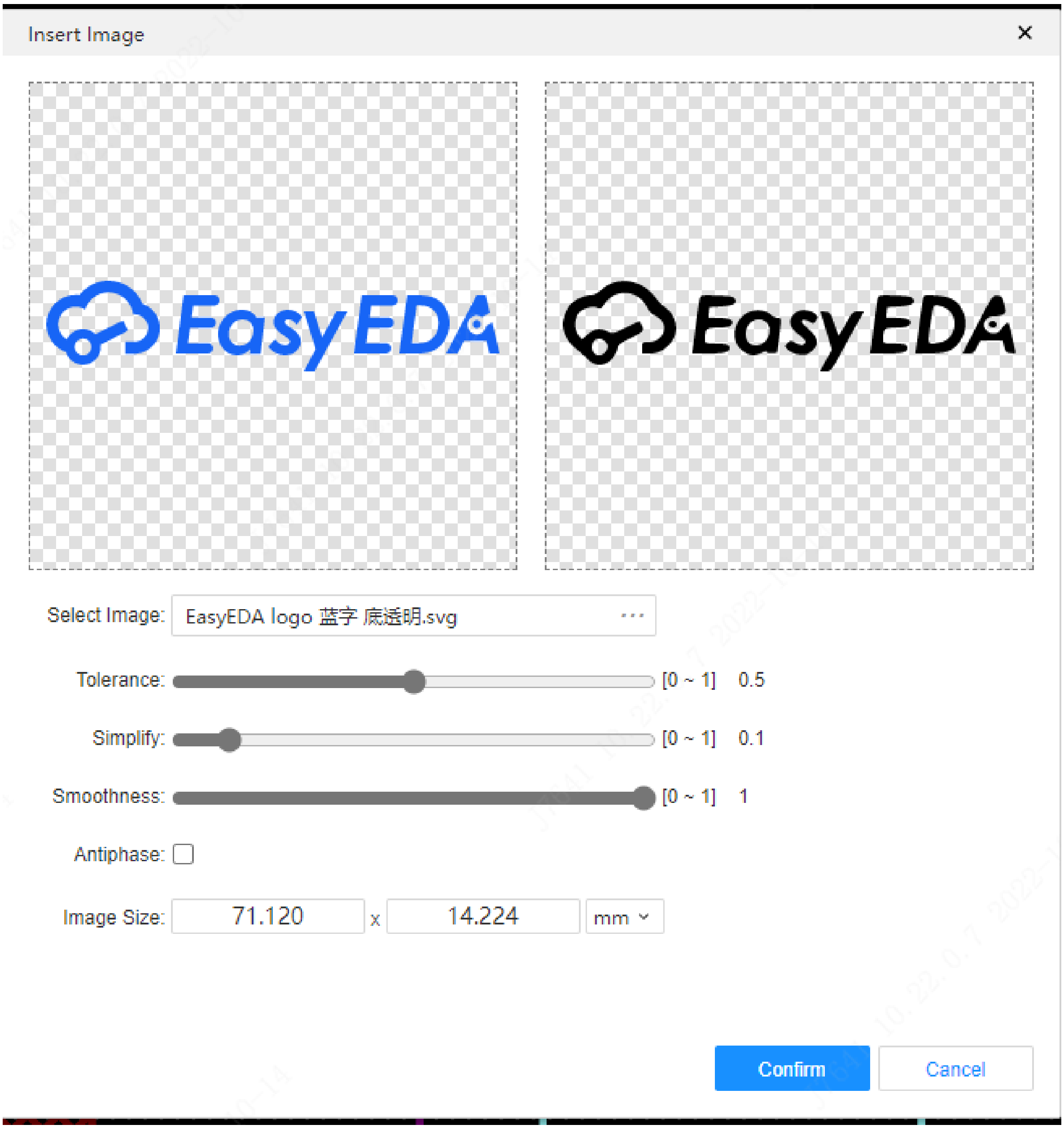
Under the PCB and PCB library canvas, EasyEDA Pro supports adding images. You can easily insert logo patterns such as silkscreen logos.

Entry: Top Menu - Place - Image



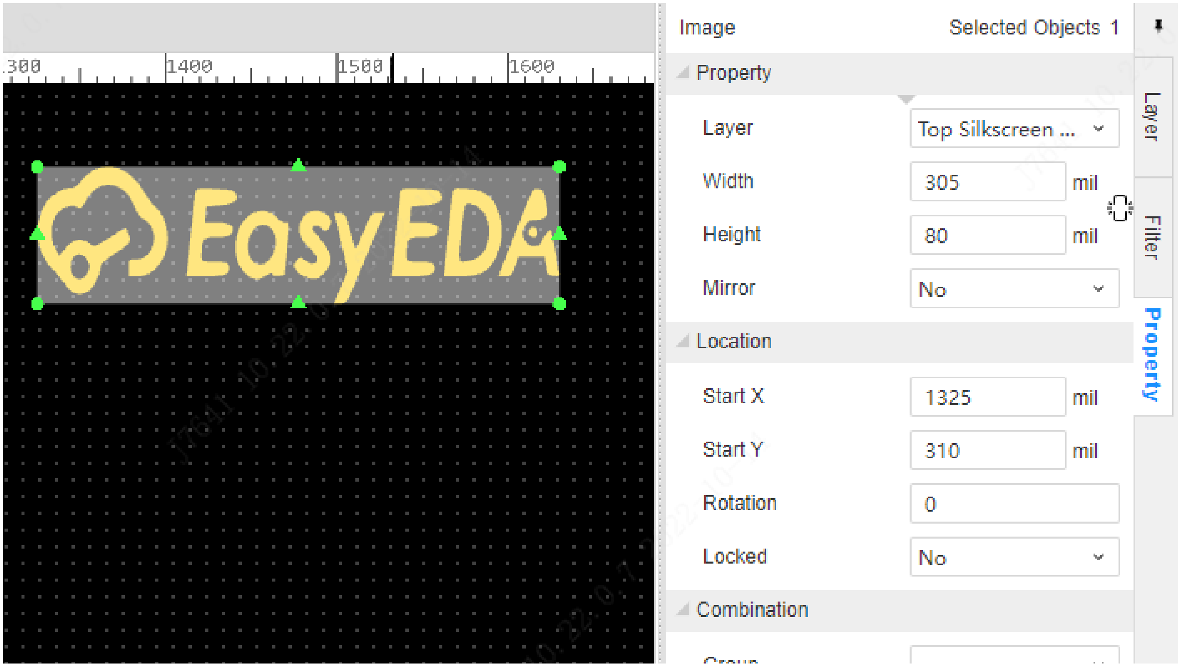
or Top Menu - Import - Image

Click the insert picture function, a window will open, you can add the picture you need, the supported picture formats are: SVGZ, SVG, PNG, PIP, JPG, PJPEG, JPEG, JFIF format.



- **Tolerance**: The larger the value, the larger the image loss
- **Simplified**: the larger the value, the more rounded the edges of the image will be
- **Smooth**: The larger the value, the smoother the imported image, and it is more obvious to turn on the quality priority
- **Inverted**: After selecting, the originally highlighted area will be excavated
- **Picture size**: Set the size you want to insert, modifying the single value will scale proportionally
- **Unit**: The system only supports two units, mm and mil

The picture will be placed in the currently edited layer, click the inserted picture, and modify the layer, width, height and mirror image in the right panel.

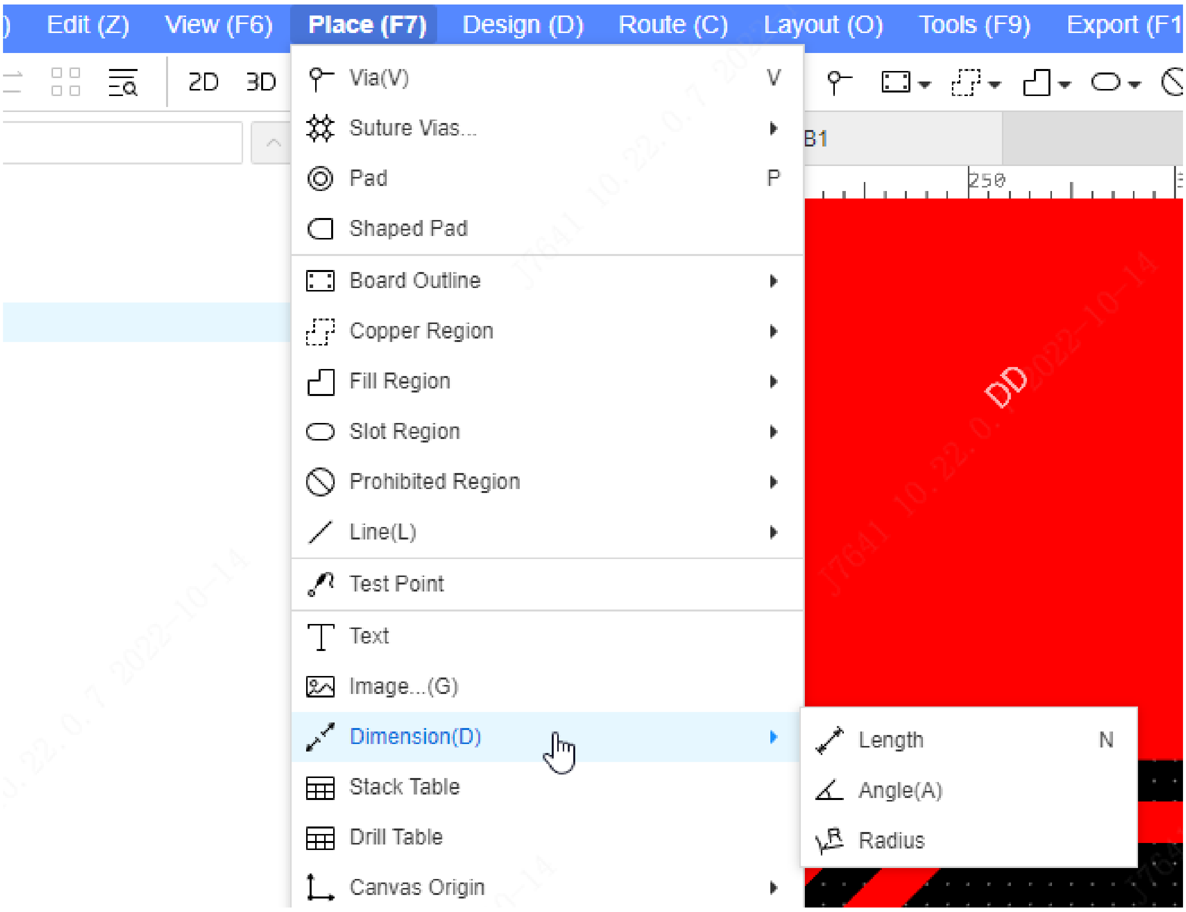


Dimension

Measure length, radius of circle, and angle. Different from the function of measuring distance, dimension Place is to mark the length and width of the PCB board.

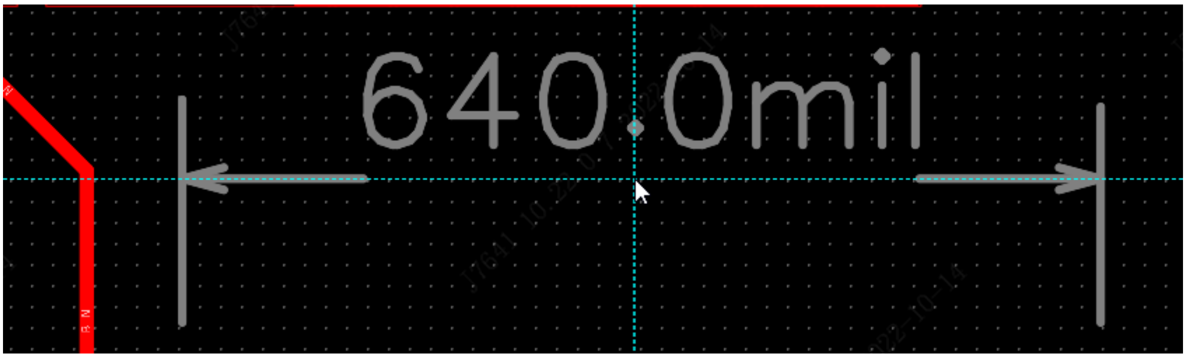
Place Dimension method:

Top Menu - Place - Dimension

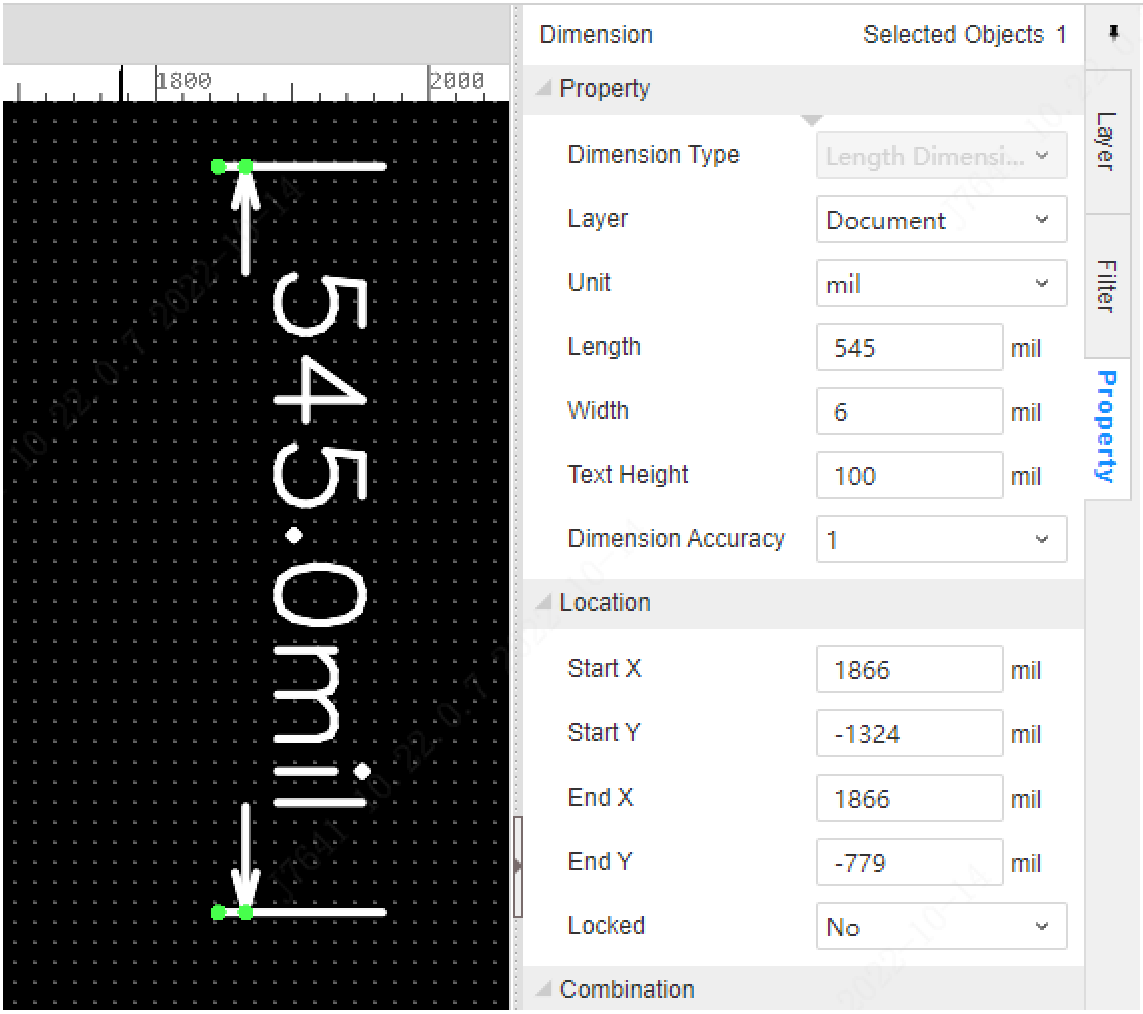


length Dimension

- Top Menu - Place - Dimension - Length



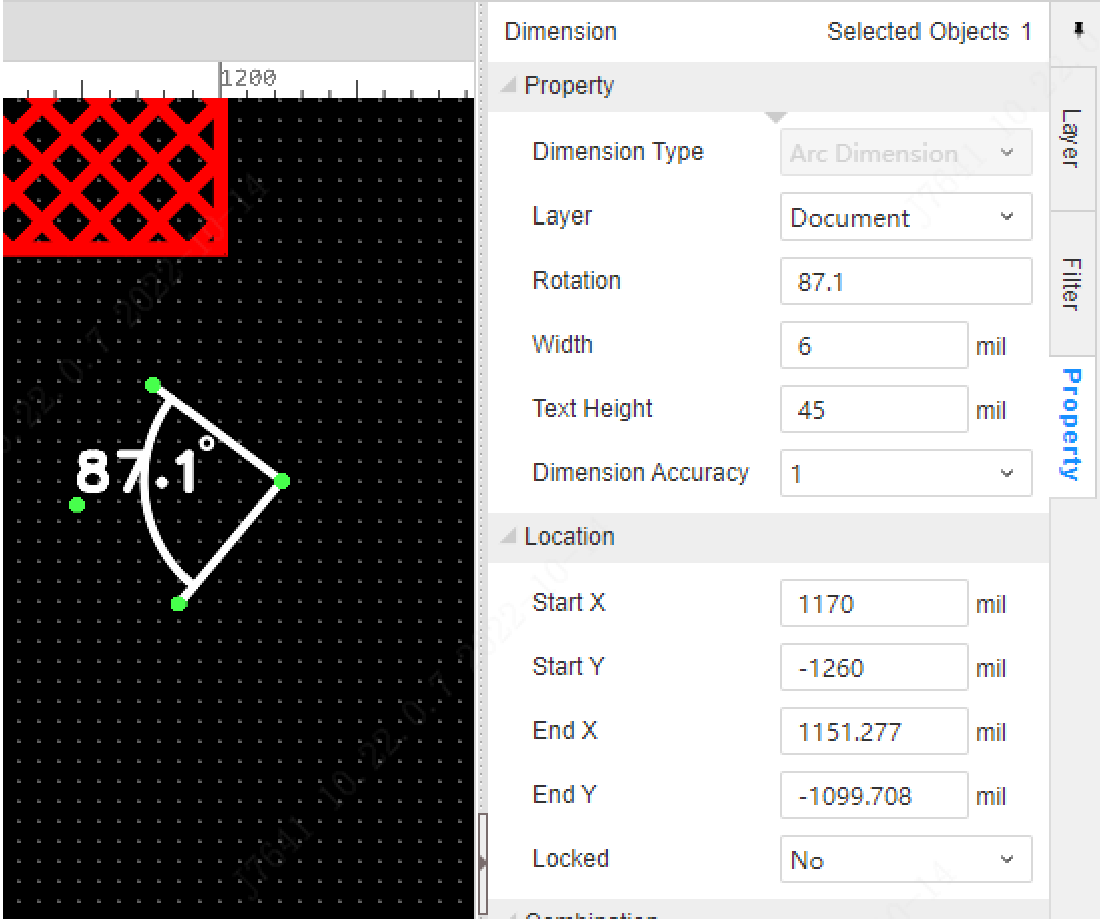
Click the placed dimension to modify the information in the right view.



- **Dimension Type**: Default length Dimension, cannot be modified
- **Layer**: Modify the dimensioned layer
- **Unit**: Four units of mm, cm, inch, mil can be modified
- **Length**: Modify the Place length of the dimension
- **Width**: Modify the width of the dimension
- **Text Height**: Modify the font heigth of the dimension
- **Dimensional Accuracy**: The precision of the dimension can be modified, up to 4 digits

Angle Dimension

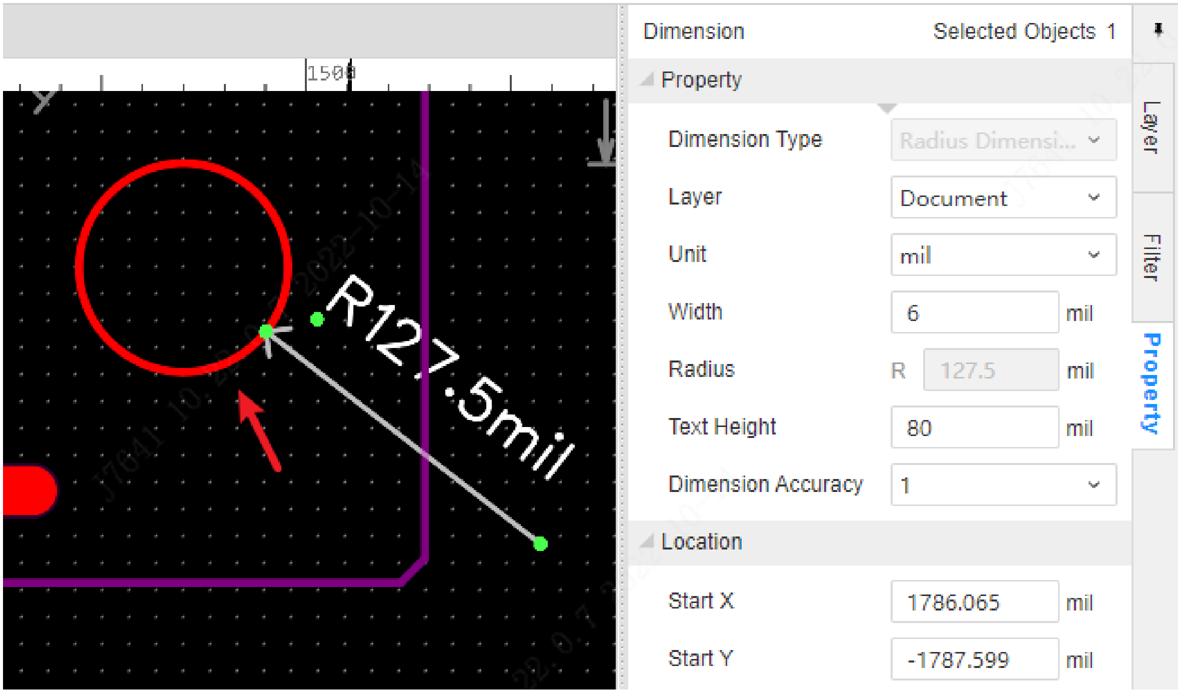
- Top Menu - Place - Dimensions - Angle.



Radius Dimension

- Top Menu - Place - Dimension - Radius,

to measure the radius Dimension of the circle or arc.

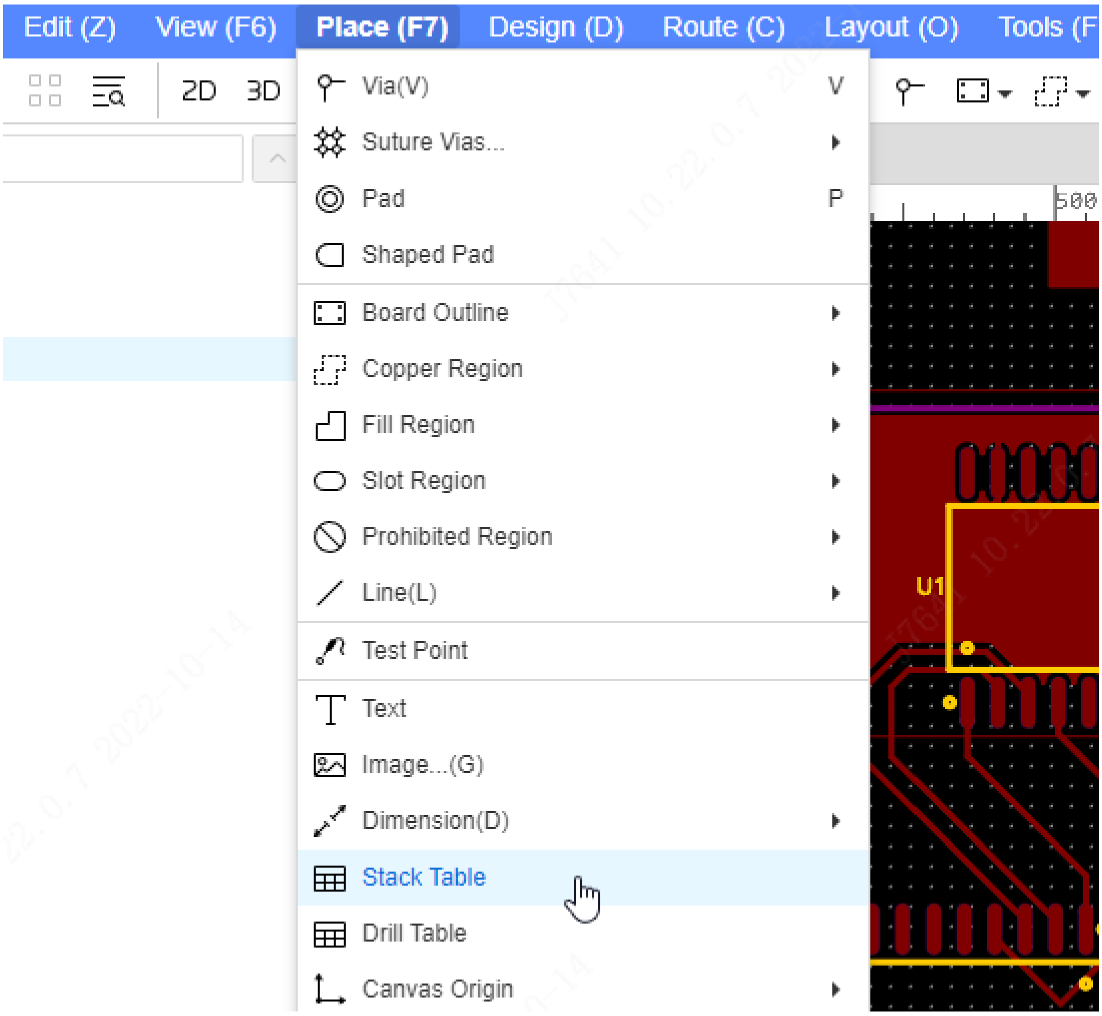


Stacked Table

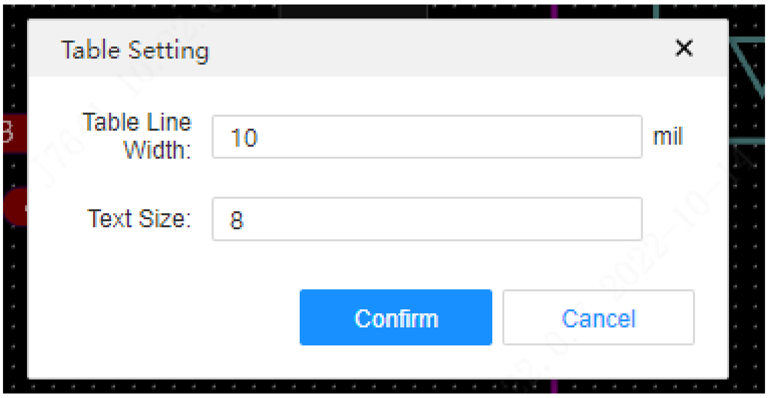
EasyEDA Pro supports a table for placing a stack of PCB layers, showing the current design.

Place method:

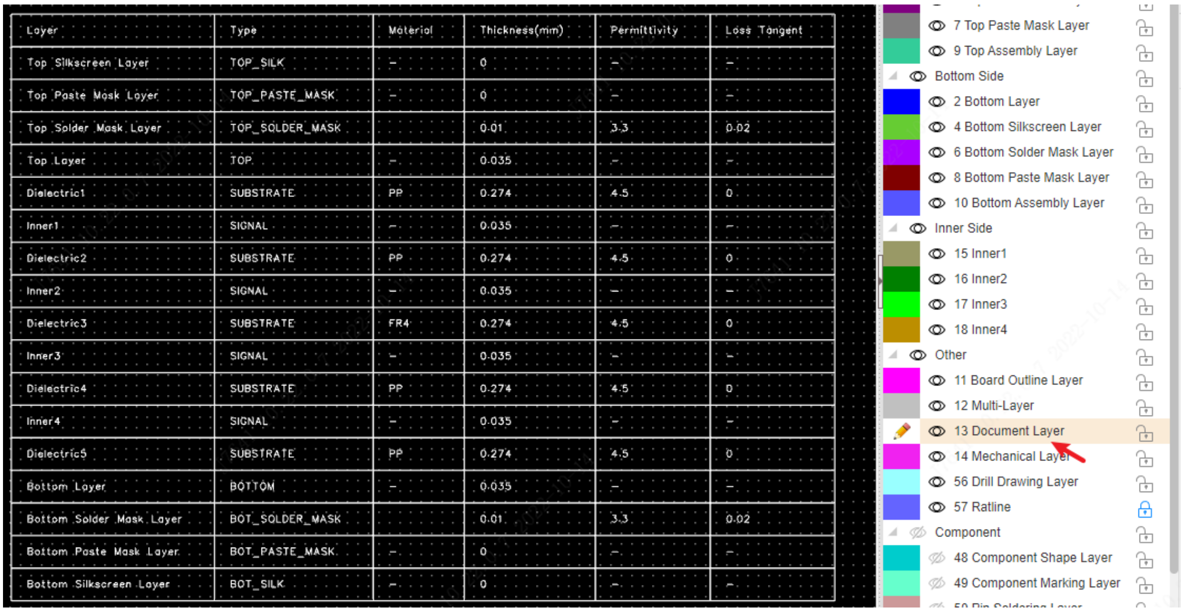
- Top Menu - Place - Stack Table



Set the parameters of the stacked table:



After clicking OK, select the position of the upper left corner of the stack table. The stack table is placed on the document layer by default.



Layer	Type	Material	Thickness(mm)	Permittivity	Loss Tangent
Top Silkscreen Layer	TOP_SILK	–	0	–	–
Top Paste Mask Layer	TOP_PASTE_MASK	–	0	–	–
Top Solder Mask Layer	TOP_SOLDER_MASK		0.01	3.3	0.02
Top Layer	TOP	–	0.035	–	–
Dielectric1	SUBSTRATE	PP	0.274	4.5	0
Inner1	SIGNAL	–	0.035	–	–
Dielectric2	SUBSTRATE	PP	0.274	4.5	0
Inner2	SIGNAL	–	0.035	–	–
Dielectric3	SUBSTRATE	FR4	0.274	4.5	0
Inner3	SIGNAL	–	0.035	–	–
Dielectric4	SUBSTRATE	PP	0.274	4.5	0
Inner4	SIGNAL	–	0.035	–	–
Dielectric5	SUBSTRATE	PP	0.274	4.5	0
Bottom Layer	BOTTOM	–	0.035	–	–
Bottom Solder Mask Layer	BOT_SOLDER_MASK		0.01	3.3	0.02
Bottom Paste Mask Layer	BOT_PASTE_MASK	–	0	–	–
Bottom Silkscreen Layer	BOT_SILK	–	0	–	–

The stacked table after Place is composed of a variety of primitives, and you can dissolve the combination into individual primitives for editing. If the layer manager has adjusted layer parameters, you need to delete and reposition the stacked table, the stacked table will not be updated automatically

Drilling Table

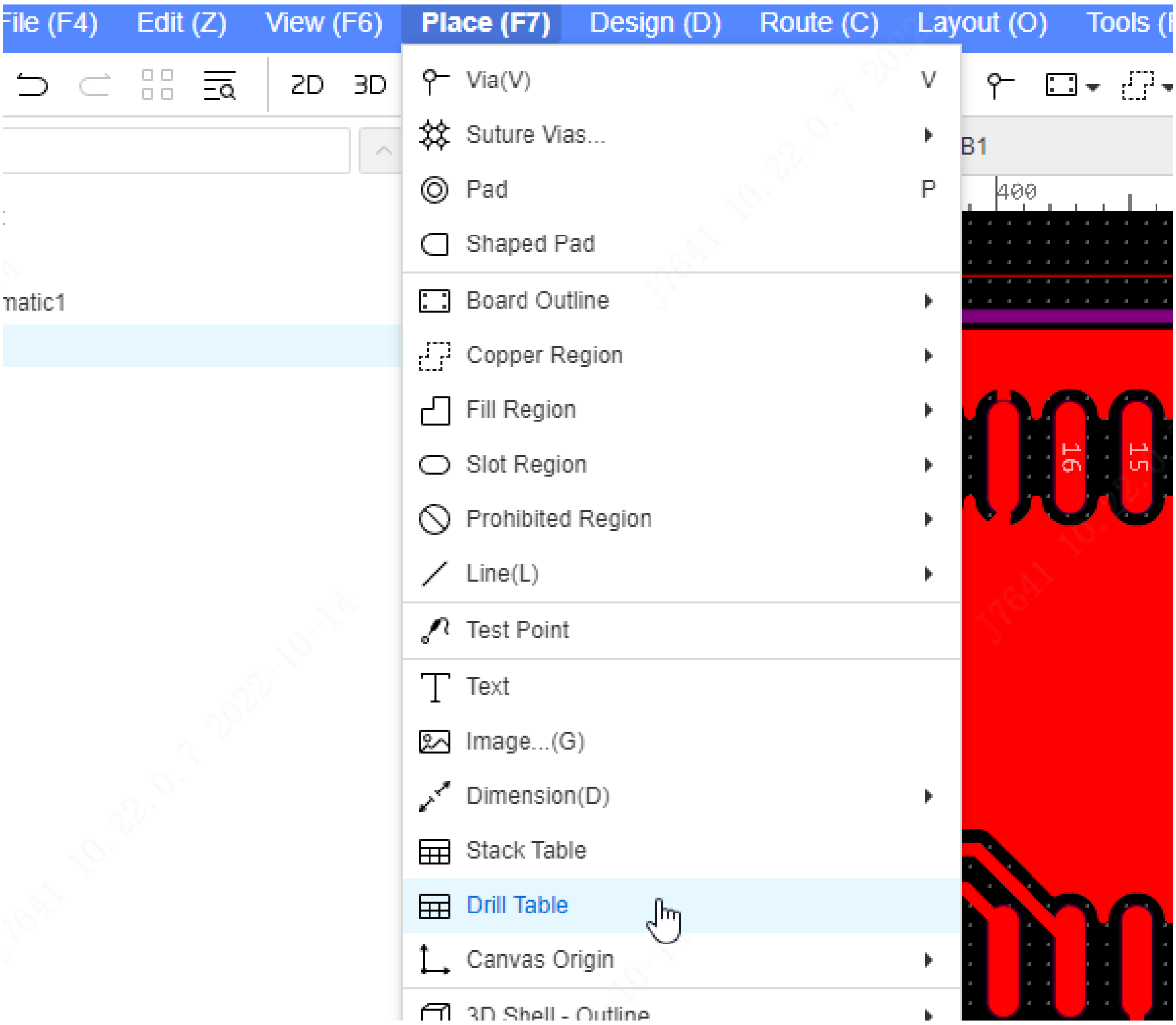
Drilling table

When the PCB design scale is relatively large, and the factory manufacturing process requires, the PCB usually needs to output the drilling table to verify the drilling type and position during PCB manufacturing.

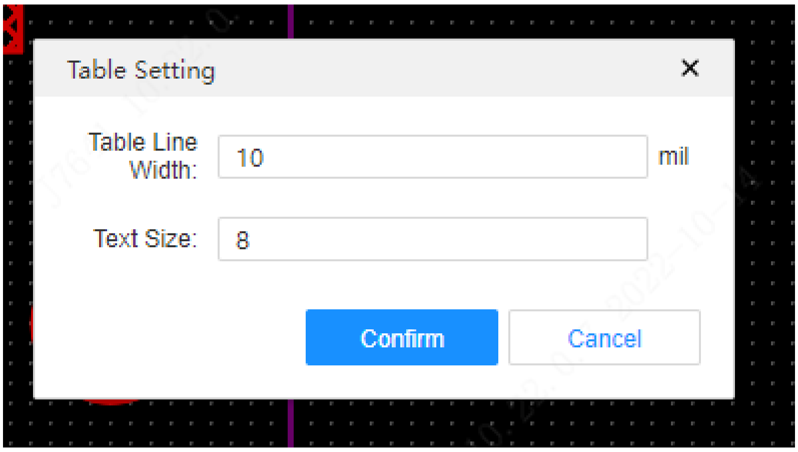
EasyEDA Pro supports placing drill tables.

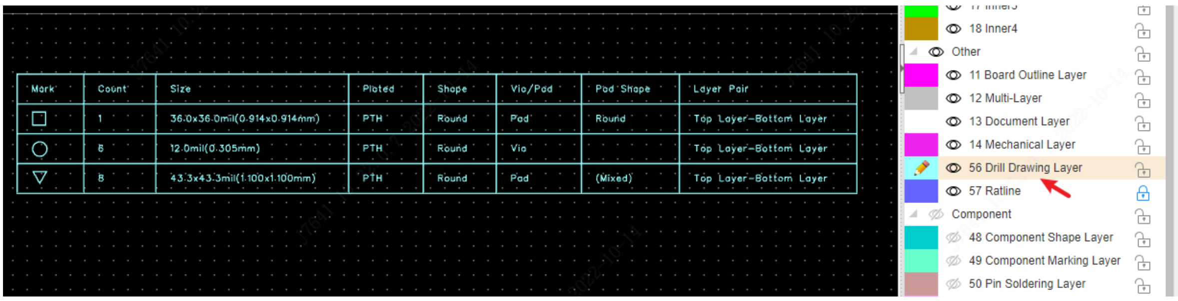
Entrance:

- Top Menu - Place - Drill Table.

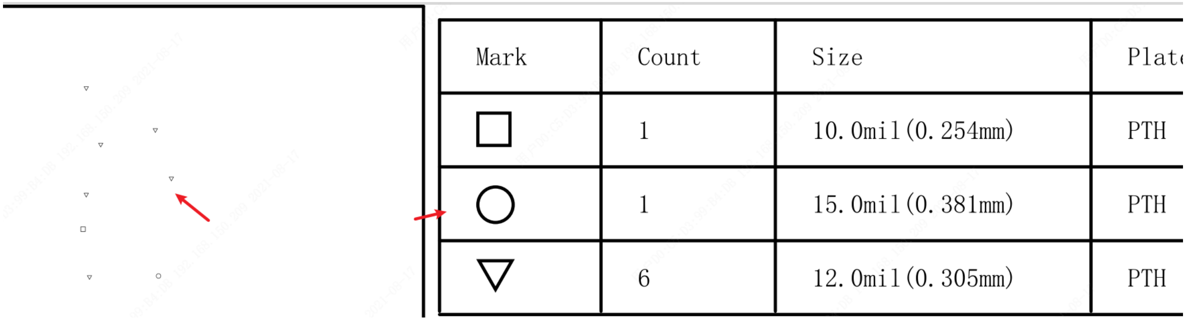


Click to set the parameters of the drilling table, and then click the position to generate the drilling table data. After Place, it is placed on the drilling layer by default.





The drilling table will be counted according to the type of drilling that appears in the current PCB: metallized hole PTH and non-metallized hole NPTH, and the size of the drilling hole, and will be identified with different identification symbols. When exporting PDF or Gerber, the drilling map Layers can be exported along with them, and corresponding identification symbols will be displayed at the corresponding drilling positions.



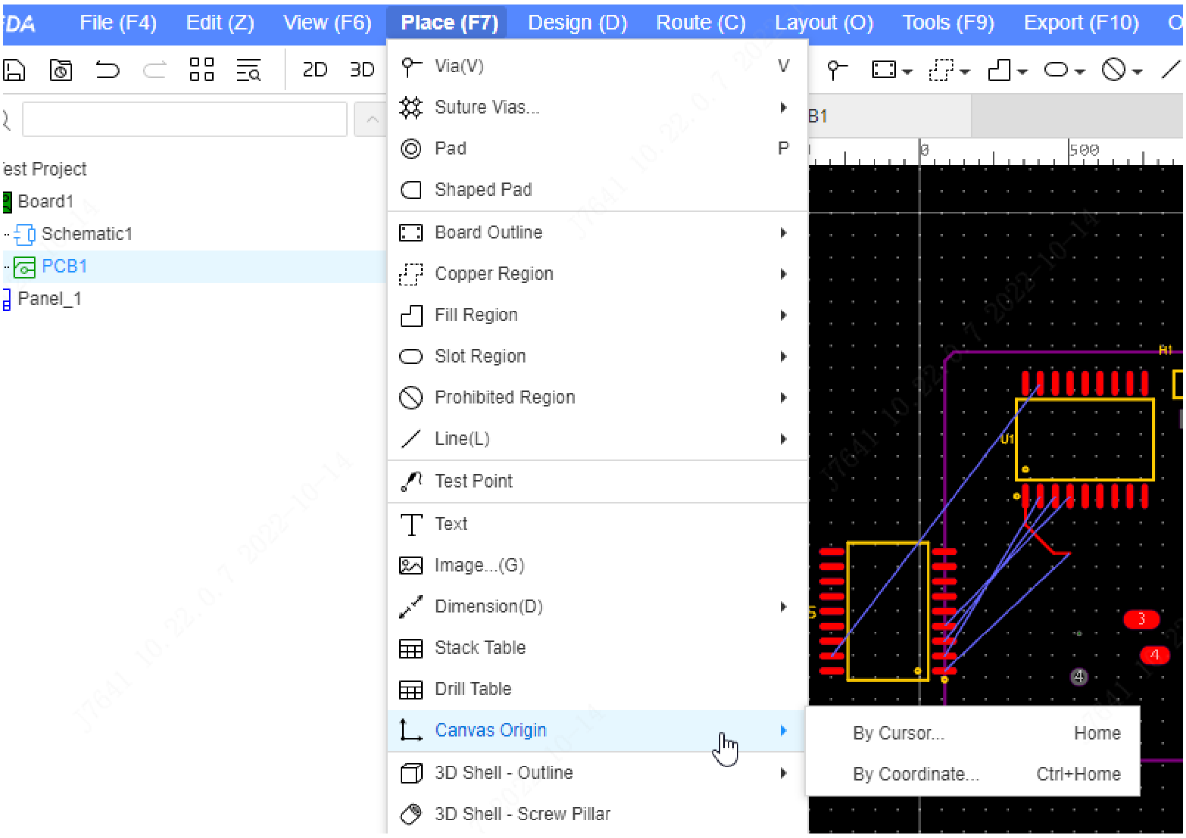
Mark	Count	Size	Plat
□	1	10.0mil(0.254mm)	PTH
○	1	15.0mil(0.381mm)	PTH
▽	6	12.0mil(0.305mm)	PTH

origin

Also known as the origin of coordinates, the role of the reference datum.

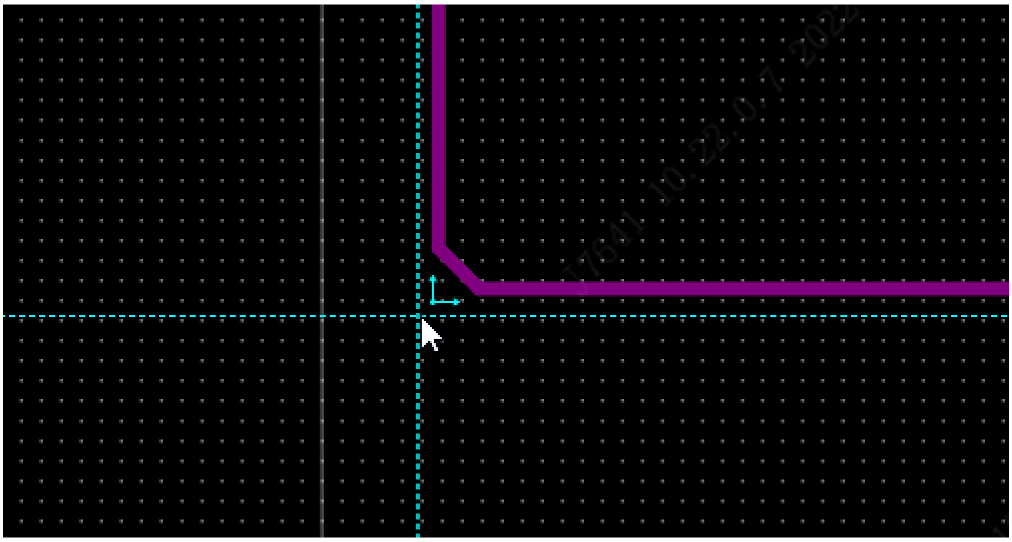
The specific operation is as follows:

- Top Menu - Place - Canvas Origin



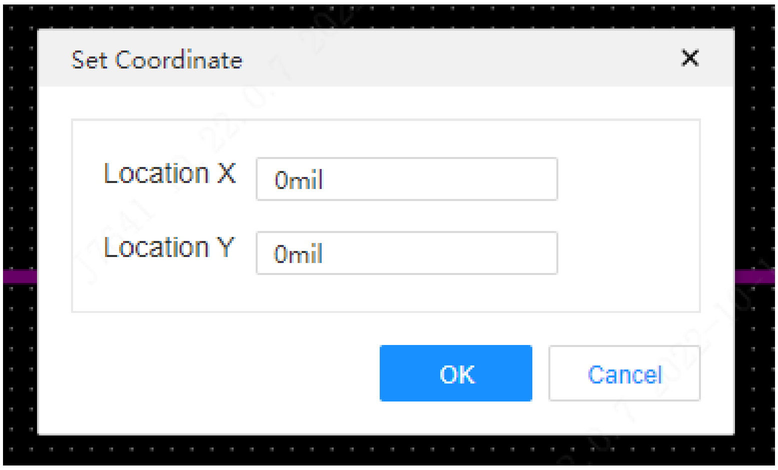
By cursor

After selecting, the mouse will turn into a cross, and then click the pad or the center of the symbol to set the coordinate origin.



By coordinates

After selecting, a view will pop up, select the coordinates you want to set the origin, click OK, and then the origin will be generated according to the coordinates you entered.



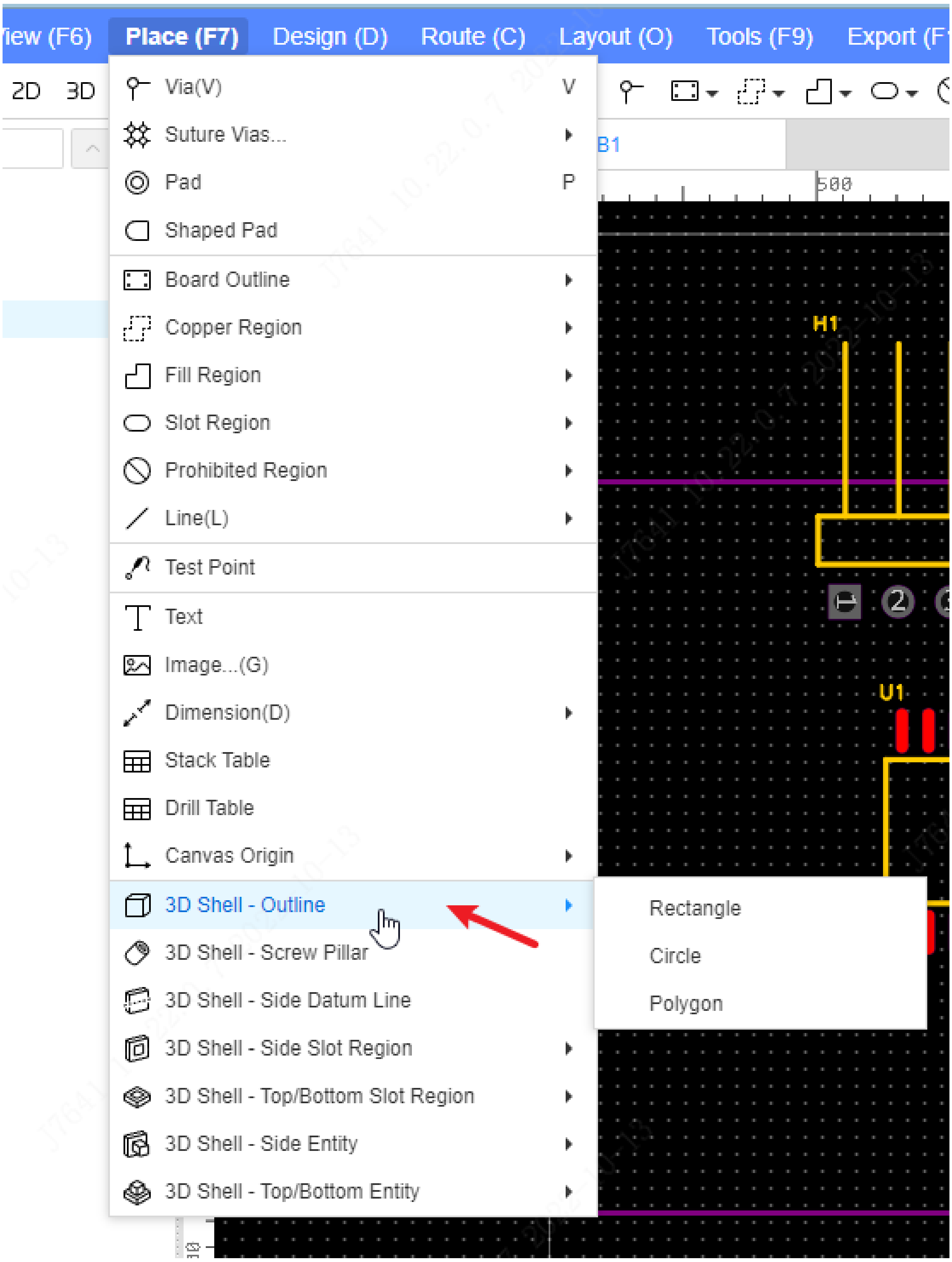
3D Shell - Board Outline

EasyEDA Pro supports simple 3D shell design, allowing users to quickly draw and create a simple shell.

If the simple 3D shell designed by EasyEDA Pro does not meet the needs, when exporting the 3D shell, choose to export the STEP or OBJ format, and continue the design in other professional 3D design tools.

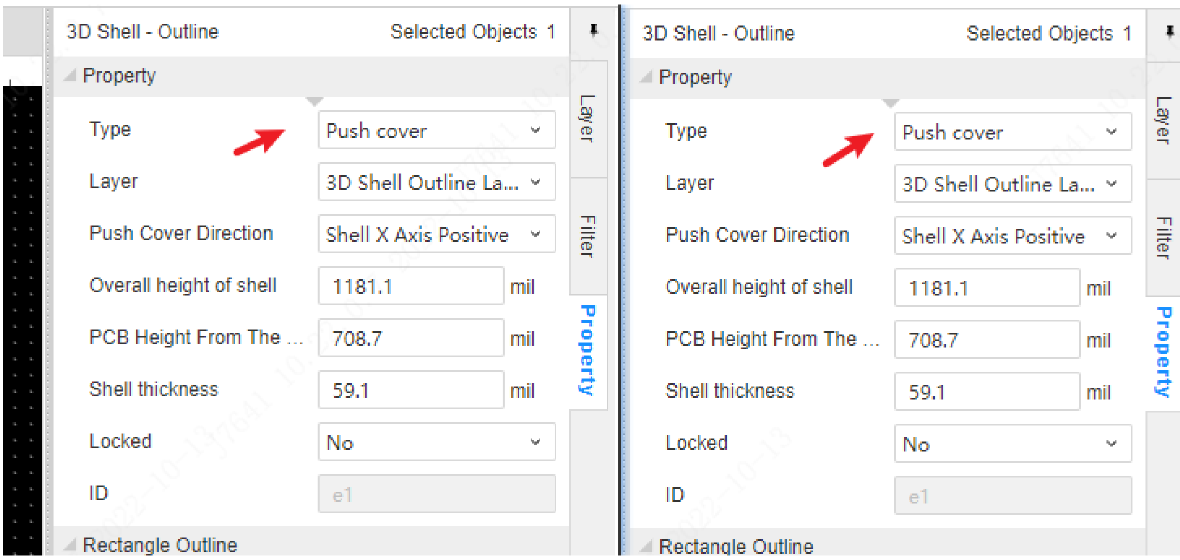
To design a 3D shell, you need to draw the Board Outline of the 3D shell first.

Entry: Top Menu - Place - 3D Shell - Board Outline:

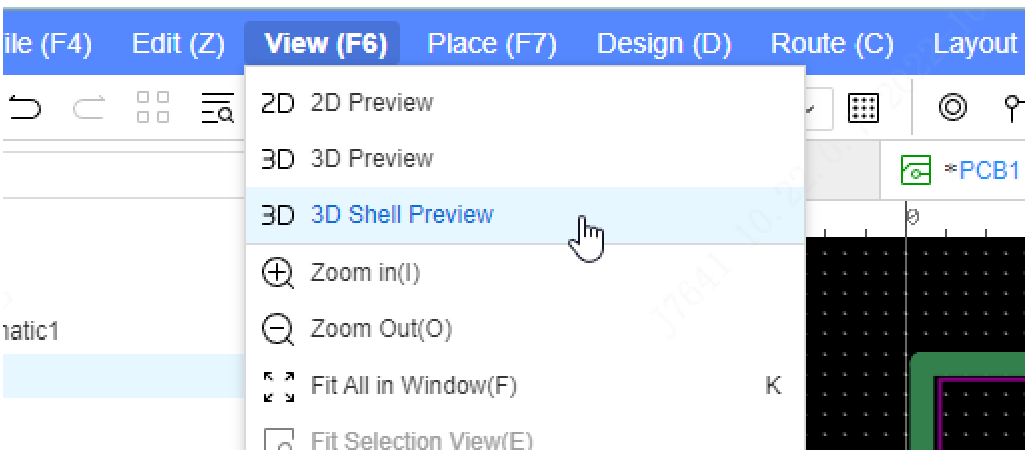


Drawn primitives are automatically switched to the 3D Shell-Board Outline layer:

Click the "3D Shell-Board Outline" primitive to set its properties in the property panel on the right:

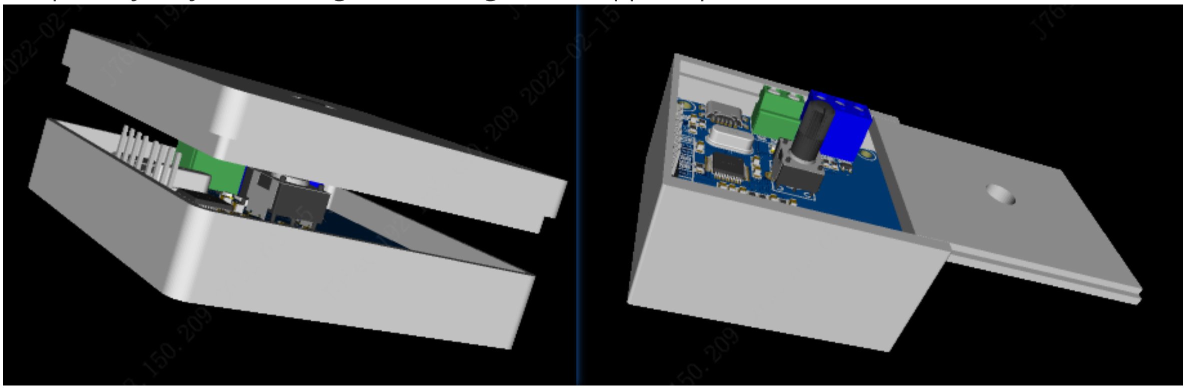


After drawing, you can go to: Top Menu - View - 3D Shell Preview to perform a synchronous preview, and the parameters related to the 3D shell modified by the PCB will be updated to the 3D preview synchronously. The 3D shell preview will open in a new separate window.



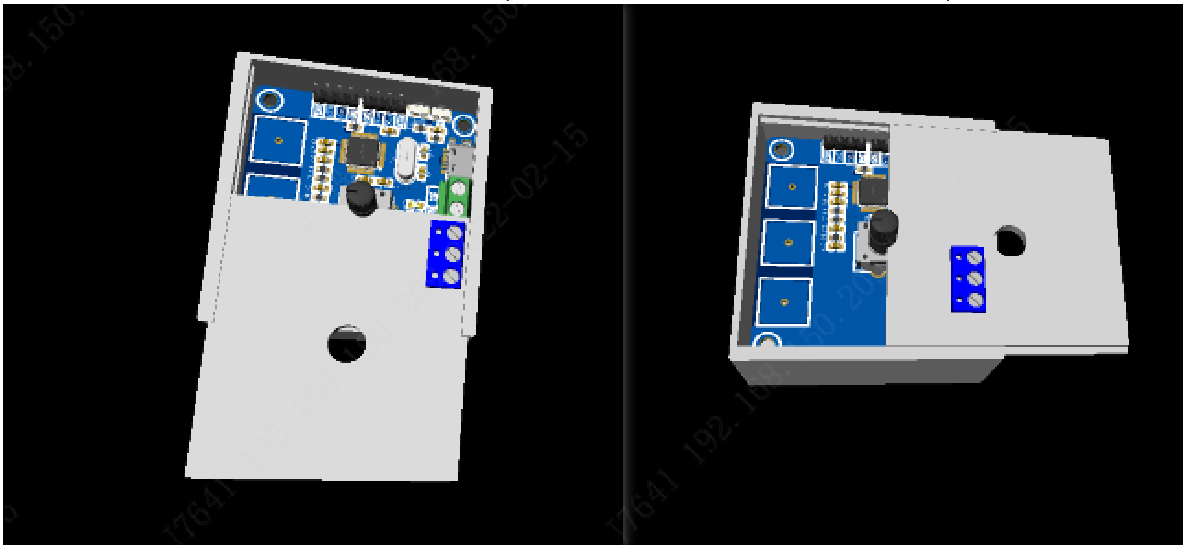
Property description:

Type: Upper and lower shell, push cover. EasyEDA Pro provides two housing types, and temporarily only the rectangular housing frame supports push-to-cover.



Layer: Like other lines, you can switch to other layers. After the layer is switched, the type will change accordingly.

Push cover direction: When the cover is pushed, the direction of the cover is pushed.



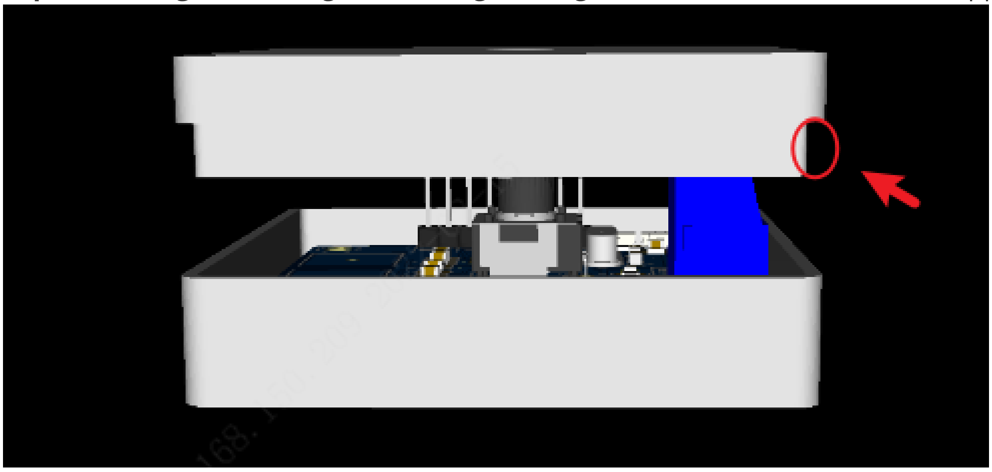
Overall height of shell: The overall height of the enclosure. For example, the height of the combined upper and lower shells.

PCB Height From The Bottom Surface of Shell: The height from the PCB board to the bottom casing surface is used to determine the flying height of the PCB.

Height of lower shell: The height of the lower shell. The height of the upper shell is automatically calculated based on the overall height.

Shell Thickness: The thickness of the case. It is recommended to be greater than or equal to 1.5mm, otherwise it is easy to break when manufactured with different materials and sizes. For details, you can check the process requirements of JLCPCB: https://jlcpcb.com/3d-printing

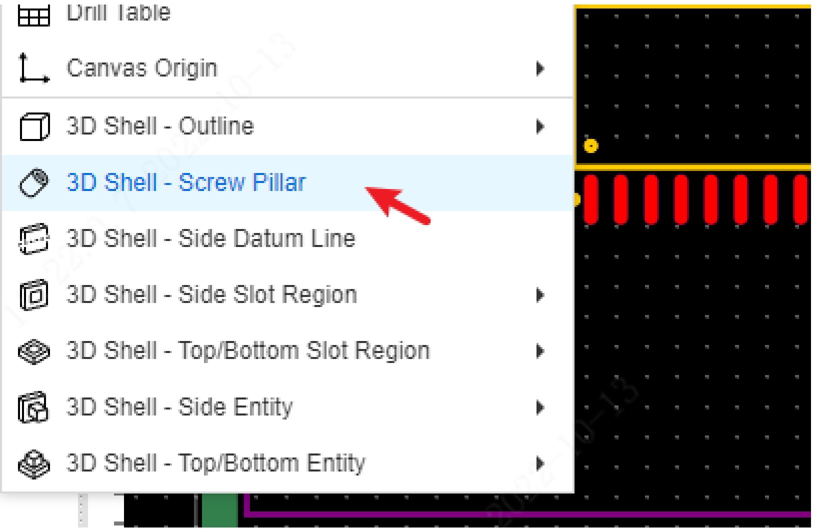
Top Inner Height: The height of the alignment groove on the inner wall of the upper shell.



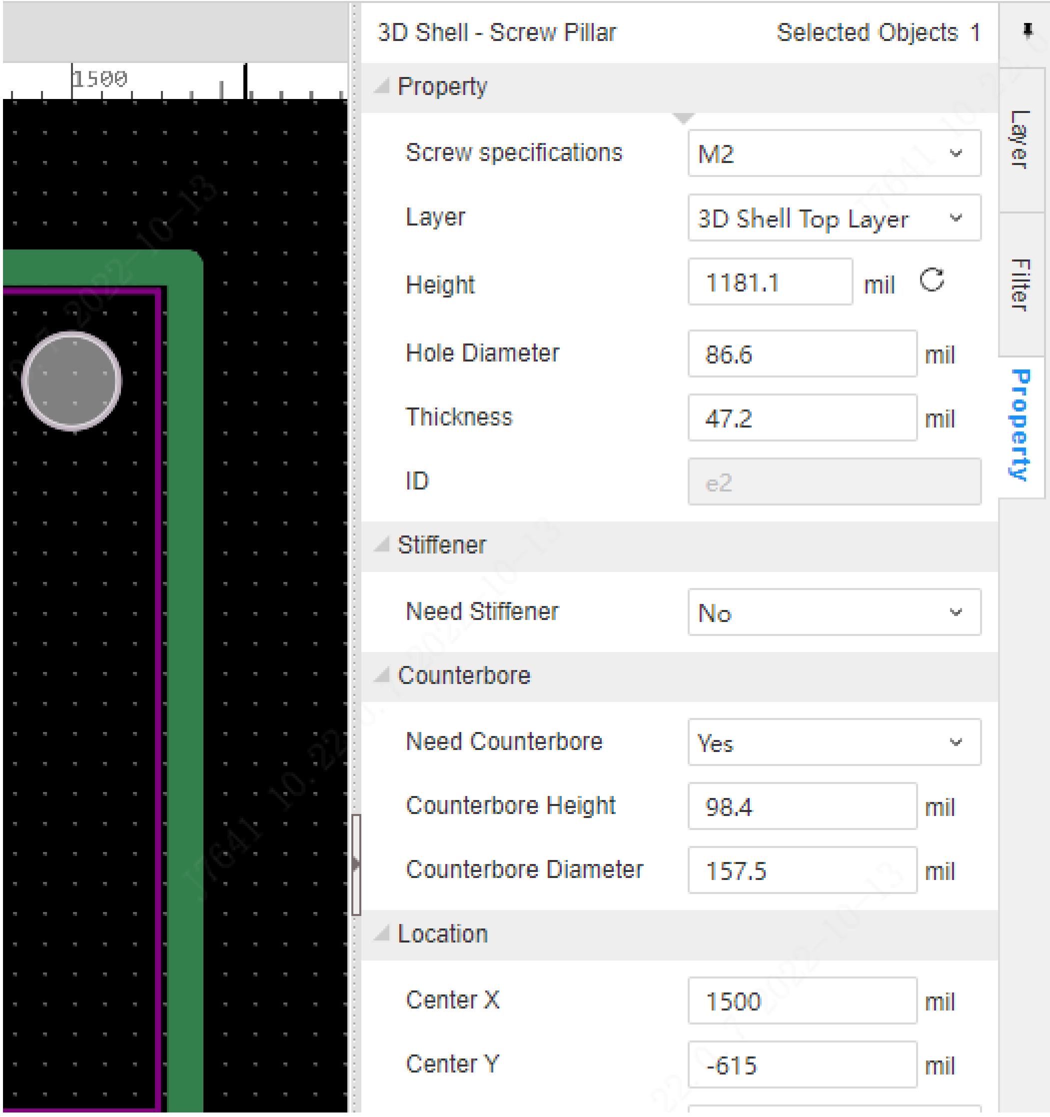
3D Shell - Screw Pillar

When drawing a 3D shell, you need to draw screw Pillar to hold the PCB board.

Entry: Top Menu - Place - 3D Shell - Screw Pillar

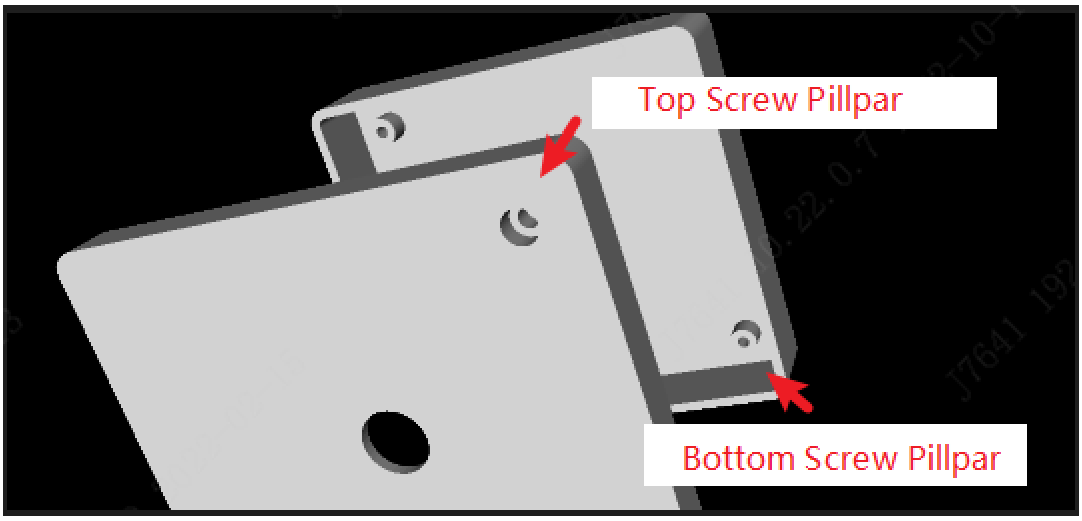


Property description:



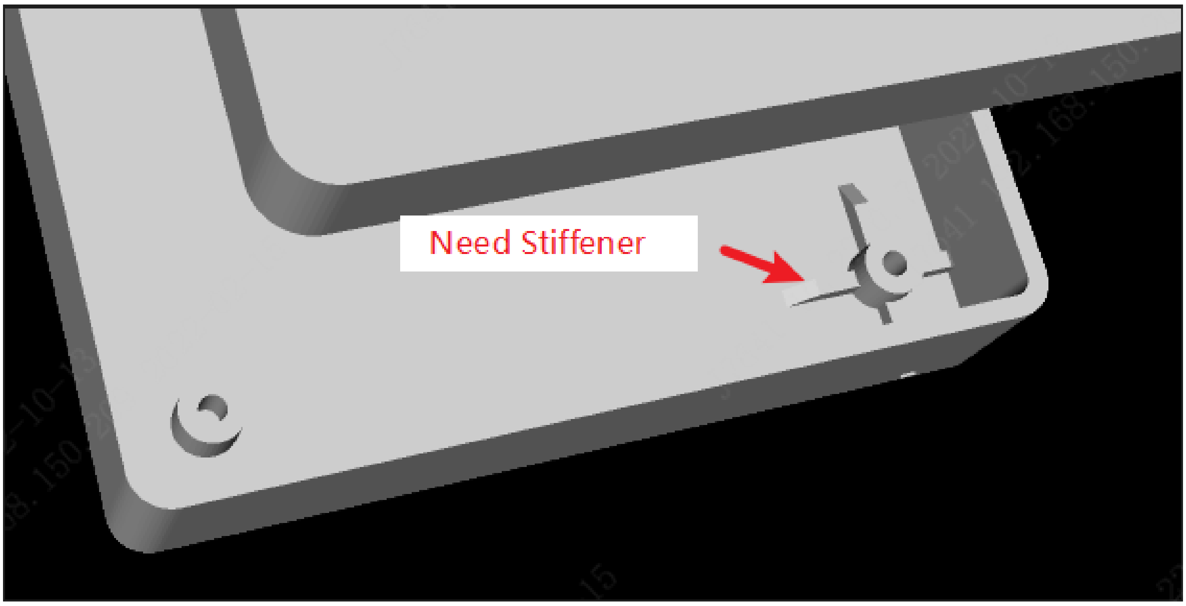
Screw Specifications: The editor has built-in six screw specifications M2~M6. Custom screw specifications are not currently supported. Please note that when drawing screw holes on the board, if you need to pass the board through the screw column, the inner diameter of the screw hole should be larger than the size of the screw column to avoid the board not being able to fit.

Layer: The spawn location of this screw post. Generates studs in the top shell or in the bottom shell. The top layer spawns directly through the top layer, with grooved screw holes.

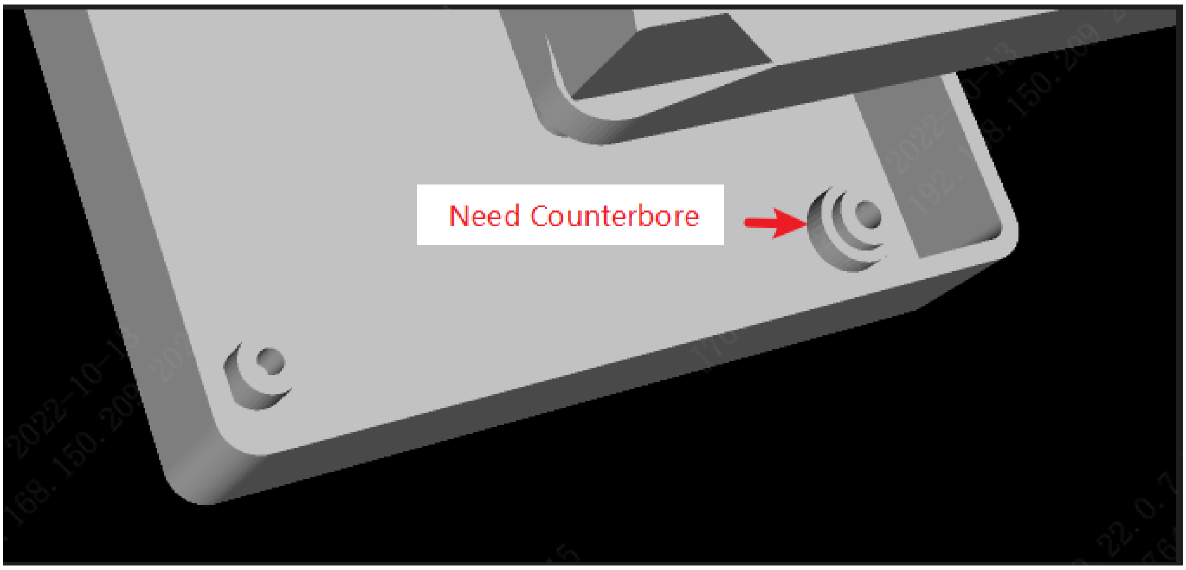


Height: The height of the screw post. Auto-calibration automatically sets a height based on the enclosure height.

Need Stiffener: The screw column is added with reinforcing ribs for fixing around it. Notice: the screw thikness contain the shell thikness.



Need Counterbore: Is there a countersunk style stud.



3D Shell - Side Datum Line

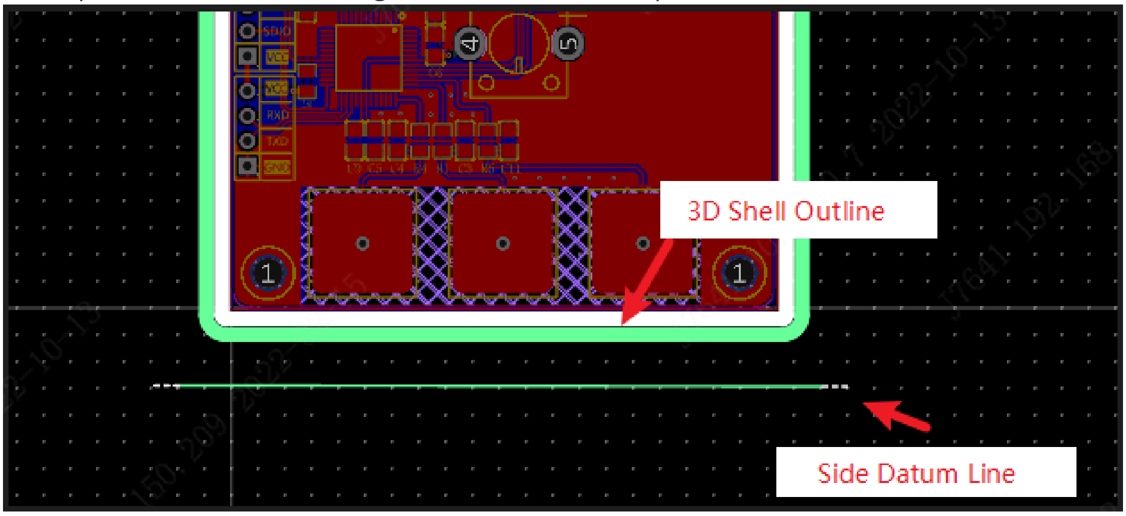
When you need to draw a side groove, you need this side reference line for positioning.

Entrance: Entrance: Top Menu - Place - 3D Shell - Side Datum Line

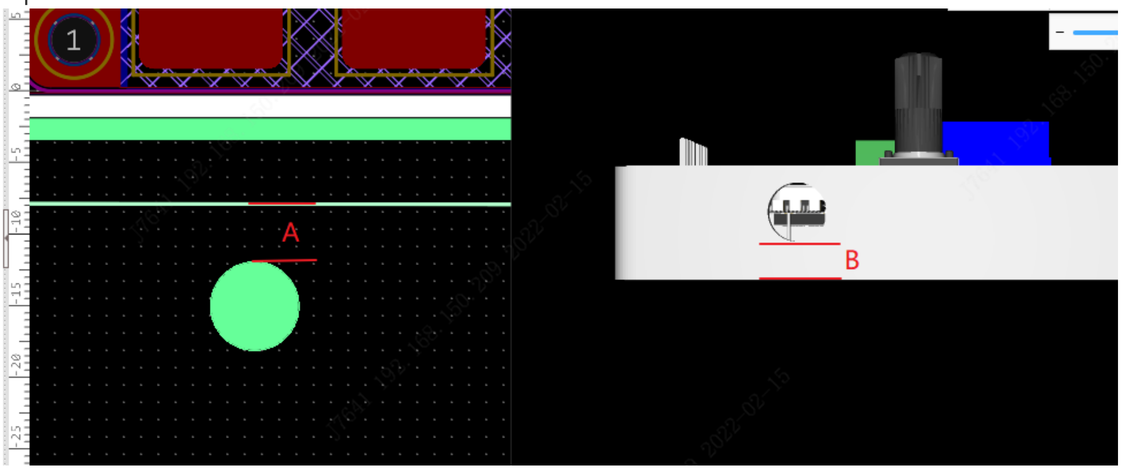
Side reference lines are only allowed to be set on the 3D shell border layer.

How to use:

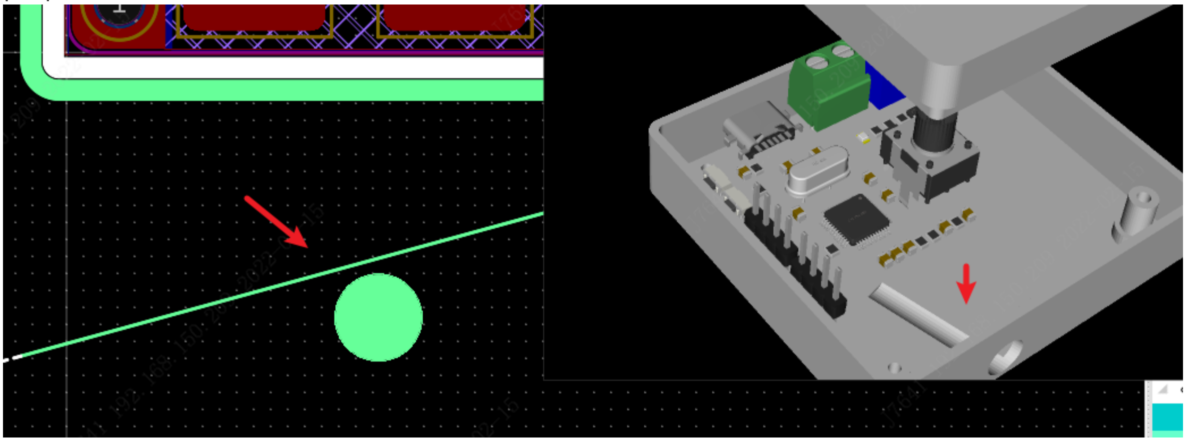
1. On the side of the shell that needs to be grooved, when determining the top view of the board, the frame line segment corresponding to the side
2. Place the side reference line, parallel to the corresponding border line segment, do not overlap with the border line segment, and need to be placed outside



3. Then place the side grooving area on the outside. The distance of the side pocket area from the baseline is the height of the pocket from the bottom. As shown below: The height of B is equal to the distance of A.



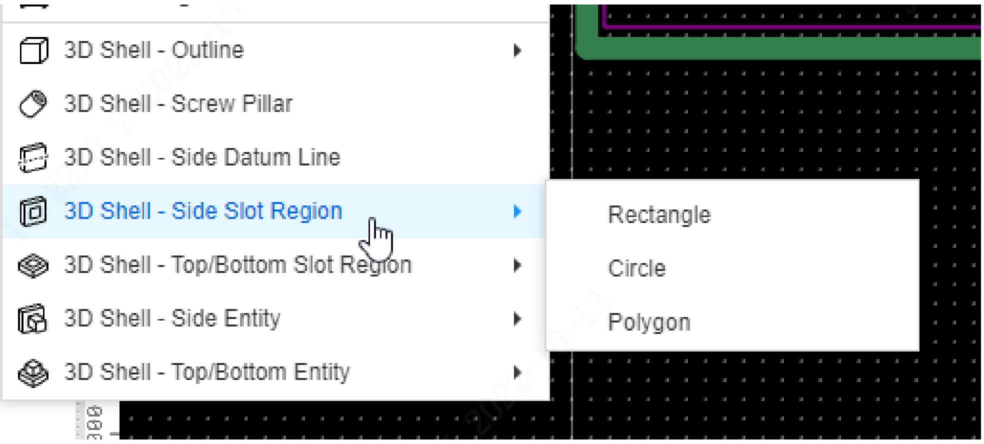
The side reference line can also be in any direction, and the side groove area will be cut perpendicular to the side reference line.



3D Shell - Side Slot Region

When the side grooves need to be drawn, the side reference lines are needed to locate, plus the side groove area.

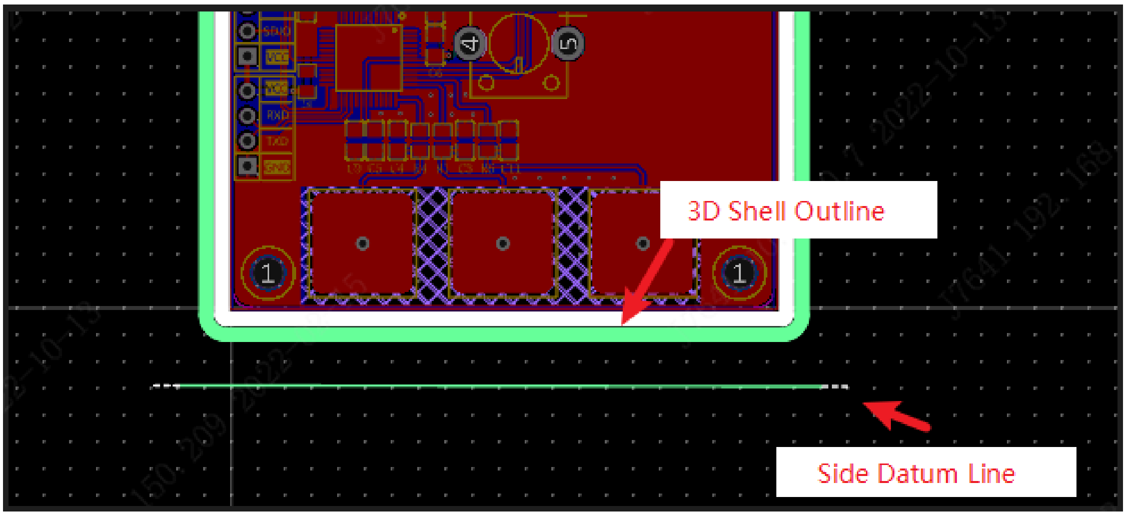
Entrance: Top Menu - Place - 3D Shell - Side Slot Region



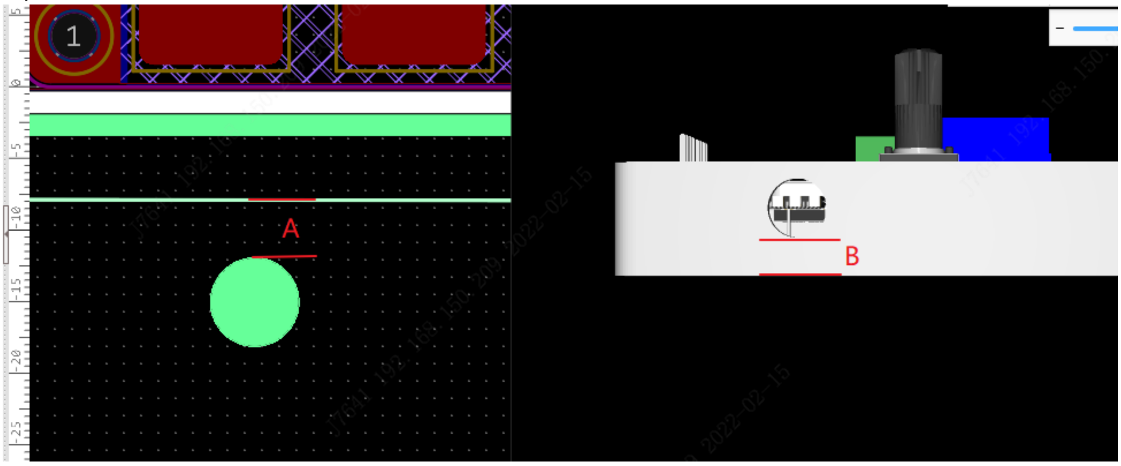
Side reference lines are only allowed to be set on the 3D shell border layer.

How to use:

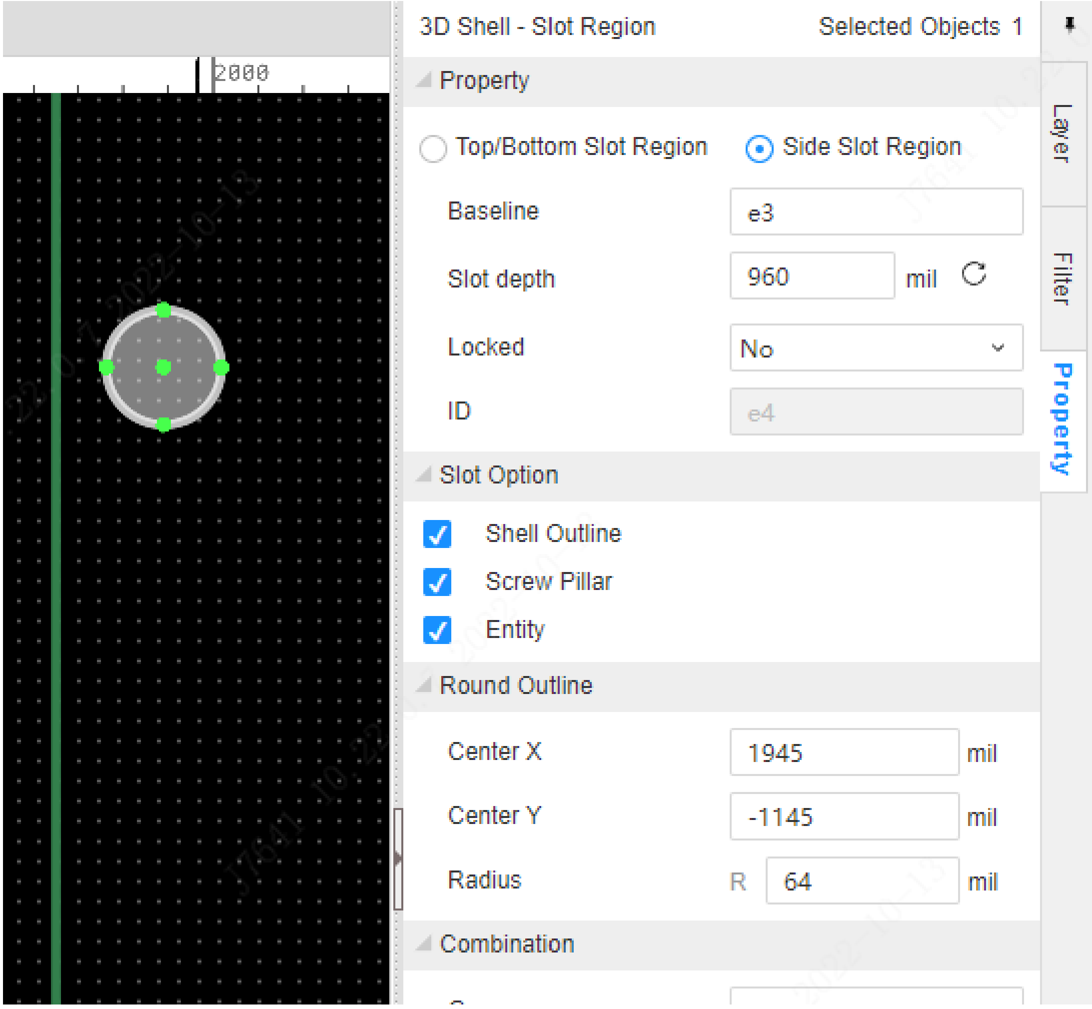
1. On the side of the shell that needs to be grooved, determine the frame line segment corresponding to the side when determining the top view of the board.
2. Place the side reference line parallel to the corresponding border line segment, do not overlap with the border line segment, and need to be placed outside.



3. Then place the side grooving area on the outside. The distance of the side pocket area from the baseline is the height of the pocket from the bottom. As shown below: The height of B is equal to the distance of A.



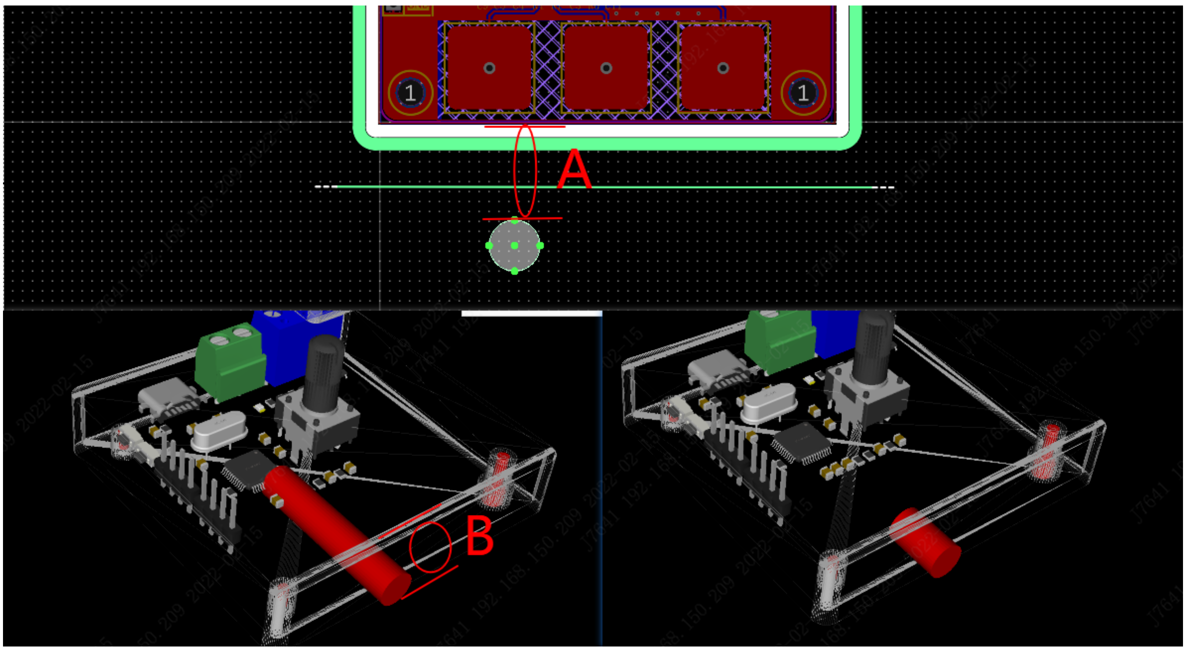
Side Solt Region Property:



Top and bottom pockets and side pockets are switchable.

Baseline: Determine the direction and face of the groove according to the corresponding base line. Click to select a new baseline.

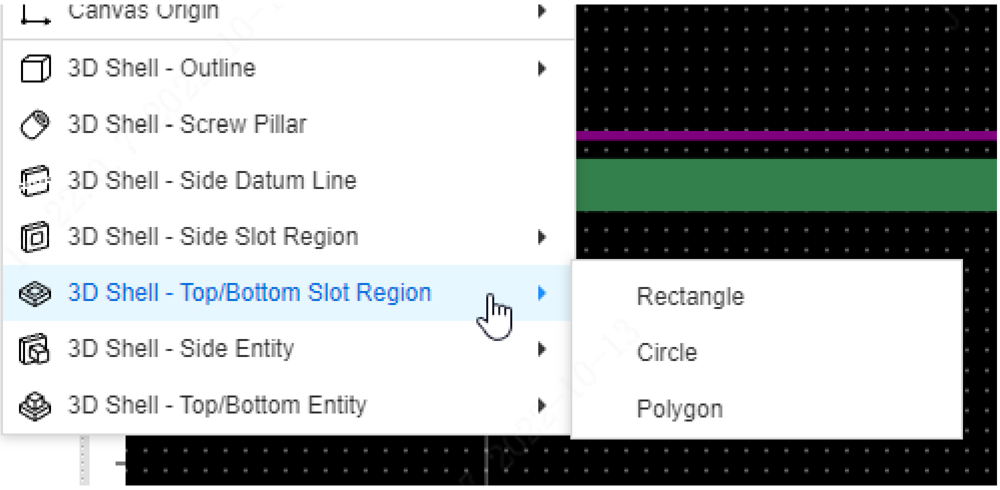
Slot depth: How deep the groove needs to be dug is determined according to your own design needs. Take the position where the side pocket area is placed as the starting position, and the distance to the shell frame is the pocket depth. As shown in the figure below: A and B are the same distance.



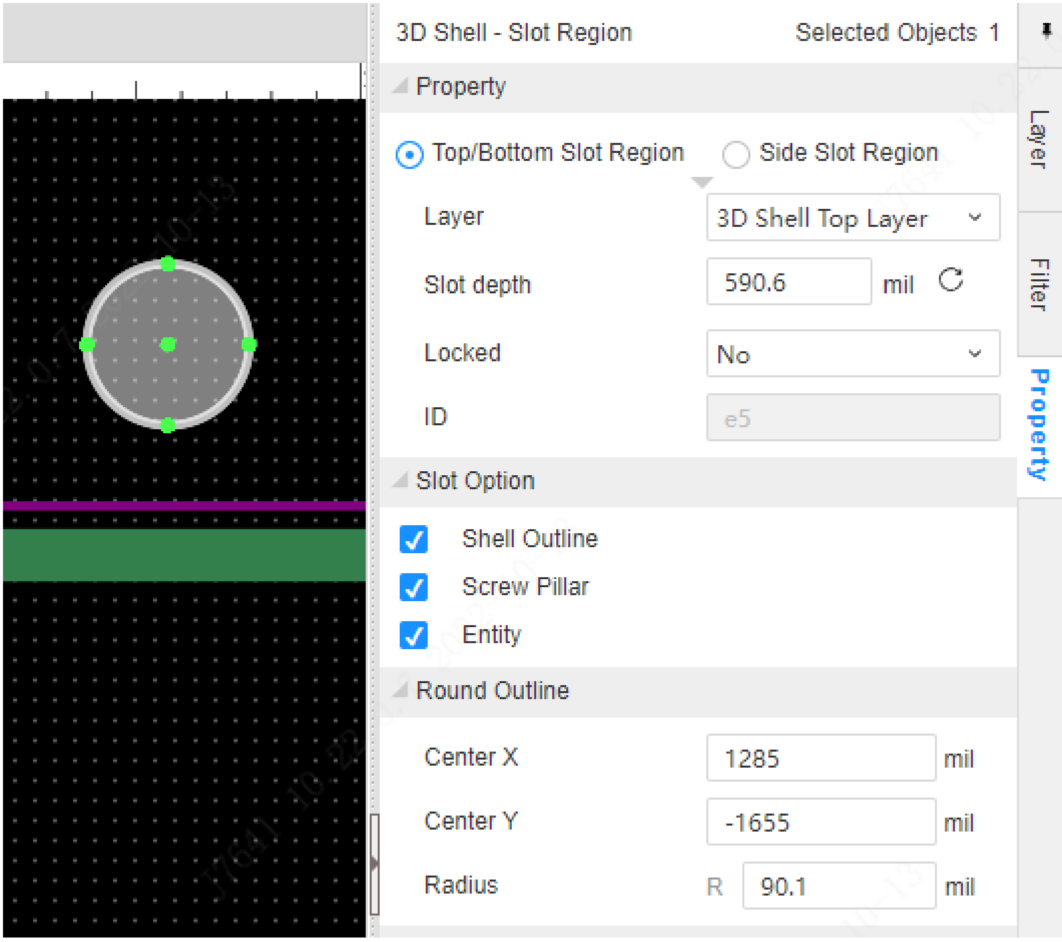
3D Shell - Top/Bottom Slot Region

Use the top and bottom pocket areas when you need to draw top and bottom pockets.

Entrance: Top Menu - Place - 3D Shell - Top/Bottom Slot Region



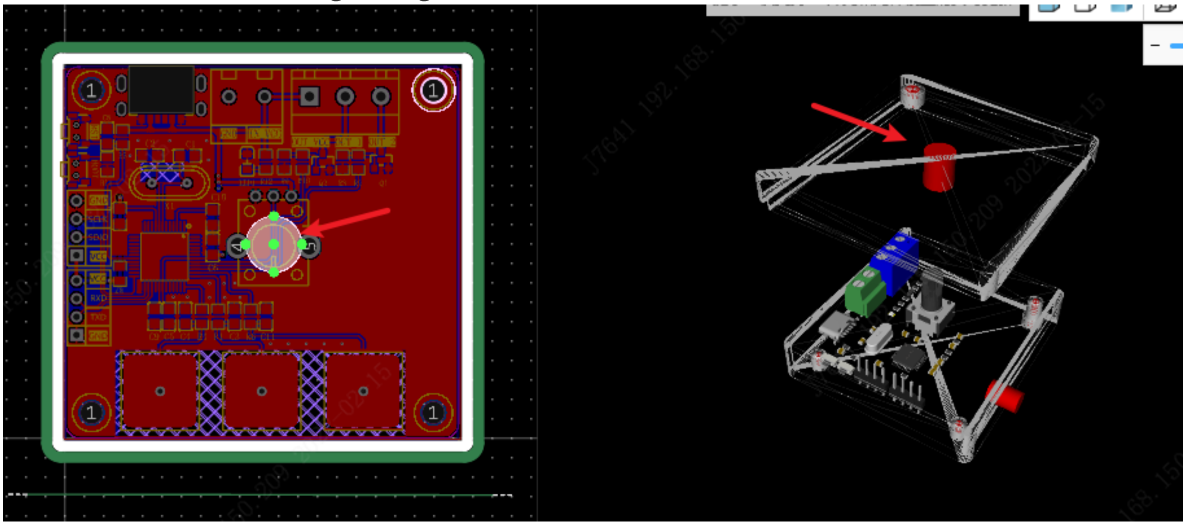
The top/bottom groove area acts directly on the top or bottom layer.



Property description:

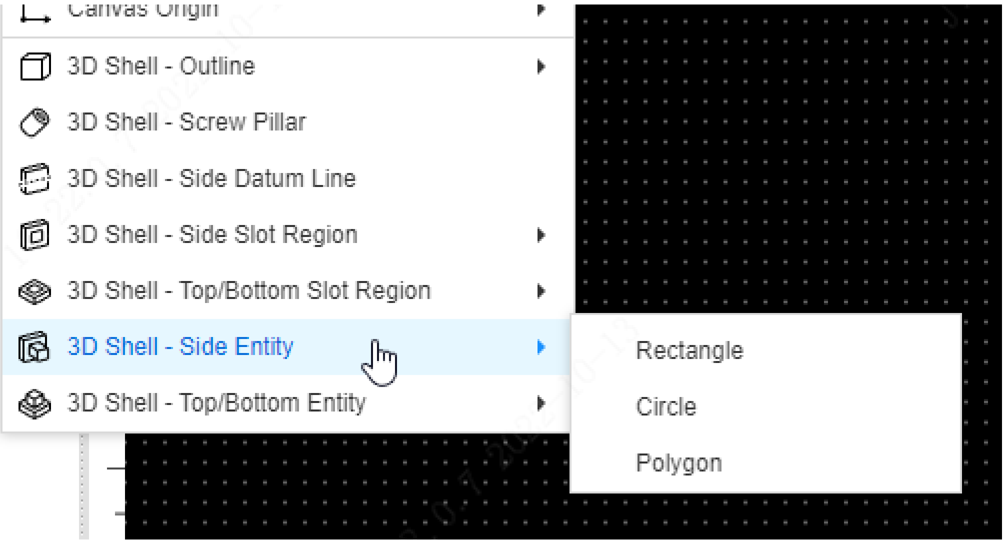
Layer: Set whether it is on the top shell or the bottom shell.

Grooving Depth: The depth of the groove. Based on the top or bottom surface as the starting surface, dig down or dig up. Auto-calibration automatically sets a value based on the thickness of the shell to ensure it can dig through.



3D Shell - Side Entity

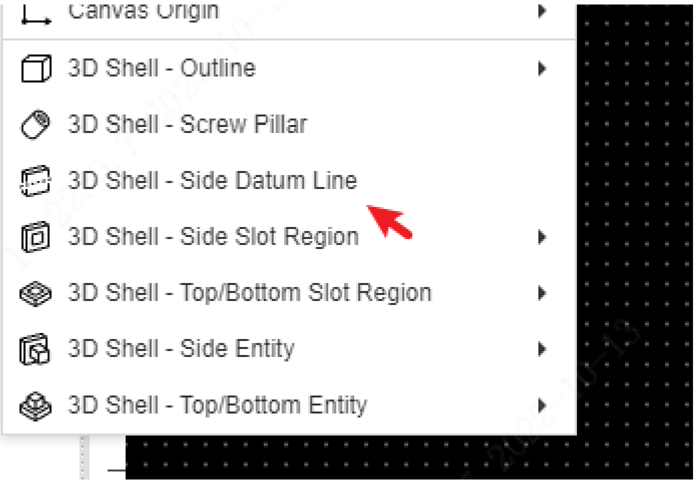
When you need to draw a protruding solid model, you can use the solid function.



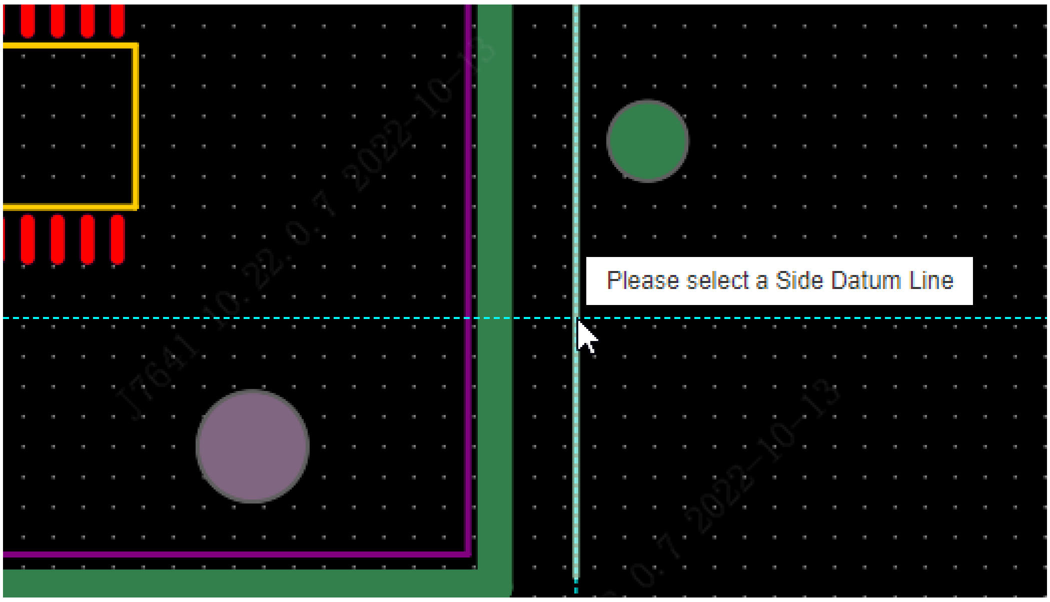
Drawing side solids is similar to drawing side pockets.

Steps:

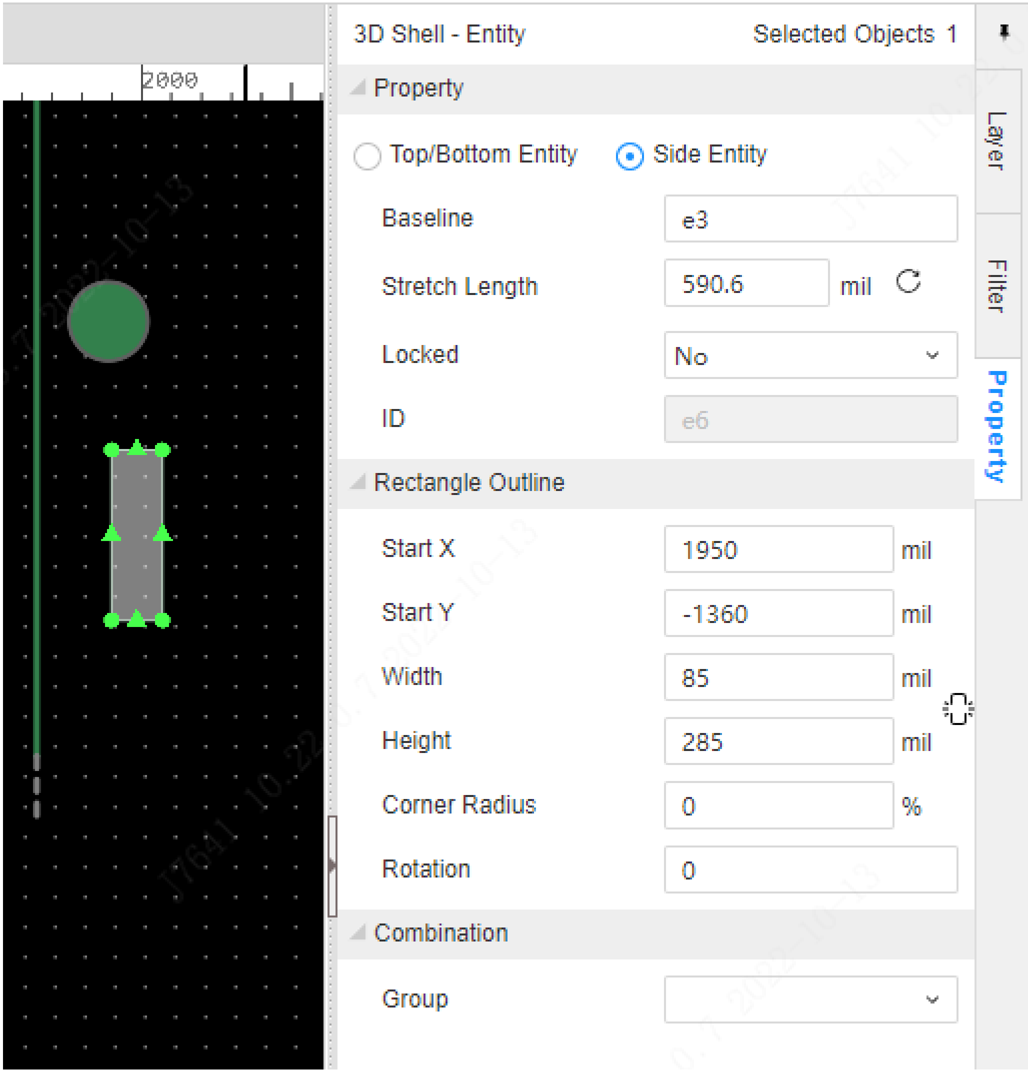
1. First draw a side datum line



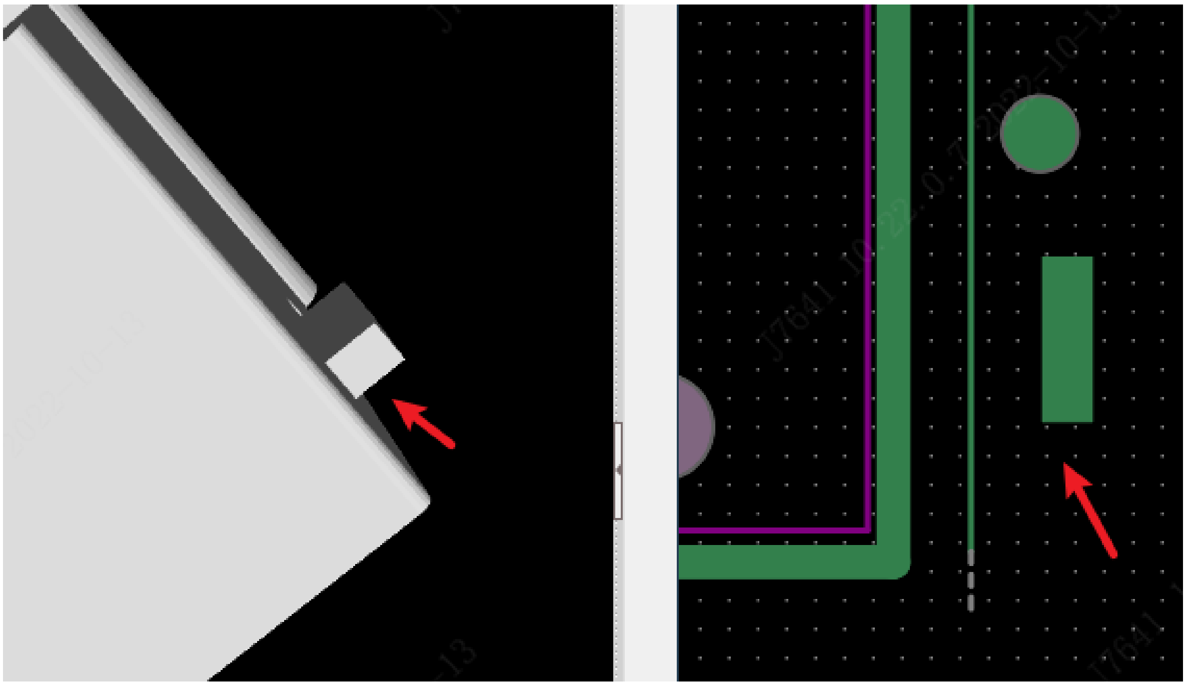
2. Draw the side entity, click the side datum line with the cursor and draw it



3. Adjust the distance and other parameters of the side entities

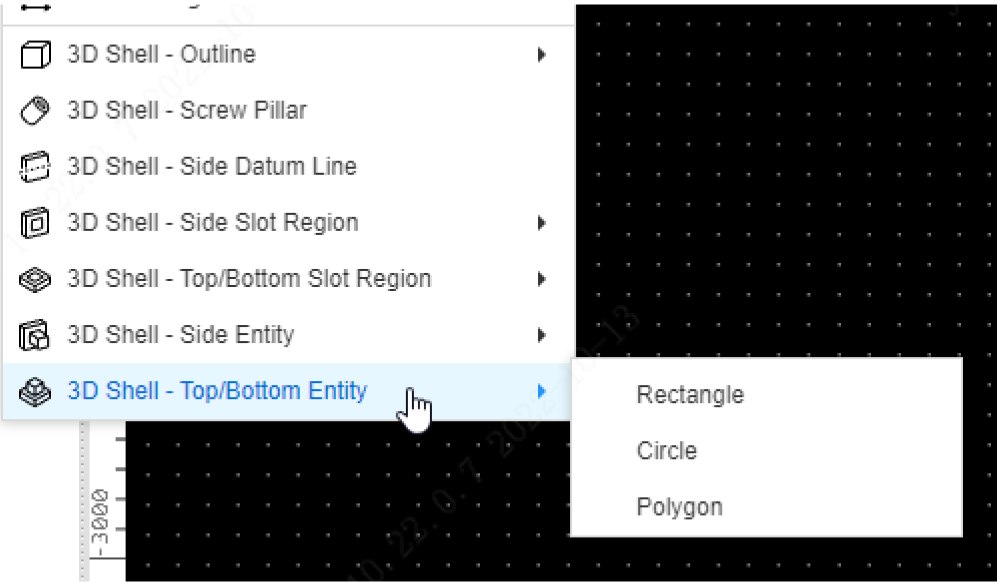


4. Preview the effect in the 3D shell preview



3D Shell - Top/Bottom Entity

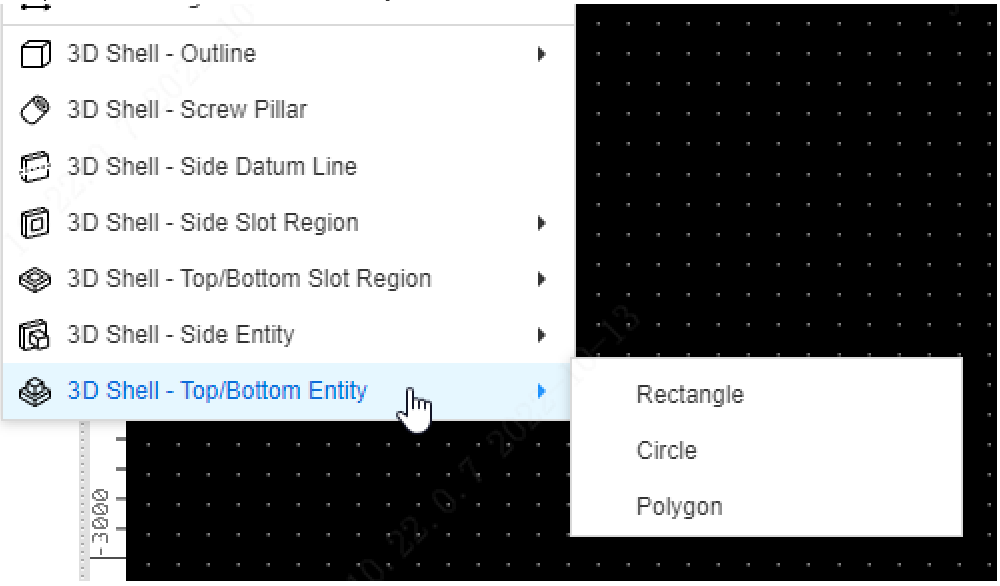
When you need to draw the top and bottom areas that need to be raised, you can use solid primitives



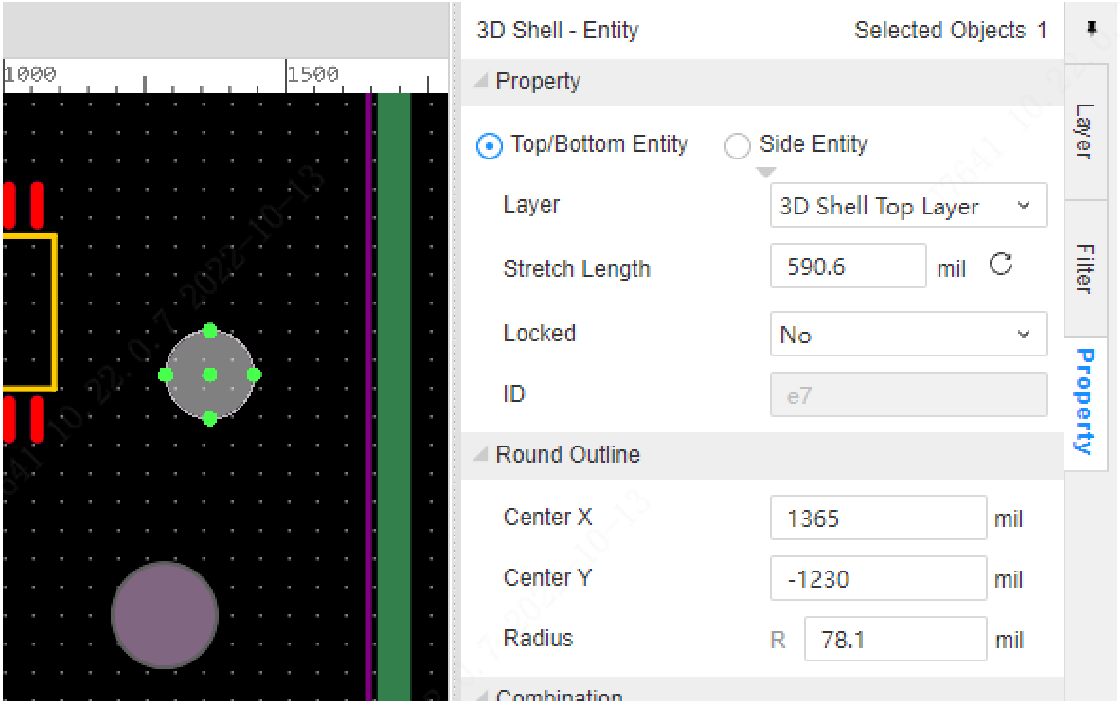
Drawing top and bottom solids is similar to drawing top pockets.

Steps:

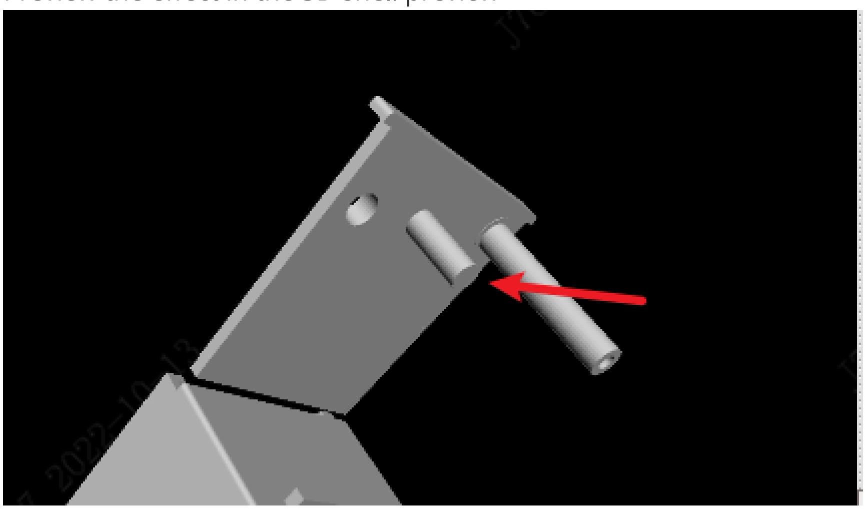
1. Draw the top and bottom entities directly inside the shell frame



2. Adjust the distance and other parameters of the side entity



3. Preview the effect in the 3D shell preview

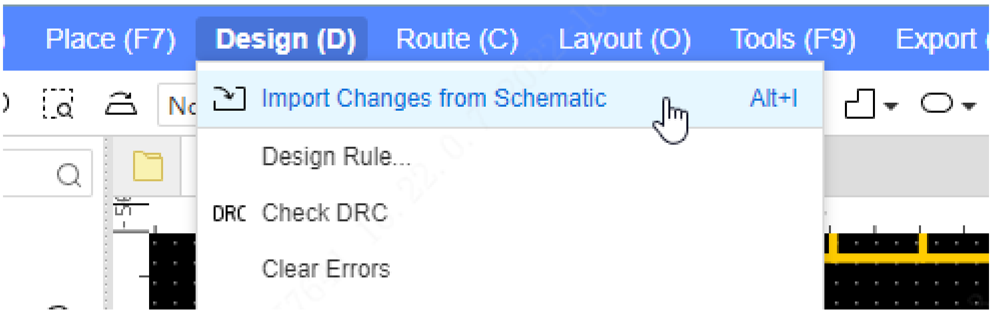


Import Changes from Schematic

When you modify the schematic, you can easily import the changes to the PCB. You can pass:

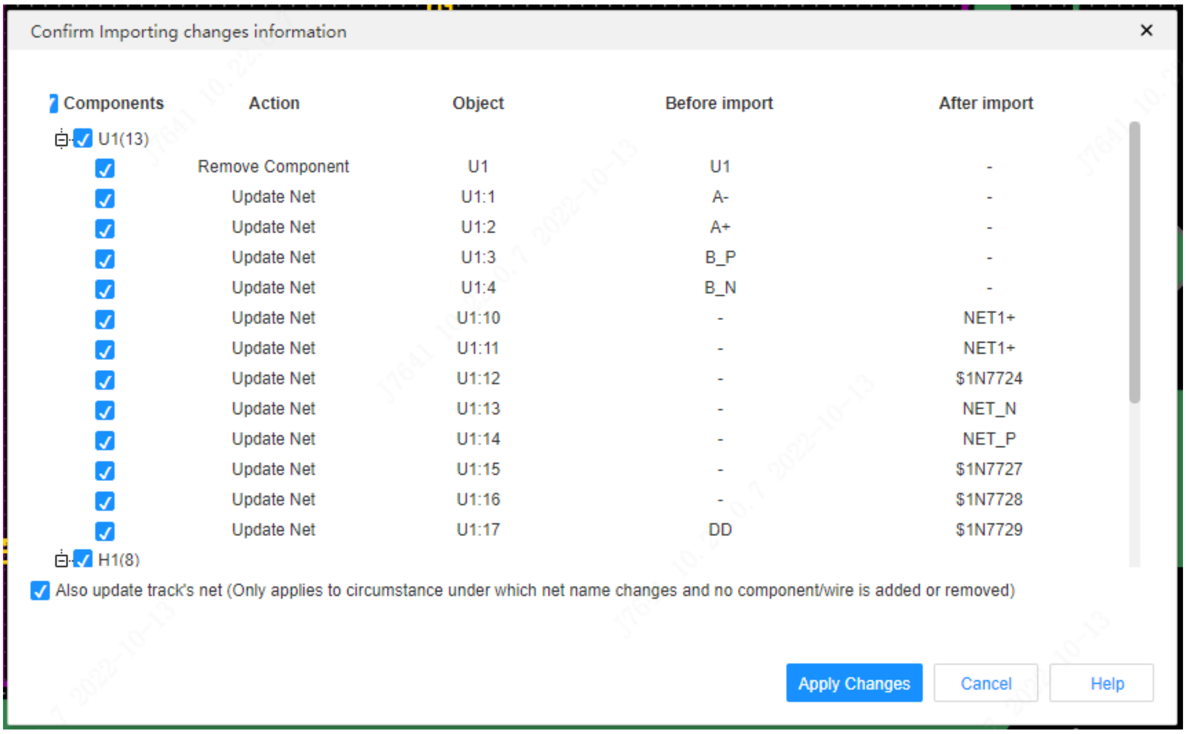
- The **Update PCB** function of the schematic can be used.

Entry: Top Menu - Design - Import Changes from Schematic

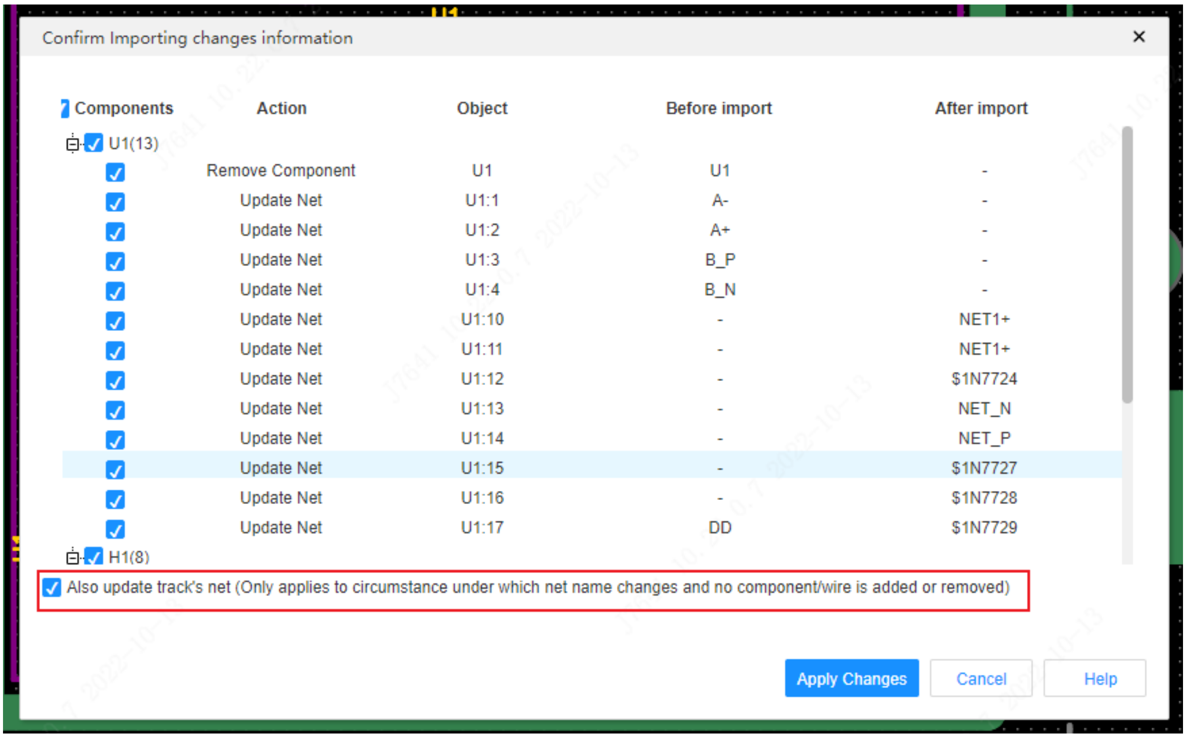


If there is an error in the schematic, a pop-up window will remind you, such as duplicate labels, missing footprints, etc.

If there is no problem, a confirmation dialog will pop up:



If you need to update the wire nets in the PCB at the same time, check the "Also update wire nets" option. The editor automatically updates the associated wire network based on the pad's network.



If it is confirmed that there is no problem, click "**Apply Modifications**" to update the PCB.

Notice:

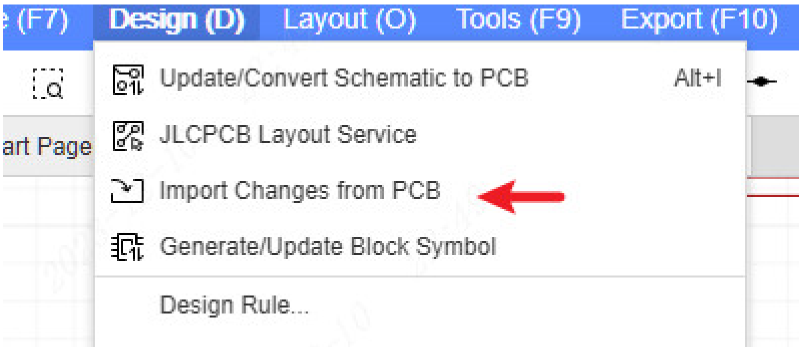
- Because the schematic net name is generated after calculation, if some nets are changed after you modify the schematic, after updating to the PCB, the traces already laid out by the original net will not be deleted.
- When "Update wire nets at the same time" is checked, update the pad net and update the pad related wires with the pad net. There may be changes in the wire net. You need to manually change the old net name of the wire. Change to a new network name! Most schematics can be updated to the PCB after updating the network, such as modifying the device number, modifying the net label, etc. However, if the schematic diagram has added or deleted devices, the wire network of the PCB needs to be modified manually. You can modify the network by clicking on the wire - right-click menu - select connection - properties on the right.
- After PCB import changes, some changes you cannot undo!

Update PCB to schematic

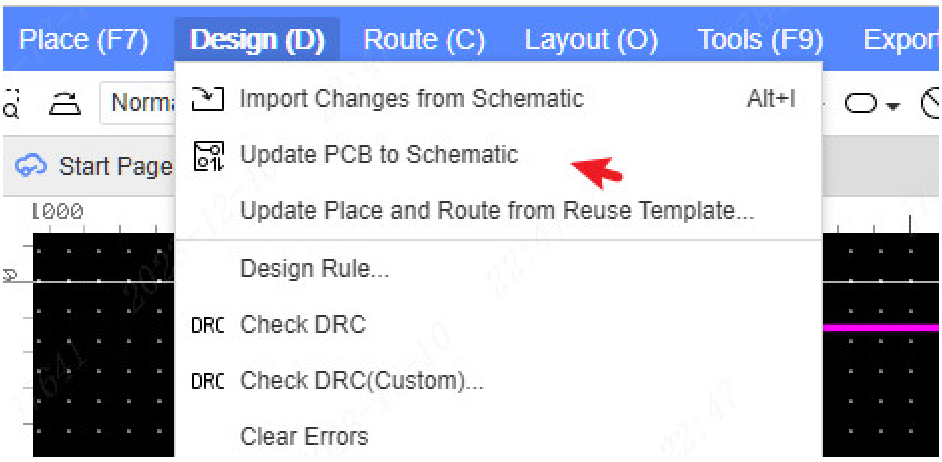
Under normal circumstances, PCB design needs to complete the schematic diagram first, and import the latest netlist into the PCB through the schematic to PCB or PCB import update function. After the PCB layout and routing is completed, if you need to reset the bit number based on the PCB layout information, or replace a certain component, you can complete such operations in the PCB first, and then update to the schematic in reverse. ways to improve work efficiency.

Operation entrance:

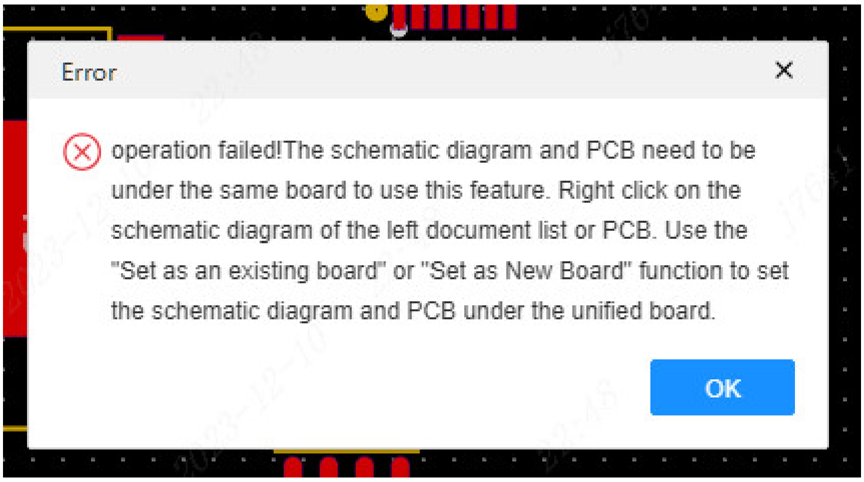
In Schematic, Top Menu - Design - Import Changes from PCB



PCB, Top Menu - Design - Update PCB to Schematic

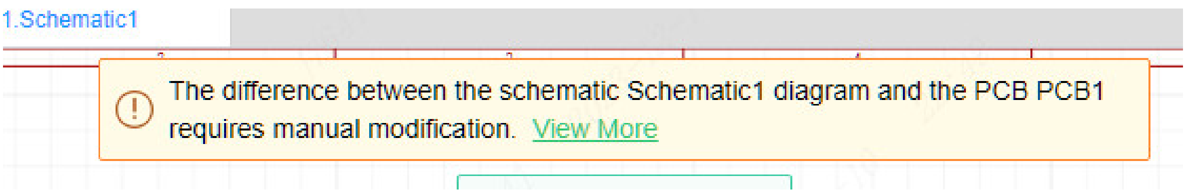


Note: When converting PCB to schematic, you need to ensure that they are under the same board, otherwise they will not be able to be converted to each other.



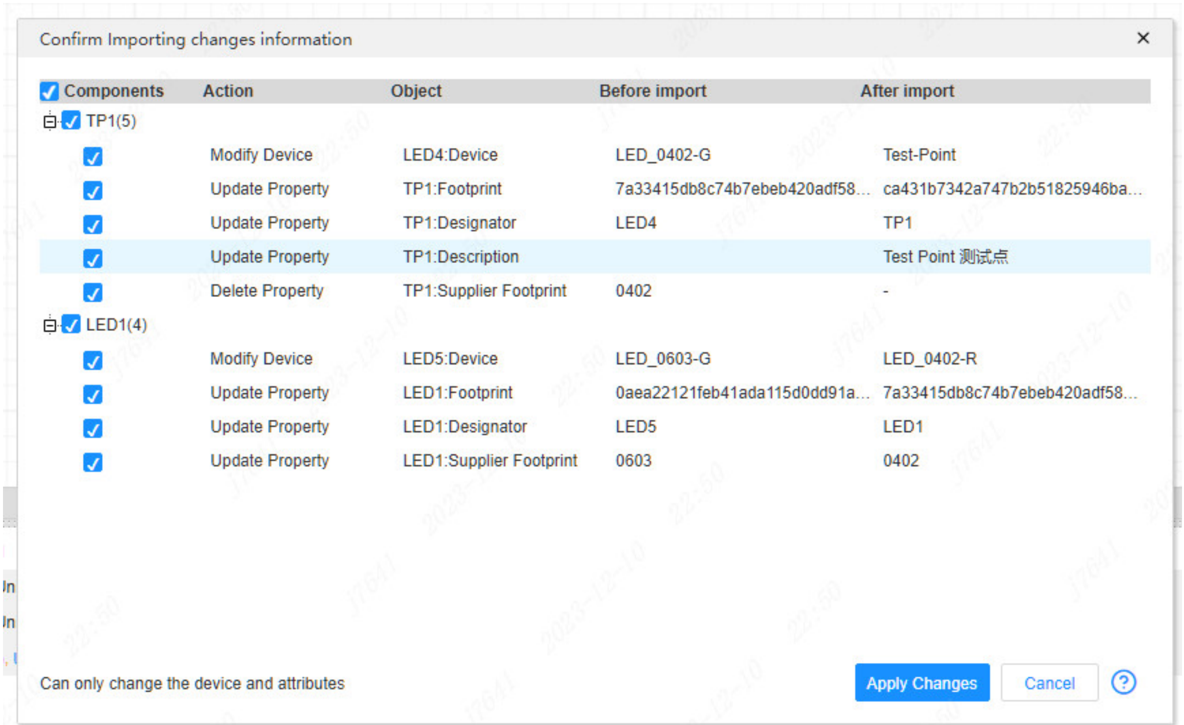
Check netlist differences

PCB to schematic diagram only supports the modification of component attributes (such as modifying the component's device and tag information). The addition, deletion of components, and network modifications will not be transferred to the schematic diagram. At this time, relevant prompt information will appear in the center of the log and canvas.



You can click on the component in the log to locate the specific location of the schematic diagram, view and make modifications.

Modifications to the component properties will appear in the pop-up confirmation import information pop-up window that pops up next.



Apply changes

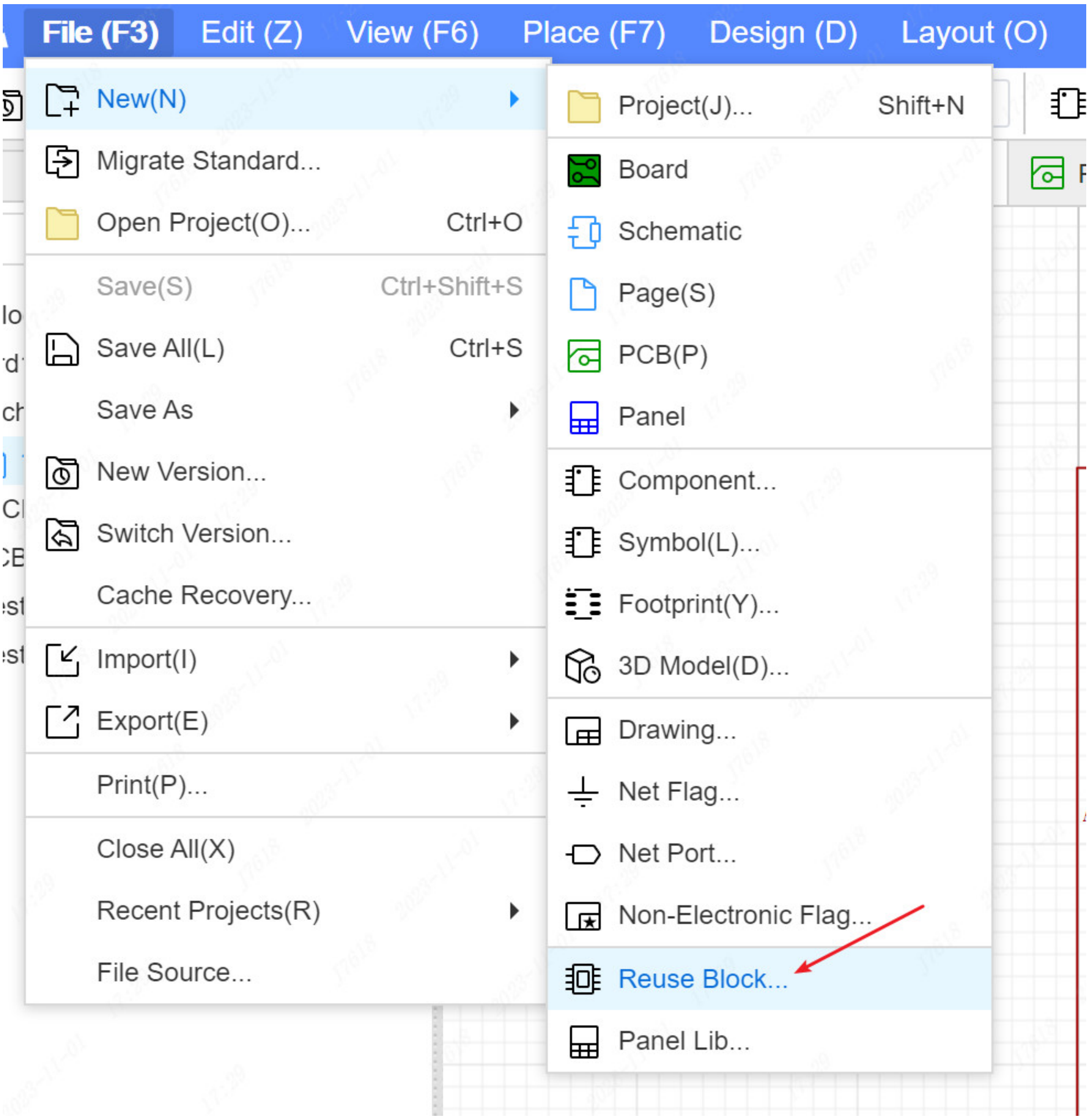
Check and check the items that need to be modified, click Apply modifications, and the corresponding properties will be updated to the schematic.

Update Place and Route From Reuse Template

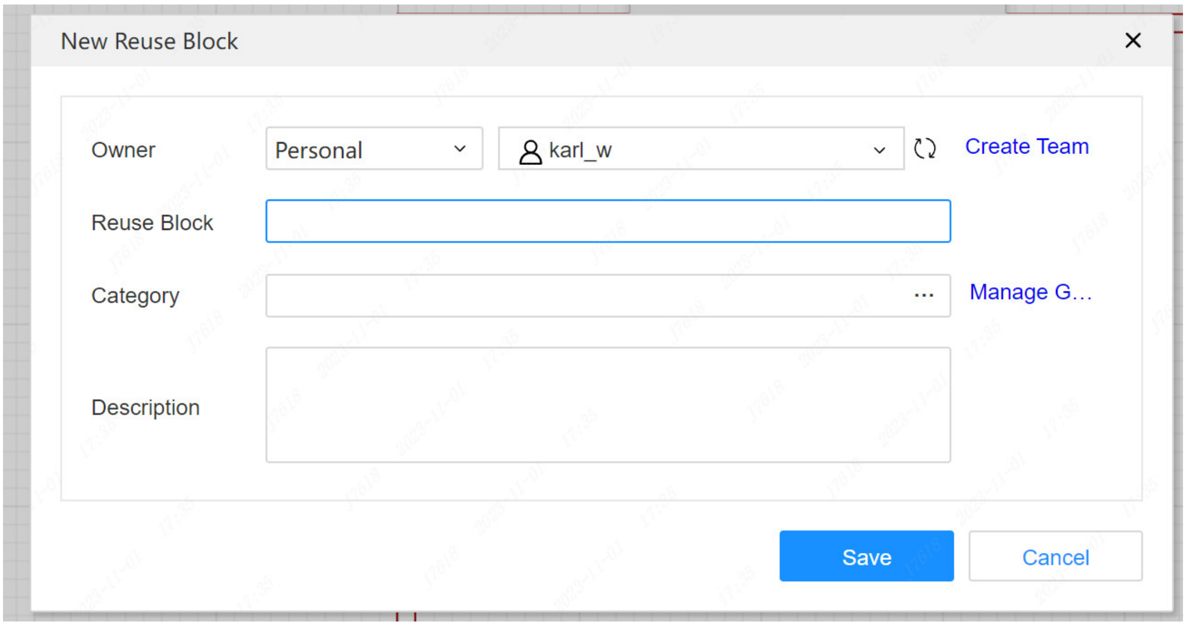
The professional version of easyEDA pro a reuse module function. Through this function, the PCB can be reused to achieve the purpose of layout wiring reuse.

Create reuse block

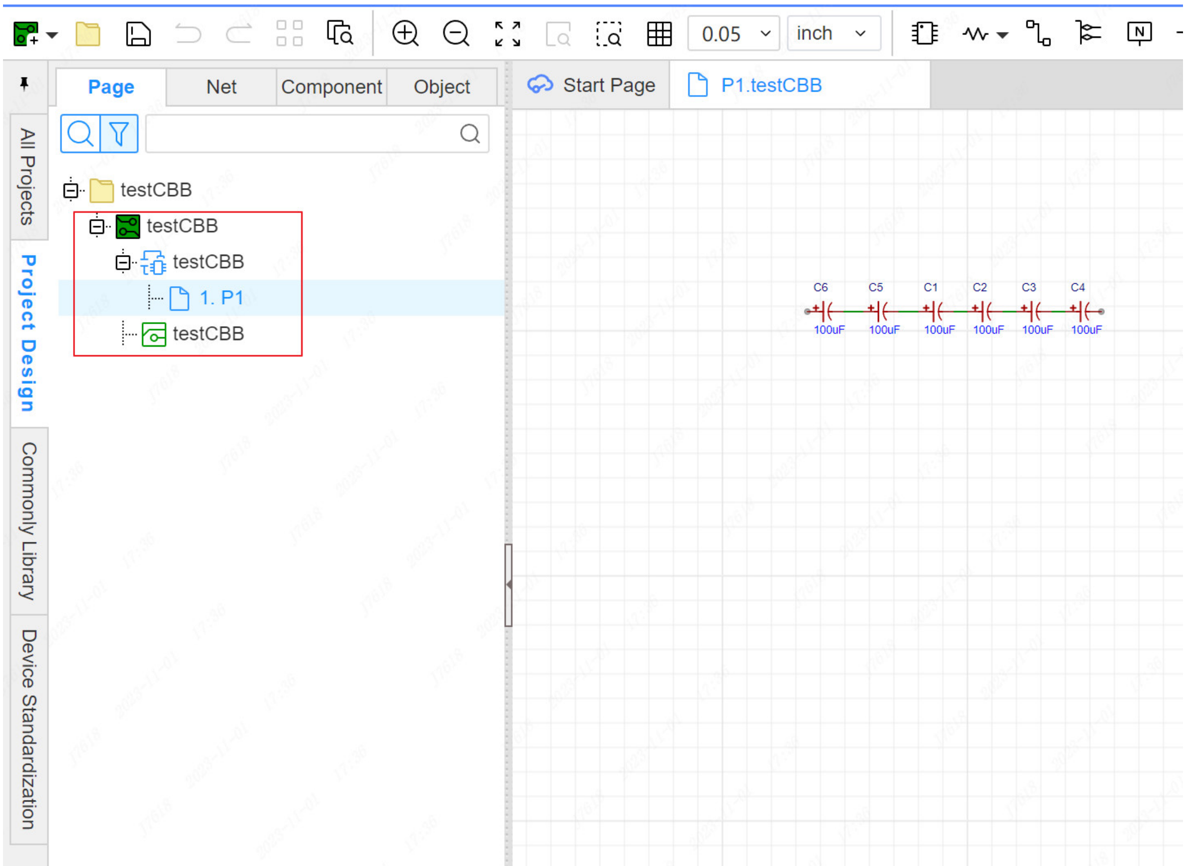
- Steps: top menu - file - new - new - reuse block



The reuse block is a special project that will be stored in the reuse module library. When newly built, the module can be set to classify and describe it.



After the successful creation of the reused module, it will create a board that contains schemes and PCB under the project by default, a symbol.

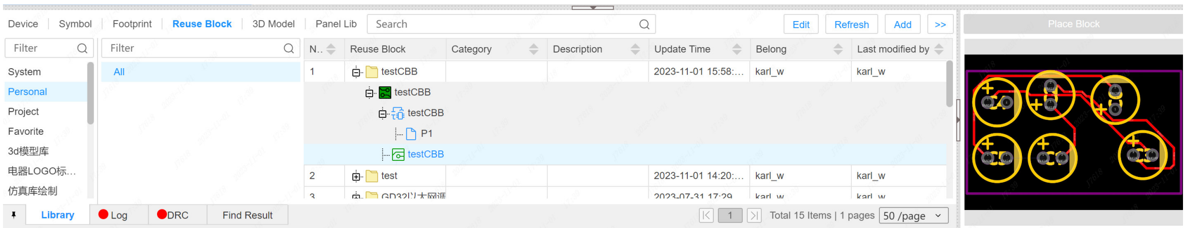


Notice:The principle diagram, PCB, and symbols are associated through the name. If the names of a certain type of type are modified separately, they will be disconnected between them when quoting, and they cannot be reused together.

In the module engineering, the principle diagram and PCB are designed. Before saving, you can click the schematic diagram to the PCB again.If you need to reference the schematic diagram in a symbol, keep the pin of the symbol with the name and number of the network port of the schematic map page consistent, and you can view the reuse block-create a reuse block.

Place the reuse block

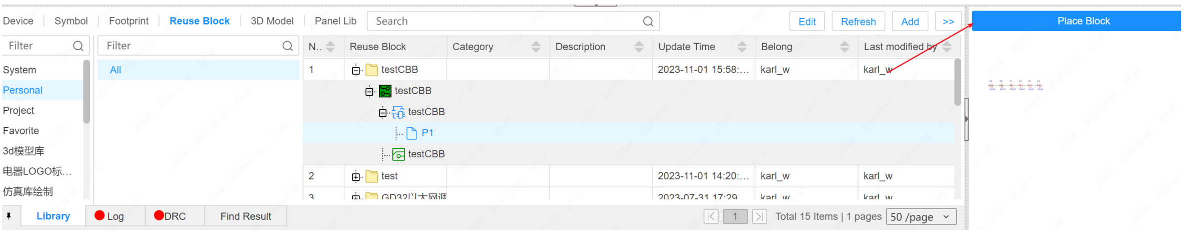
- Steps:The bottom panel - library - reuse block



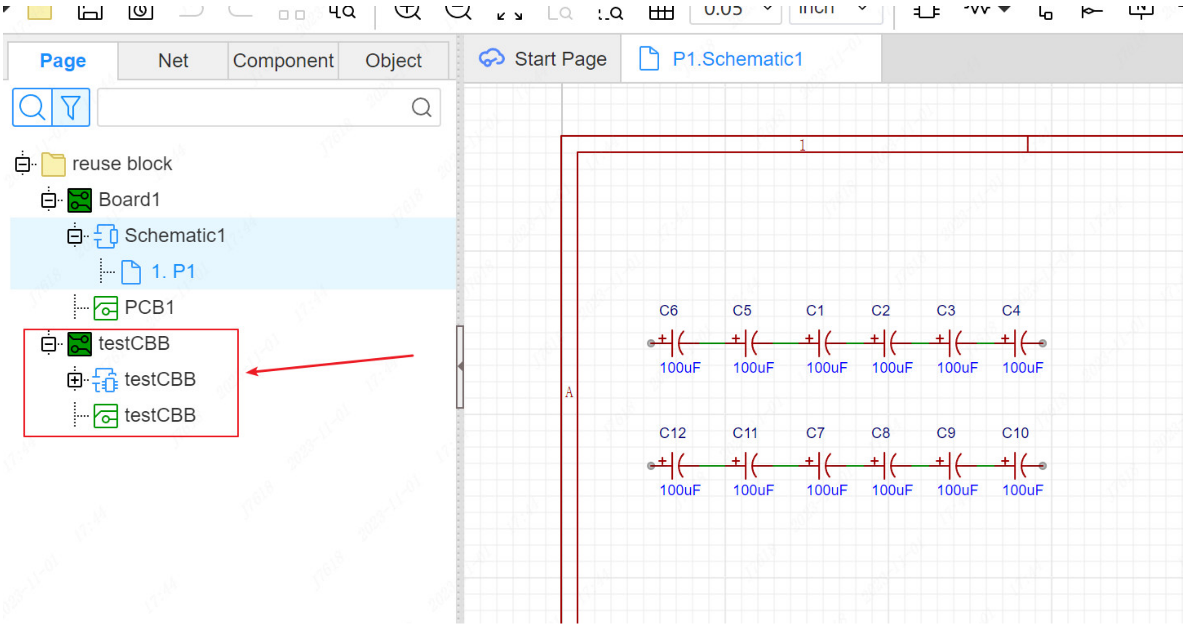
After placing the reuse block in the schematic diagram, transfer to the PCB, the PCB will automatically form a module based on the layout of the layout in the reuse module; it can also be directly placed in the PCB, but this method is placed in the schematic diagram and the PCB network.The table is not associated.

Schmatic Place

In the schematic diagram, you can directly place the reuse module, which are divided into two methods: symbolic method and combined method. These two methods will be displayed in the schematic graphic cloth, but when transferring to PCB, you can use it to the PCB, you can use it to the PCB.The entire PCB module is combined together to avoid re -layout of wiring operations and improve design efficiency.



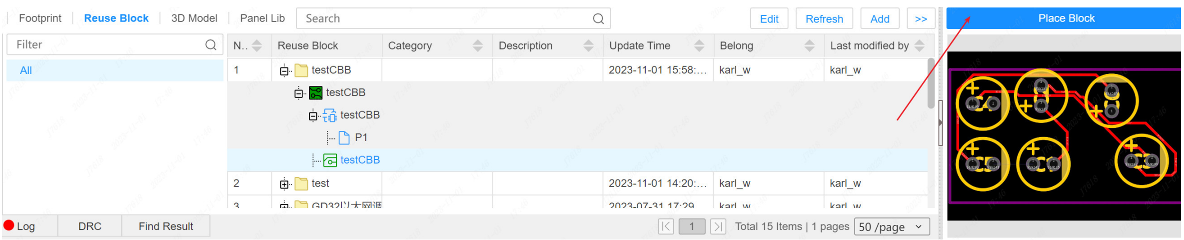
During the placement, the page of the pages of the left panel will automatically create the schematic diagram and PCB template under the reuse module project. The subsequent schematic diagram to PCB and the layout of the renewal template will be reused from this template.

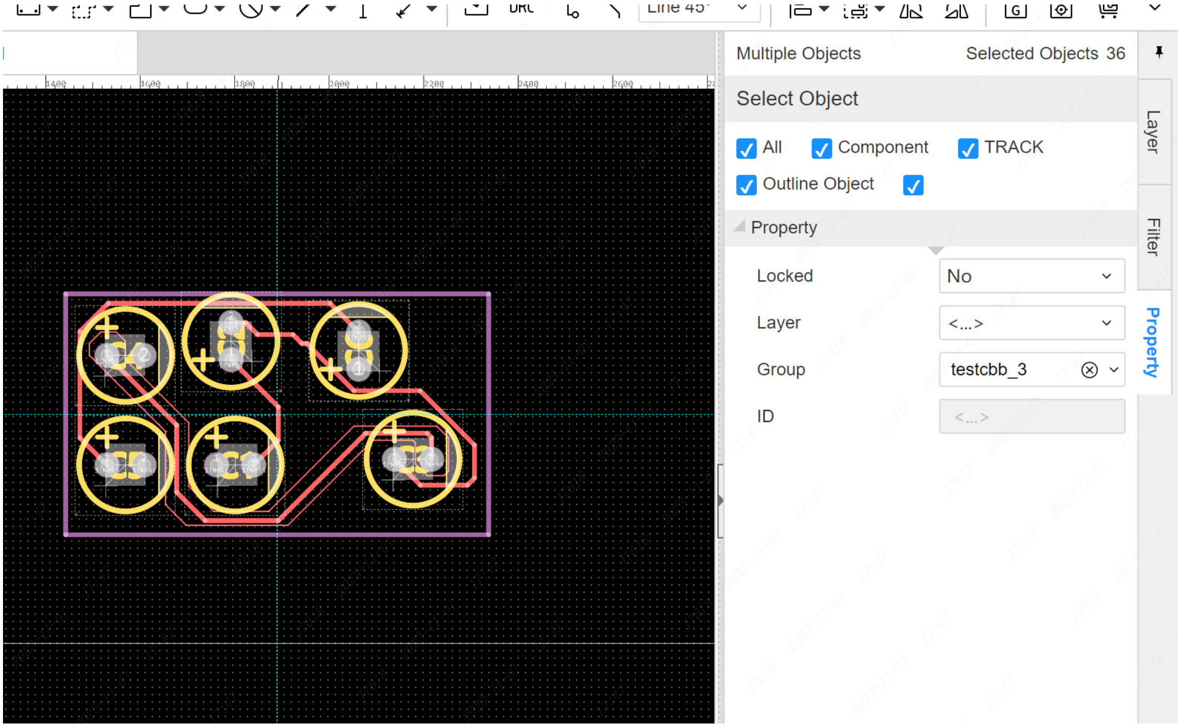


After the schematic diagram is placed, the different forms of different forms generated in the schematic diagram, please check in the reuse module-place the reuse module chapter.

PCB Place

The PCB under the module library in the PCB can also directly place the module. After clicking



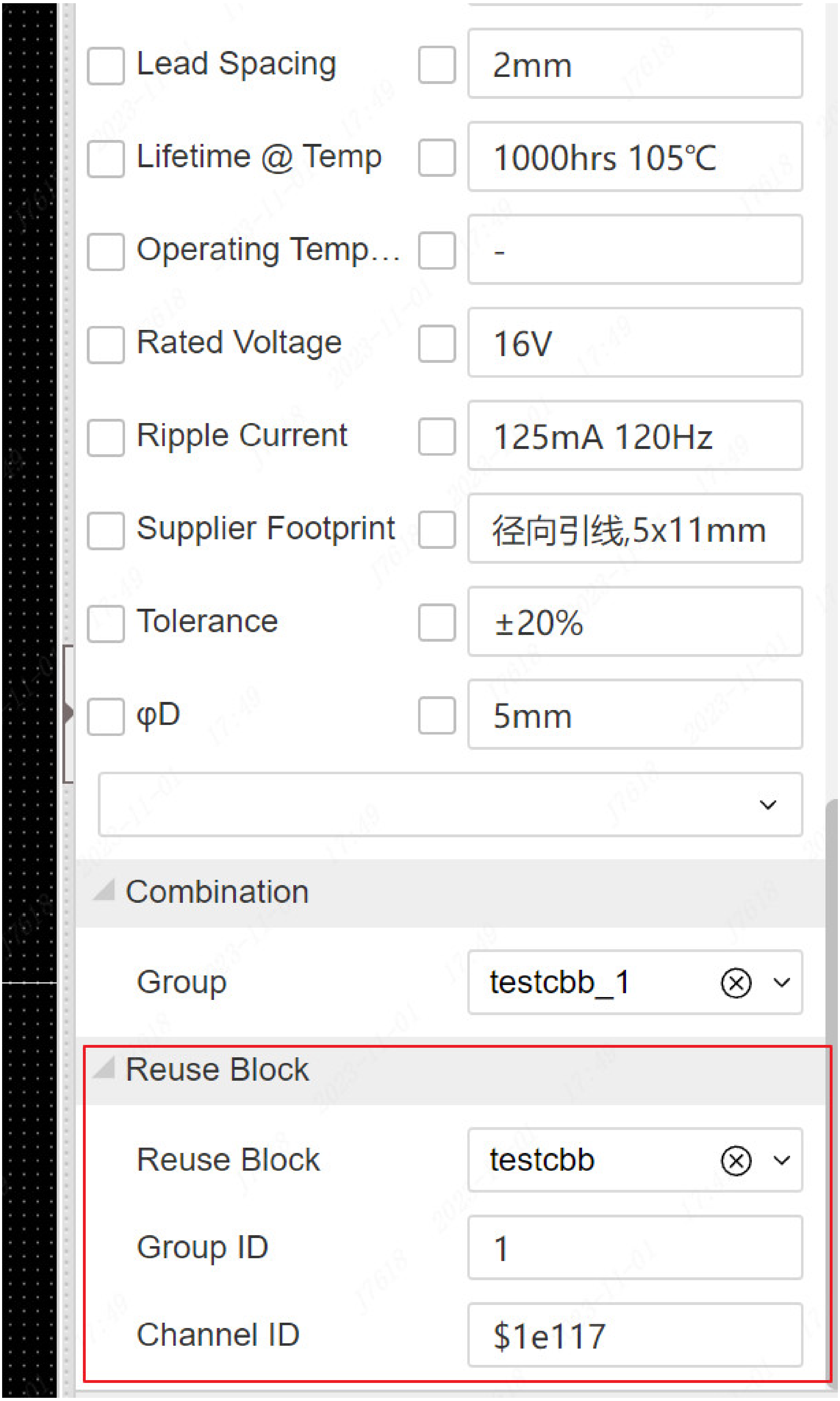


Update Place and Route From Reuse Template

After the layout of the PCB module is updated, it is very troublesome to modify it separately in each module in a PCB.PCB templates once, and then update to all the current PCB citations.

Reuse block properties

After the principle diagram and PCB are placed, each component in the module automatically generates the attributes related to the reuse module, which is used for the follow -up schematic diagram to the PCB automatic layout wiring and update the layout wiring from the template.

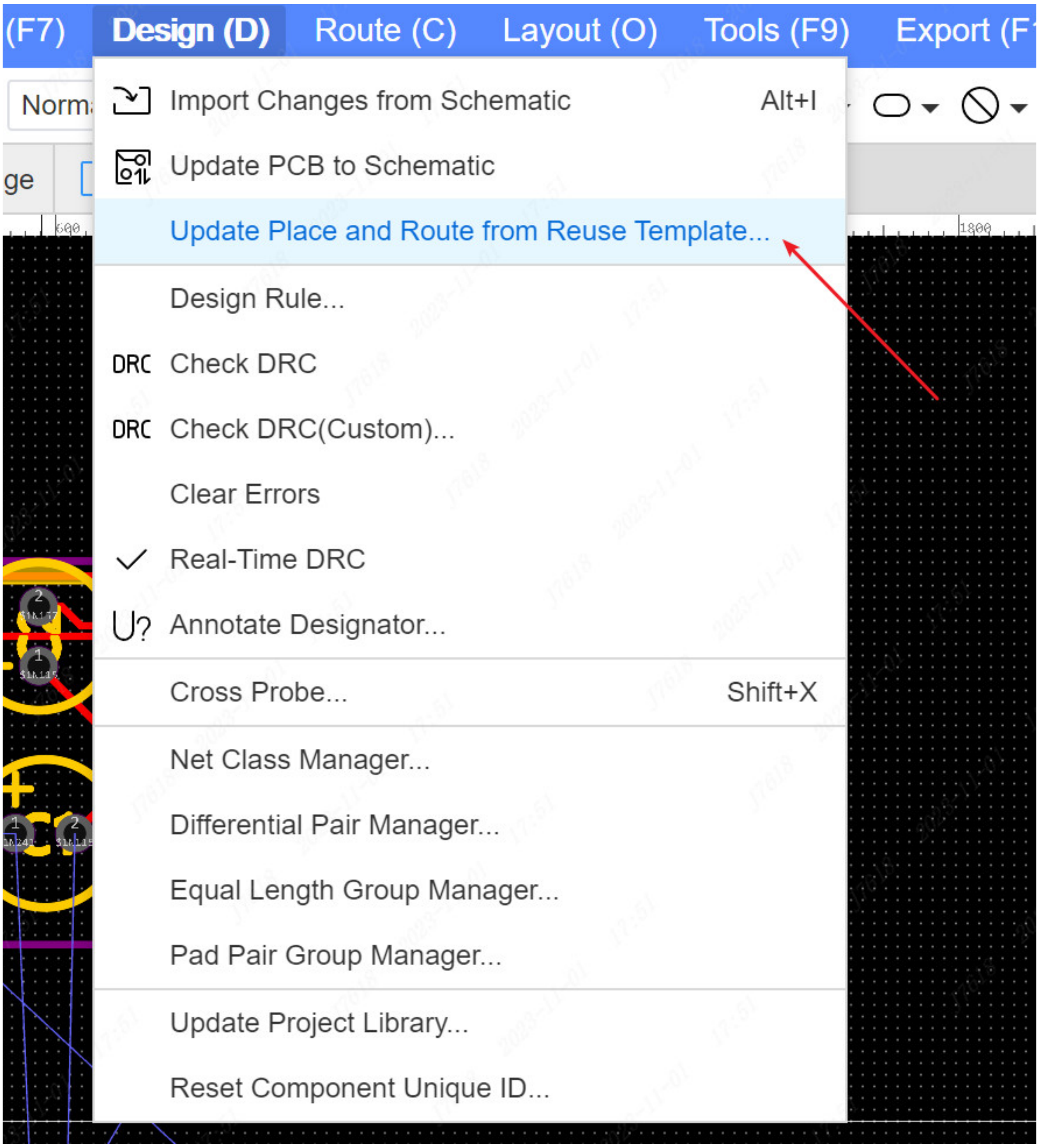


- Reuse block: The component with the same name of the reused block belongs to a template. The PCB of the same name in the left map tree is their template PCB

- Group ID: One module may be repeatedly used, and multiple groups will be generated. The group ID attribute is used to distinguish the current component belongs to that group
- Channel ID: There may be components that may be encapsulated by multiple same devices in a module. The channel ID is used to confirm the specific location of the component in the module

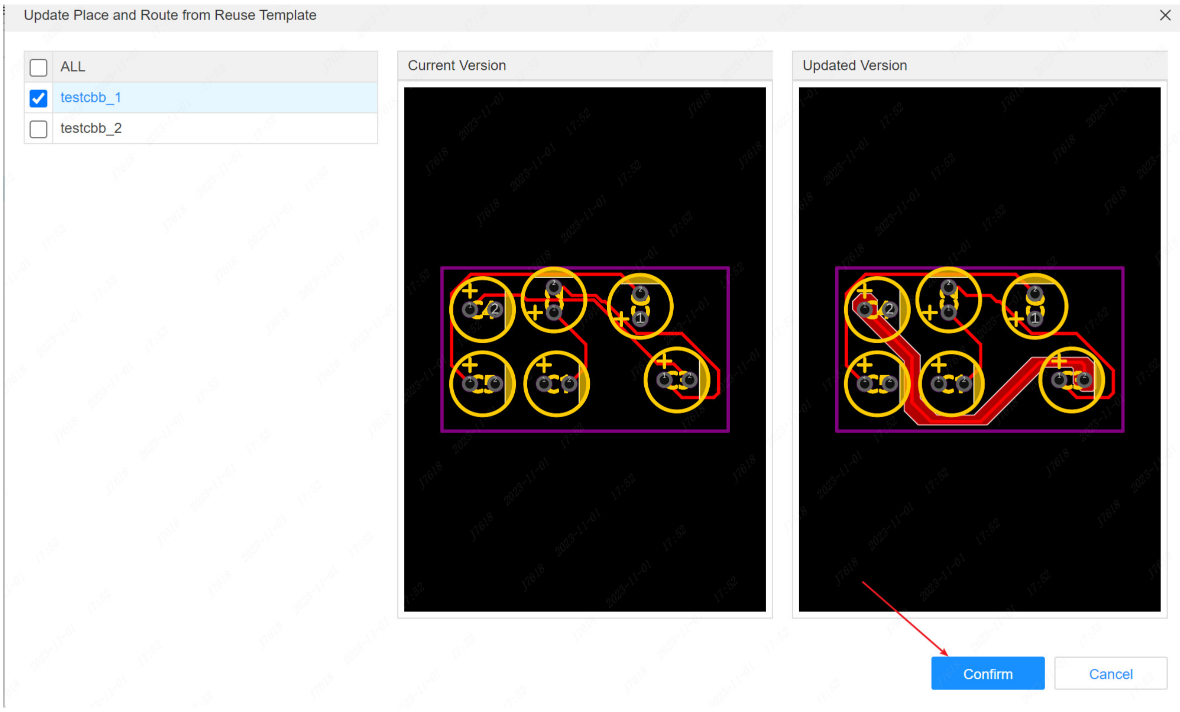
Update from template

Steps:Top Menu -Design -Reproduce template update layout wiring



From the pop -up layout of the reuse template, the list of the module+group ID of the module+group ID is listed on the launch of the layout layout. Select the corresponding module. You can preview the difference between the current version and the latest version of the template on the right side

Check the module that needs to be updated. Click to confirm that the corresponding module in the canvas will re -lay back to the latest template to lay out the wiring.

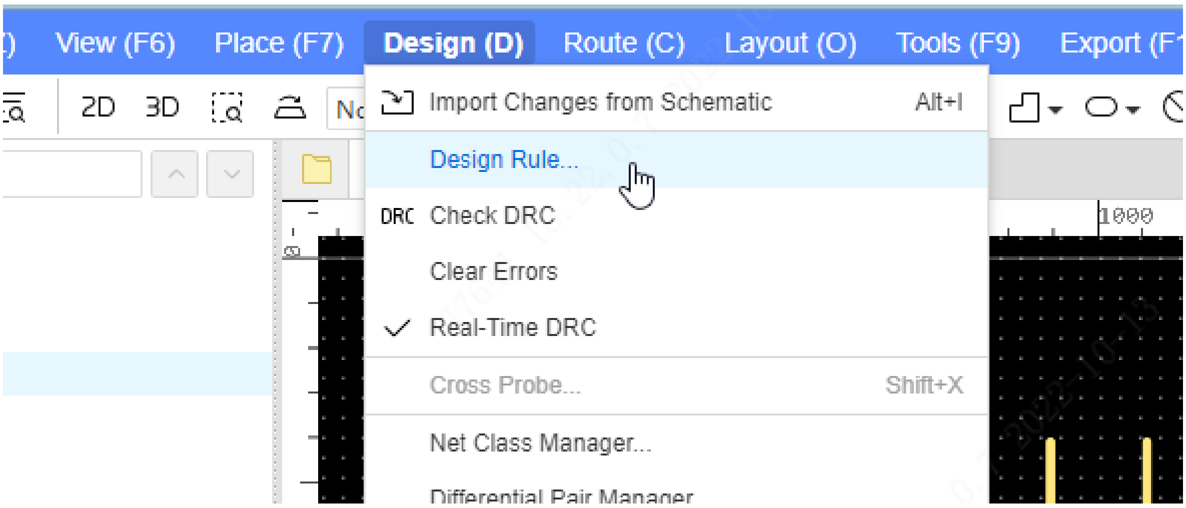


Design Rules

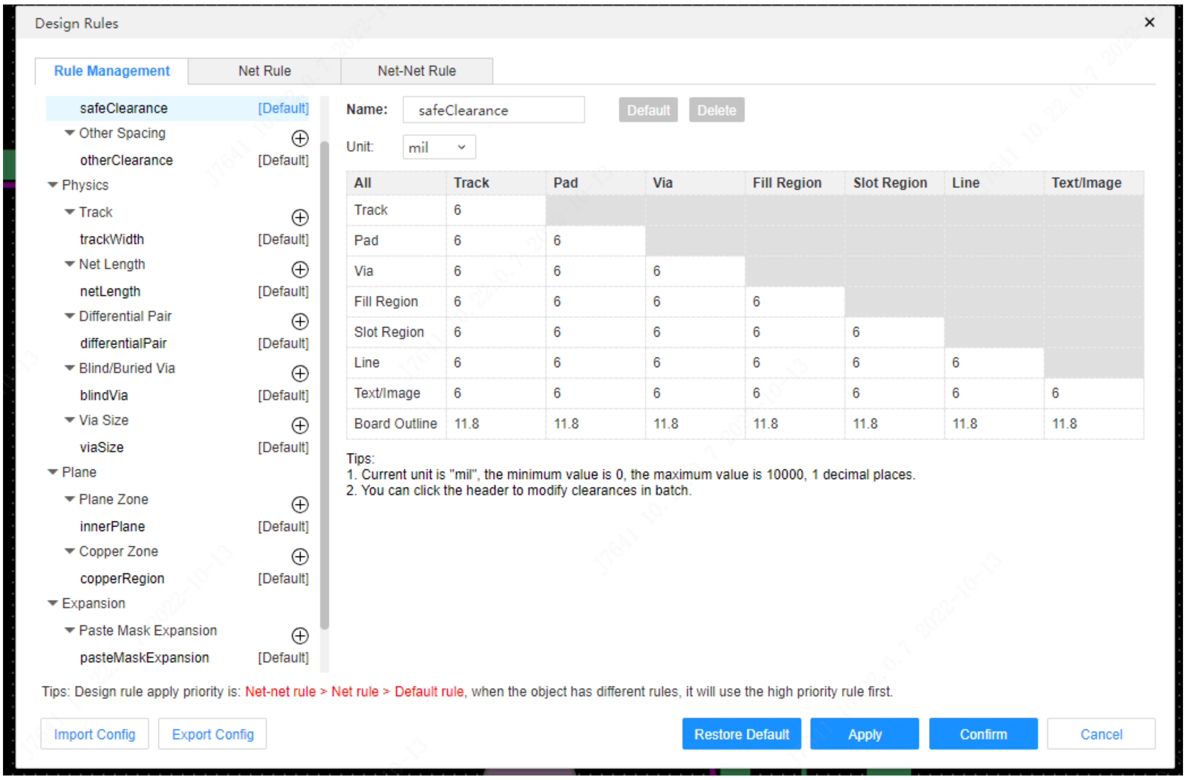
Design rules are used to set the basic design principles of the PCB. Entering a safety design rule in the design rules can ensure that there will be no design problems in the design of the PCB.

Steps:

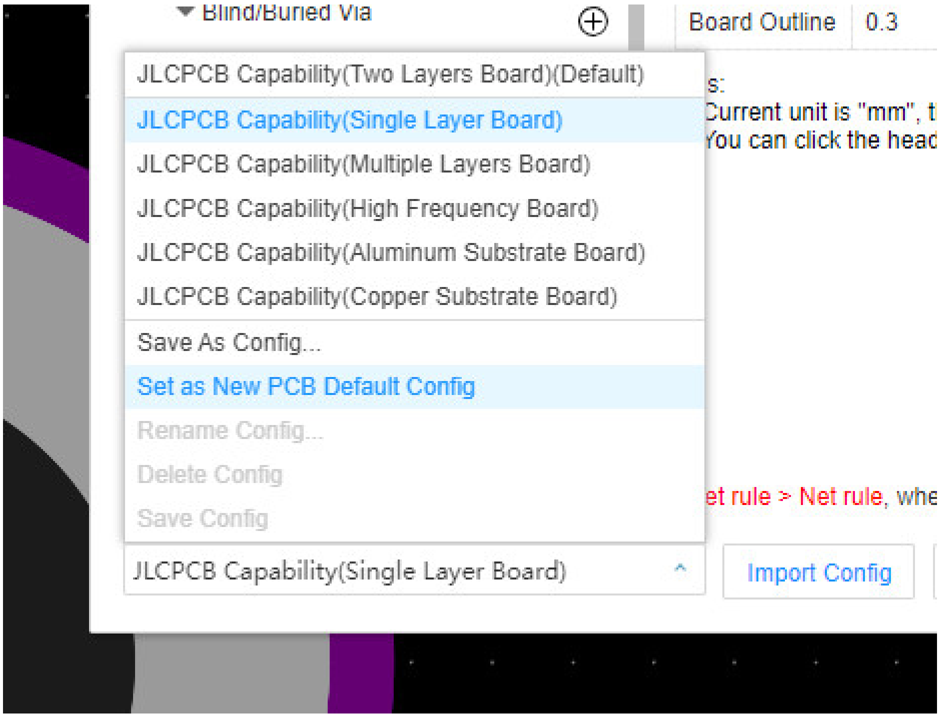
- Top Menu - Design - Design Rules



at Design Rule dialog, you can set many rules for the PCB

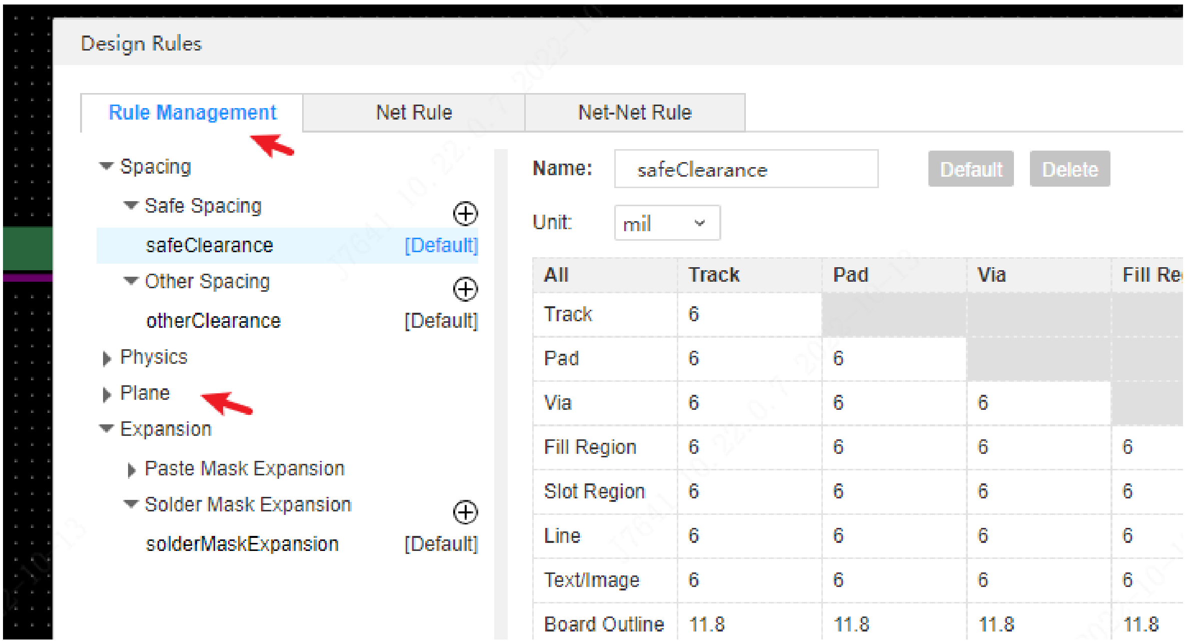


The lower left of the design Rules dialog box supports multiple preset process configuration switches



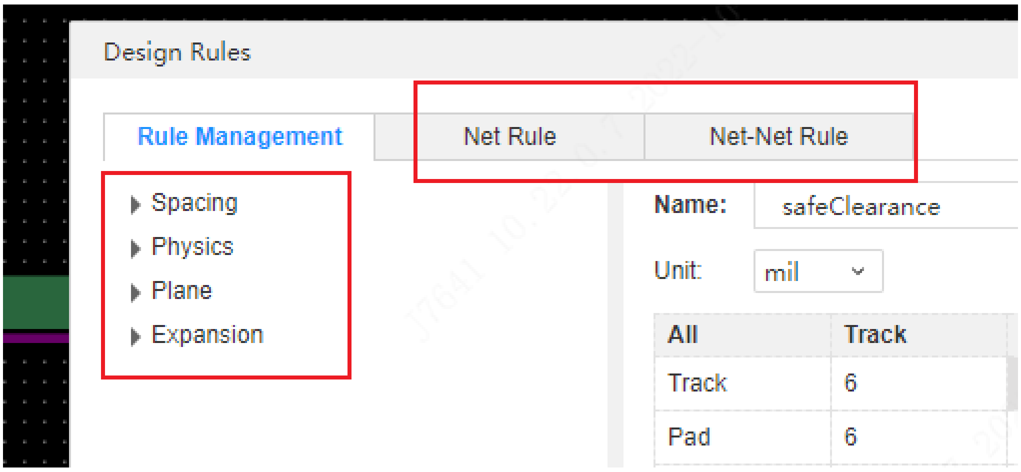
Rules management

Under the rule manager, you can add, modify, and delete rules under each type of rule. For networks without special rules, the default rules will be used.



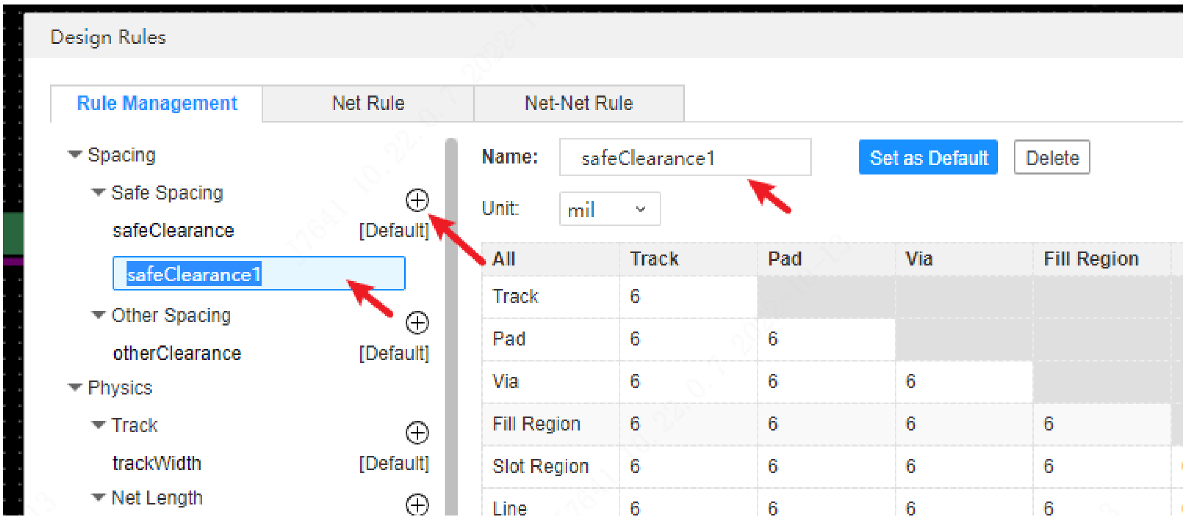
Rules type introduction

There are currently four categories, and one or more rules are created under each specific rule



rules editing

Click the + icon to the right of the rule type to be added to add a new rule. After entering the rule name, click the mouse outside the input box to create the rule successfully.

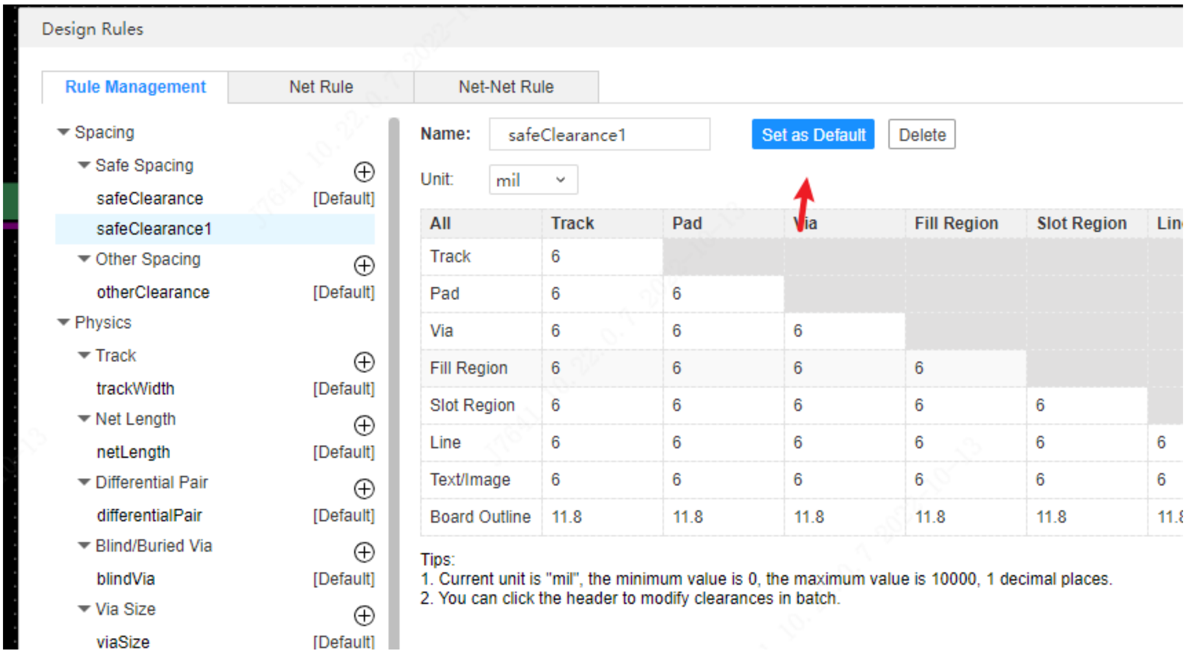


After adding a new design rule, you need to rename the rule. It should be noted that the rule name of the same type cannot be repeated.

1. Default rules

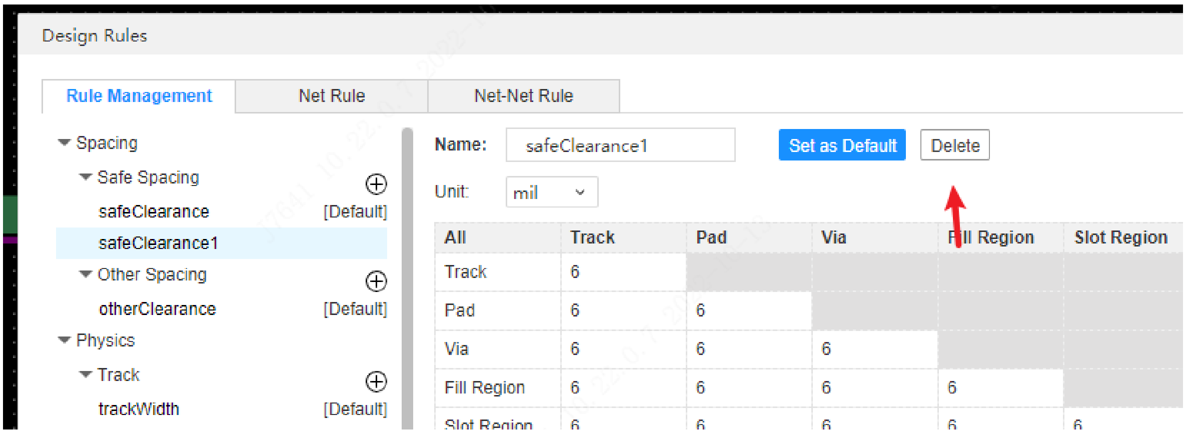
There is only one under the default rule type in the left column, and this rule will be pinned to the top.

If you want to set a rule as the default rule, click the Set as Default button in the rule view.



2. Delete rules

Non-default rules support deletion. Click the delete button in the rule view to be deleted to delete the rule.

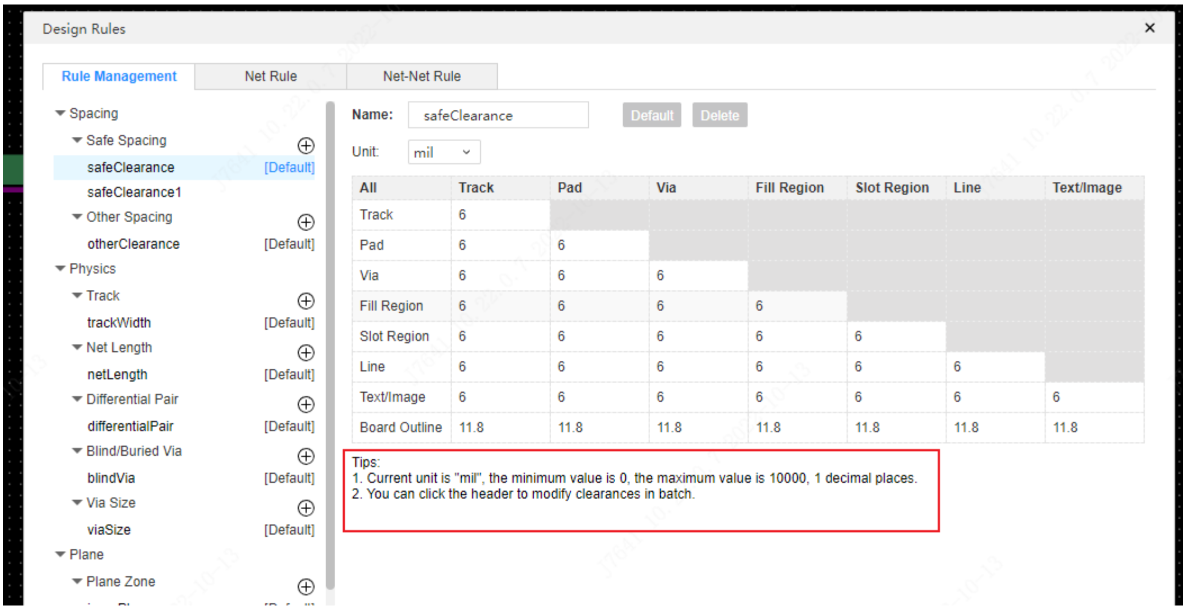


Safe spacing rules

The safe spacing table allows you to set spacing requirements between two different network elements.

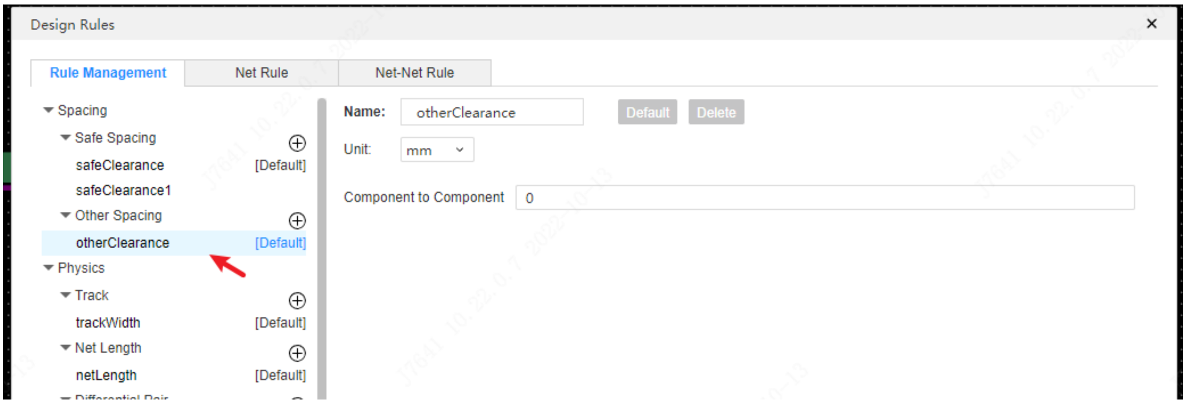
Double-click any table to modify the value of the rule.

Click the name at the top to edit values in bulk.



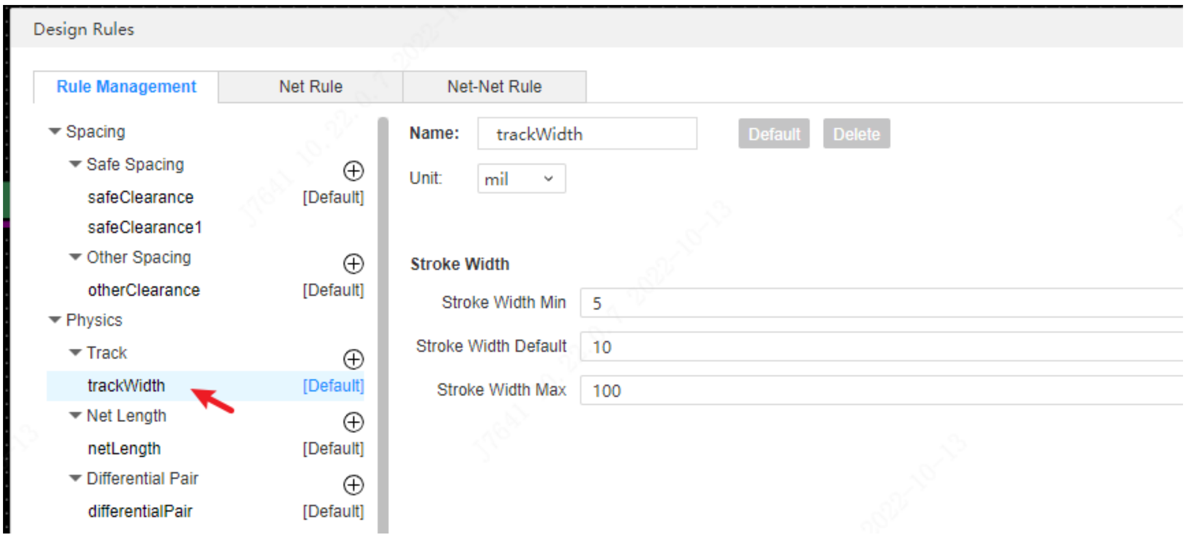
Other spacing rules

Currently, the device-to-device spacing rule is supported, which is detected by a rectangle surrounded by the overall outline of the component.



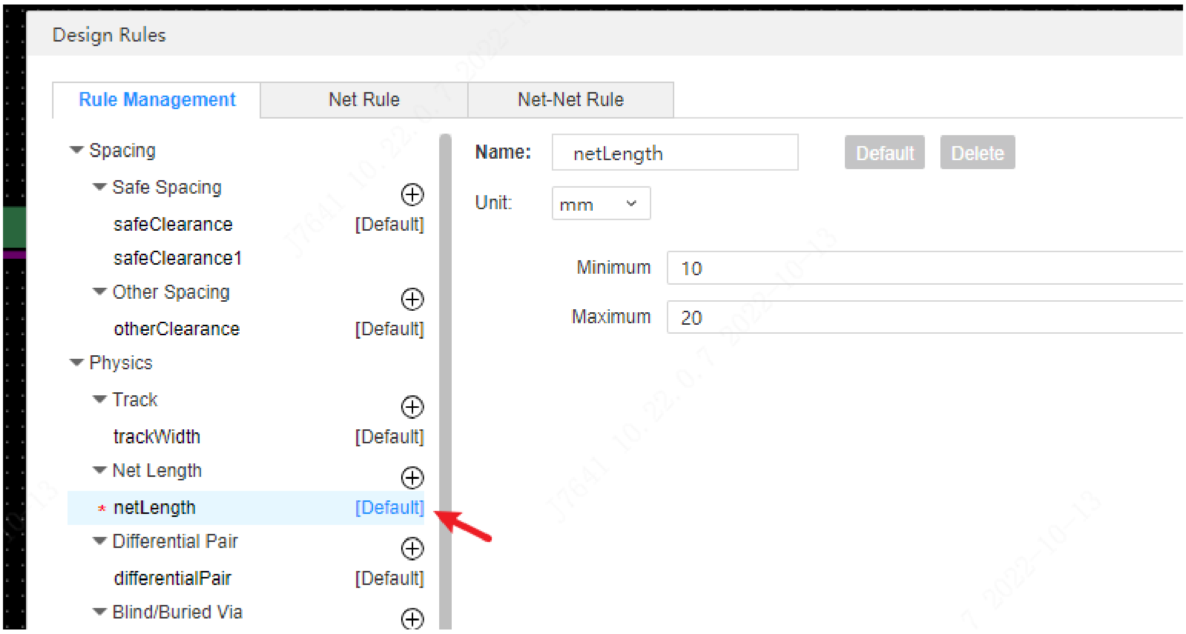
Track Rules

Minimum, default, and maximum line widths for wires can be set in a wire rule. If the line width of the wires in the PCB does not meet the range from the minimum line width to the maximum line width, it will be detected by DRC. The default line width refers to the default line width taken for each routing, but when routing starts from a wire, the line width of that wire will be used as the starting line width.

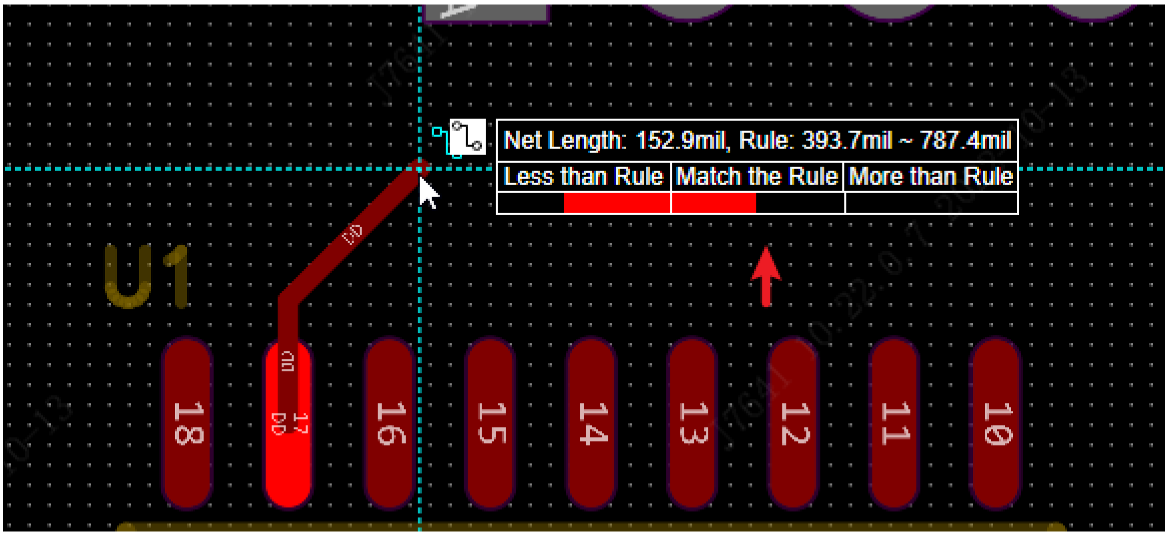


Network length

The net length rule is used to set the overall length of the checklist net trace. This rule cannot be applied to design constraints, but can be viewed during wiring.

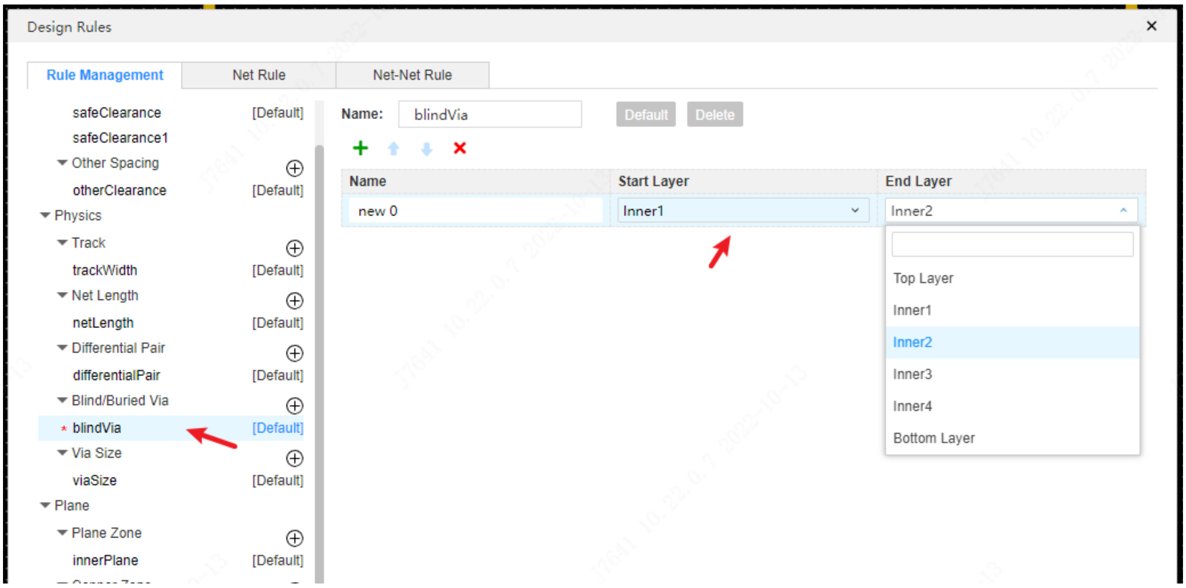


There will be an indication when the line meets the set rule length. In the case of conforming to the rule, the indication will be marked in green, and if it is larger or smaller than the rule, the indicator will be marked in red.



Blind and buried via rules

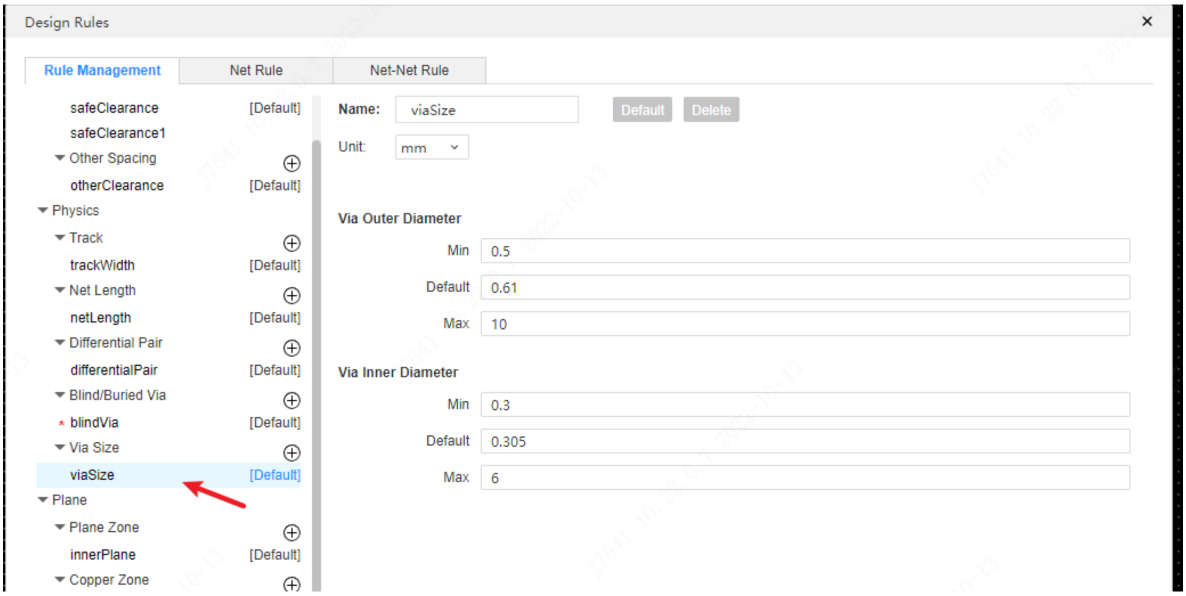
Blind and buried vias are usually used in the design of multi-layer boards. Before adding blind and buried vias, it is necessary to determine whether the designed PCB has been set to multi-layer. If it is not set, it needs to be set in the Tools - Layer Manager, to enable more copper layers.



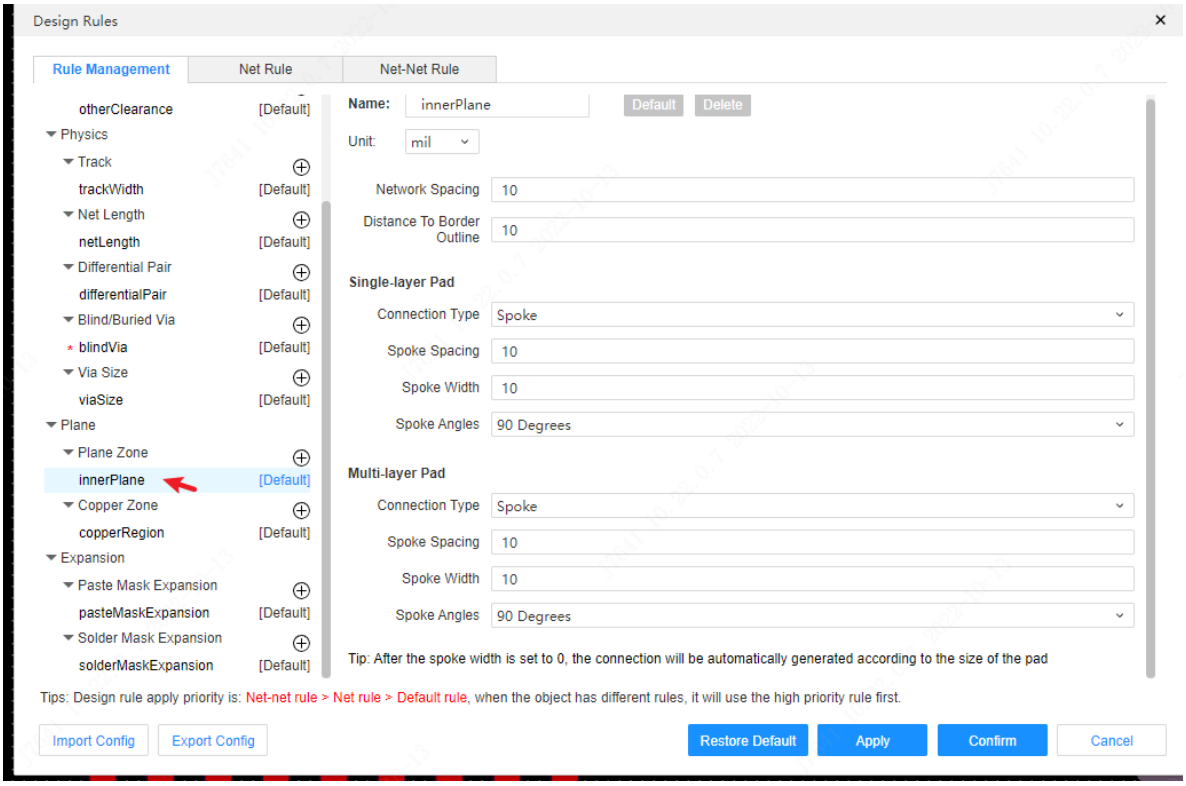
If the PCB needs blind and buried vias, it needs to be added in the design rules first.

Via Size Rules

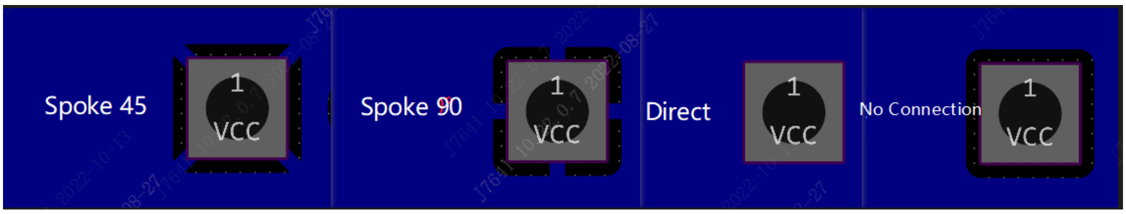
The minimum, default and maximum dimensions of the via OD/ID can be set in the via size rule. If the via size in the PCB does not meet the minimum to maximum range, it will be detected by DRC. The default aperture refers to the default size taken each time a via is placed.



Plane Zone Rules

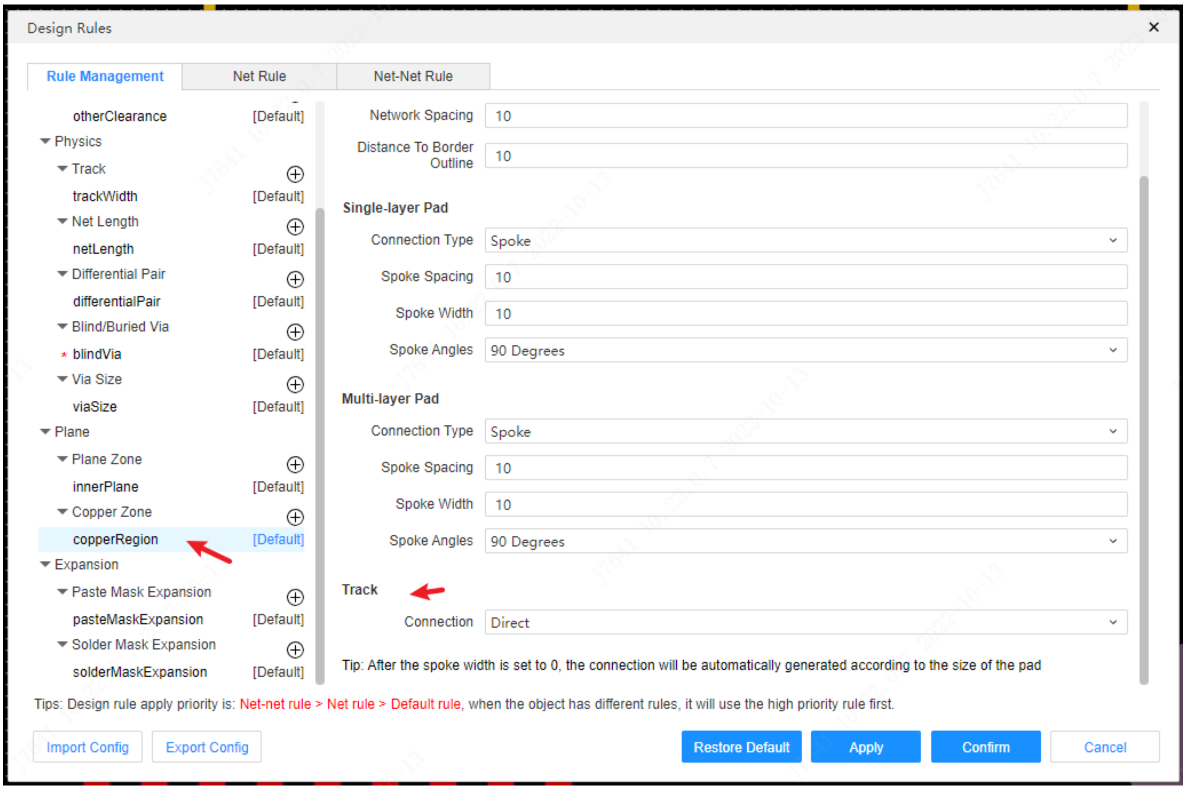


- **Network Spacing**: Set the spacing between the copper filling and different network elements when placing copper.
- **Distance To Border Outline**: Set the spacing from copper filling to border and slot area.
- **Pad Connection Type**: It is divided into divergent and direct connection. When set to divergent, divergent line width and divergence spacing can be set respectively.
- **Spoke Spacing**: Set the divergence spacing between the pad and the copper.
- **Spoke Width**: The setting of the divergence distance between the copper skin of the connection pad wire.
- **Spoke Angles**: Supports the setting of the angle of the inner copper layer connection method.



Copper Zone Rules

The copper laying rules are the same as the plane zone rules, and will not be repeated here.

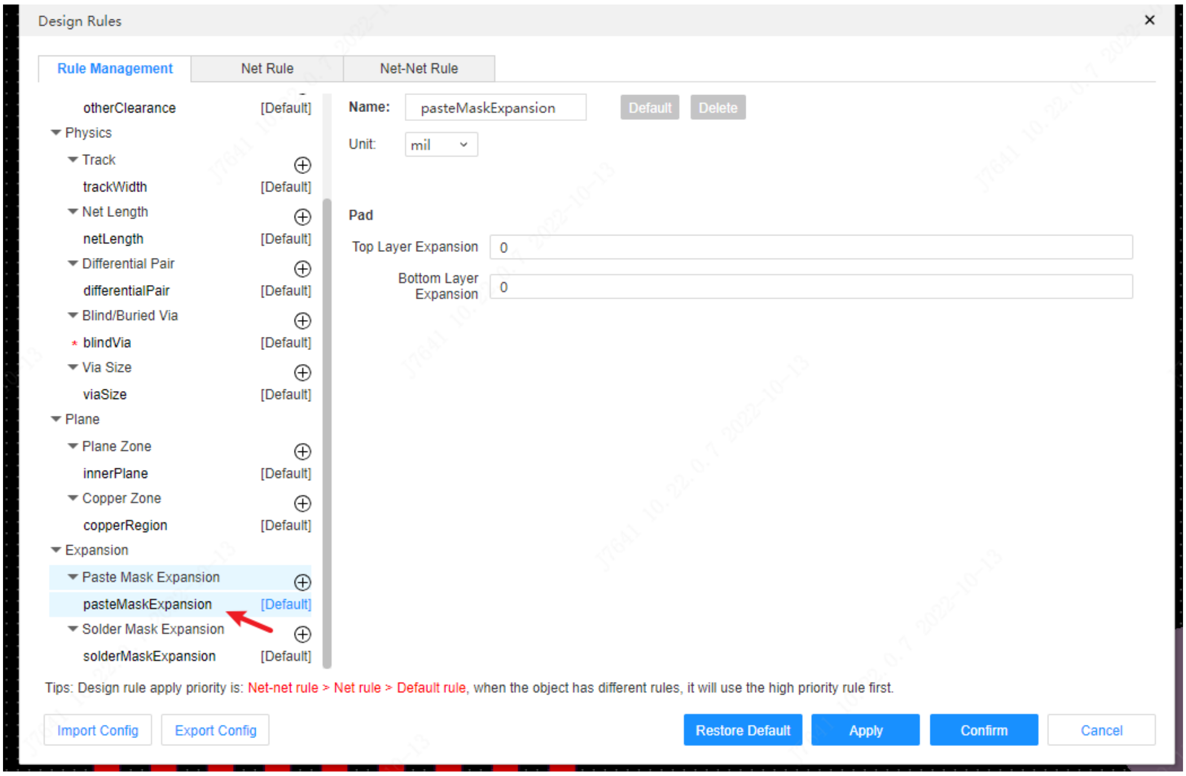


Note: When the copper-plated pad divergence line width is set to 0, the connection line width will be automatically generated according to the size of the pad.

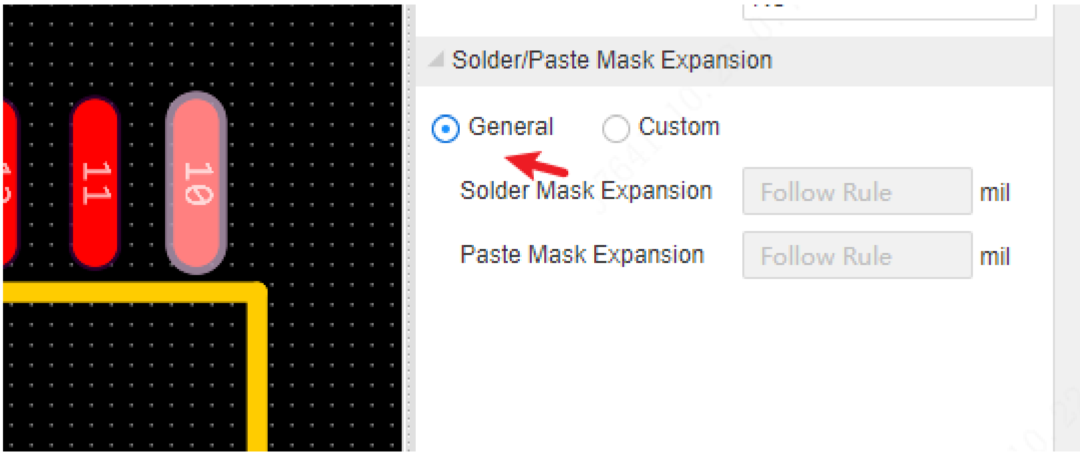
Compared with the internal electrical layer rules, copper laying has an additional wire connection method setting, which supports setting it to no connection. When set to no connection, the copper will not be connected to the wire, and it will not appear that the wire is covered by the copper.

Paste Mask Expansion Rules

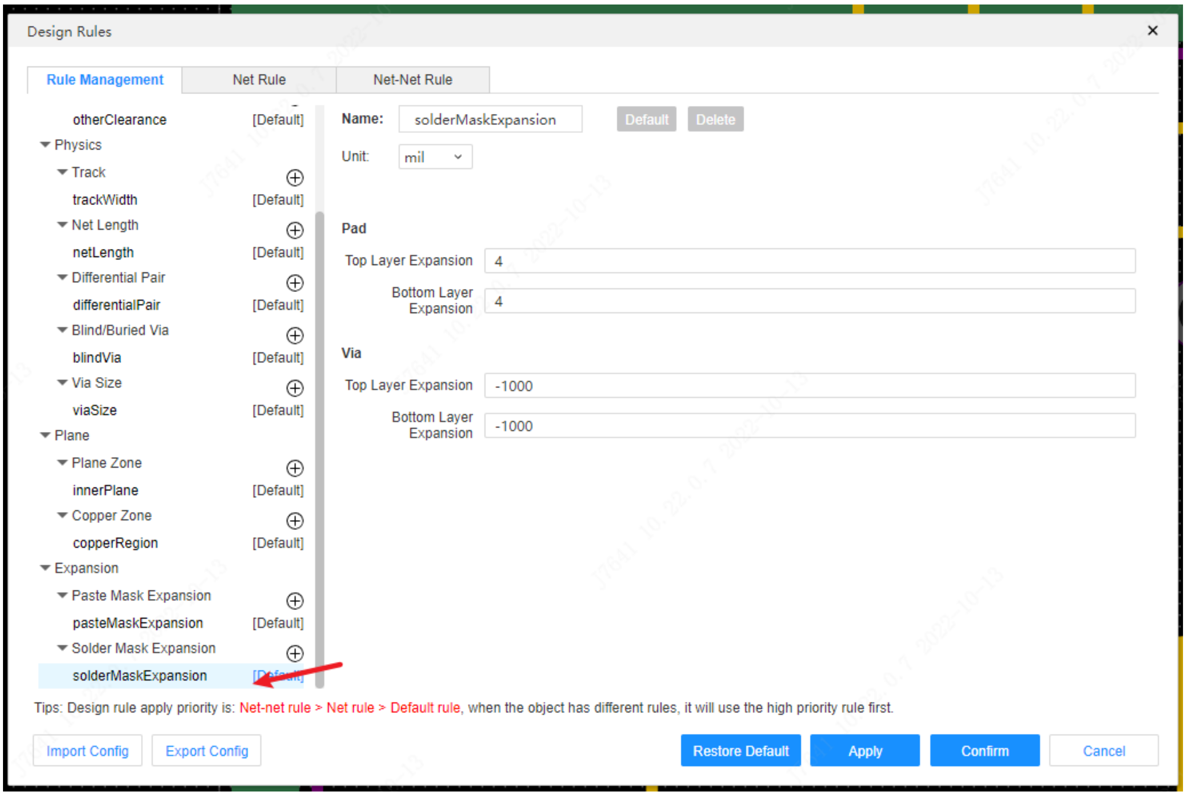
The paste mask expansion is used for the pad of the SMD footprint. When the paste mask expansion property of the pad is set to general, the mask expansion in the design rule will be used as its flux.



when you select a pad at PCB, you can set its solder or paste expansion value



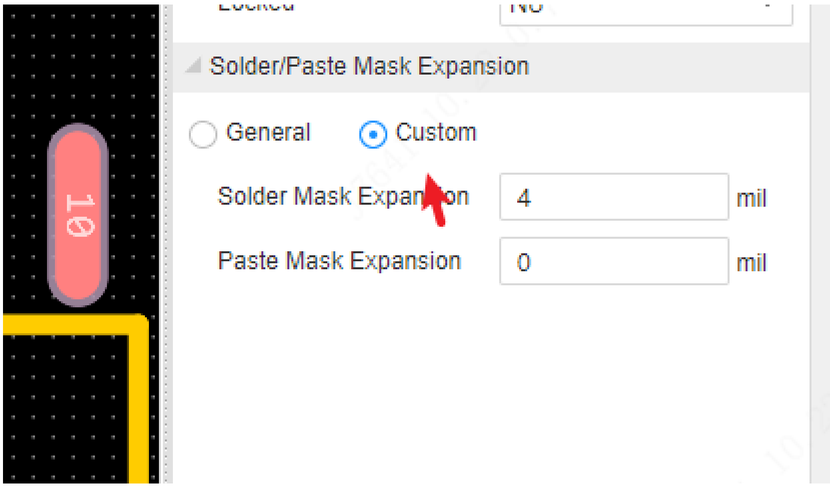
Solder Mask expansion Rules



The solder mask expansion of pads and vias can be set separately in the solder mask expansion. If you do not want to open the vias, set the solder mask expansion data to a number smaller than the aperture (such as -1000mil)

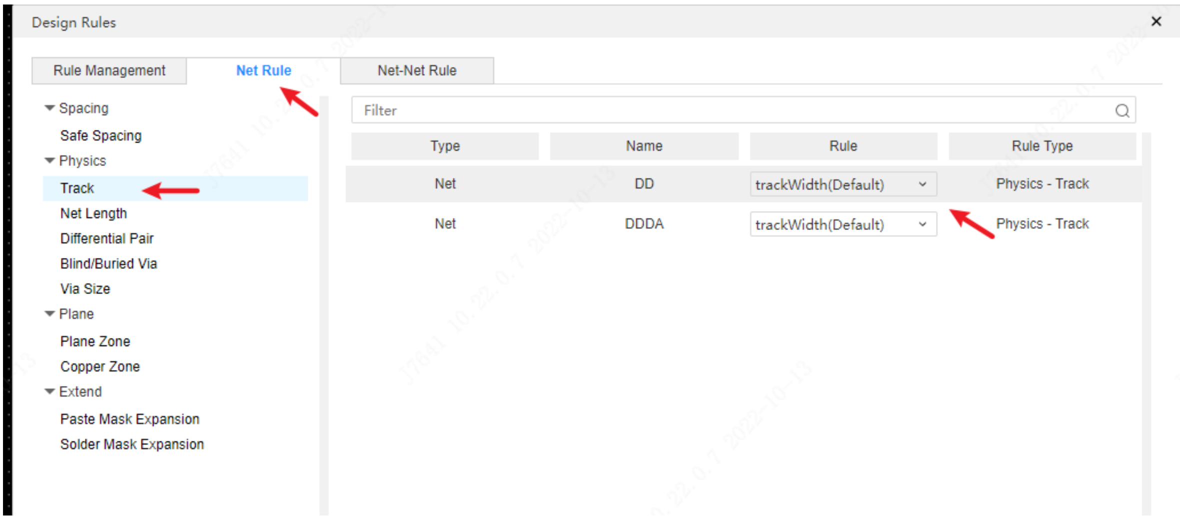
Similar to mask expansion, the solder mask expansion of the default pads and vias follows the rules

If you need to customize the solder mask of a pad or via, you need to modify its solder mask expansion parameters in the properties panel

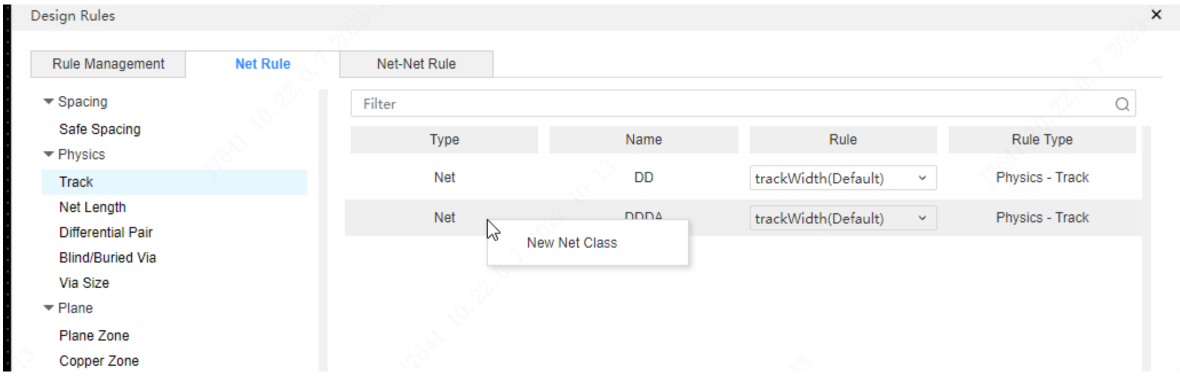


Network Rules

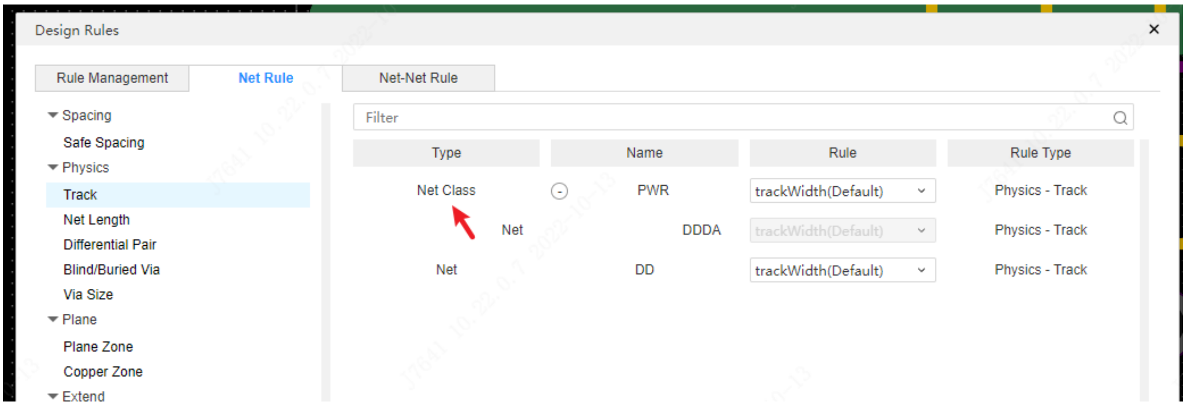
In the net rules, you can assign rules to all nets of the current PCB. First, select the type of rule you want to assign in the left tree, and then view the network list in the right view. By default, all networks are under the default rules. If you need to modify the rules of a network, directly in the drop-down box of the corresponding network rules Just switch.



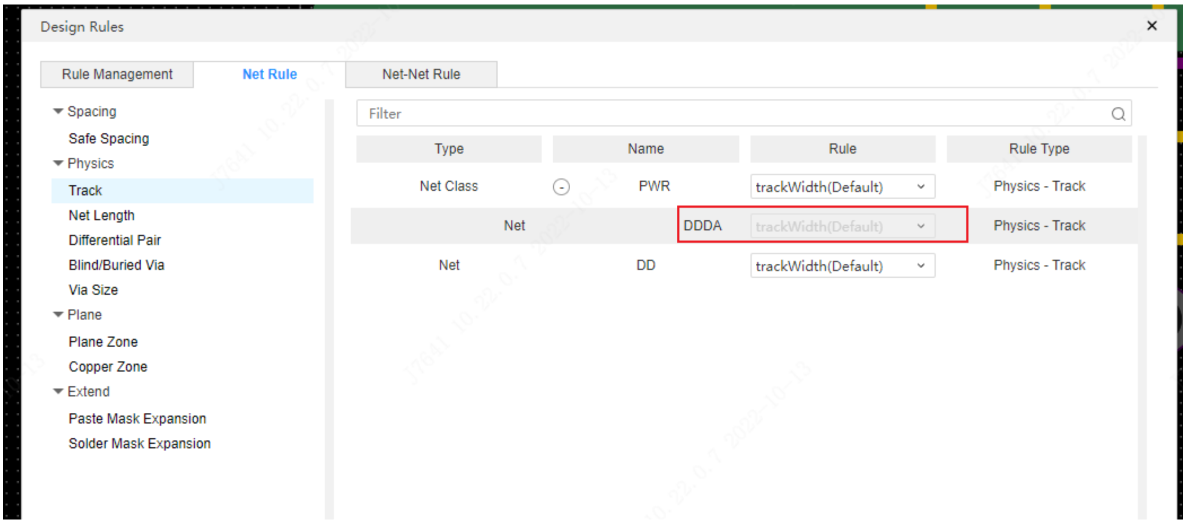
In the network rules, the function of quickly creating a network class and adding a network class is provided. Right-click in the network list and select the function of creating a new network class to create a new network class.



After the new one is created, the network class will be pinned to the top.



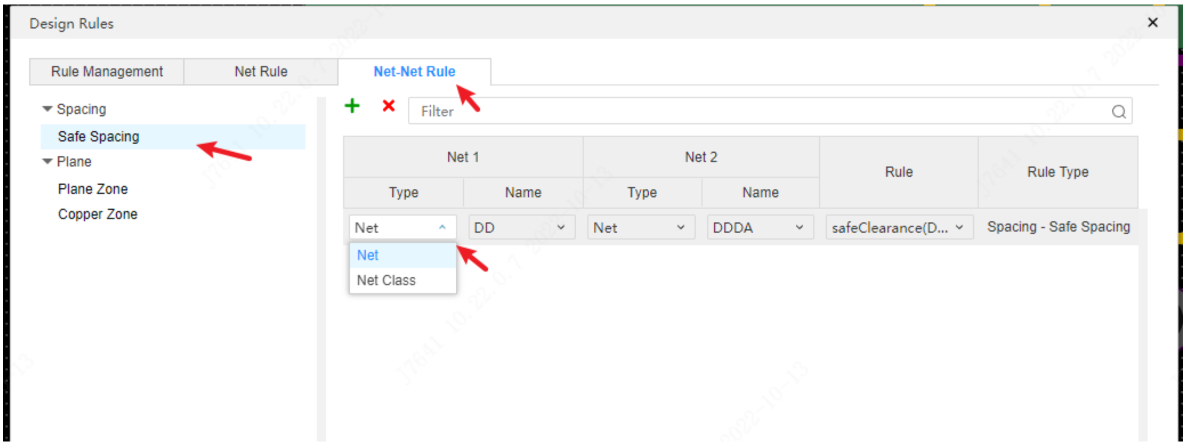
Assign a design rule to a net class, and all nets under the net class will become this rule



Network - Network Rules

Supports setting rule constraints between two different network rules, and supports rule settings for safe spacing, internal electrical layer, and copper laying.

Networks, or network classes, can be set up and assigned different rules.



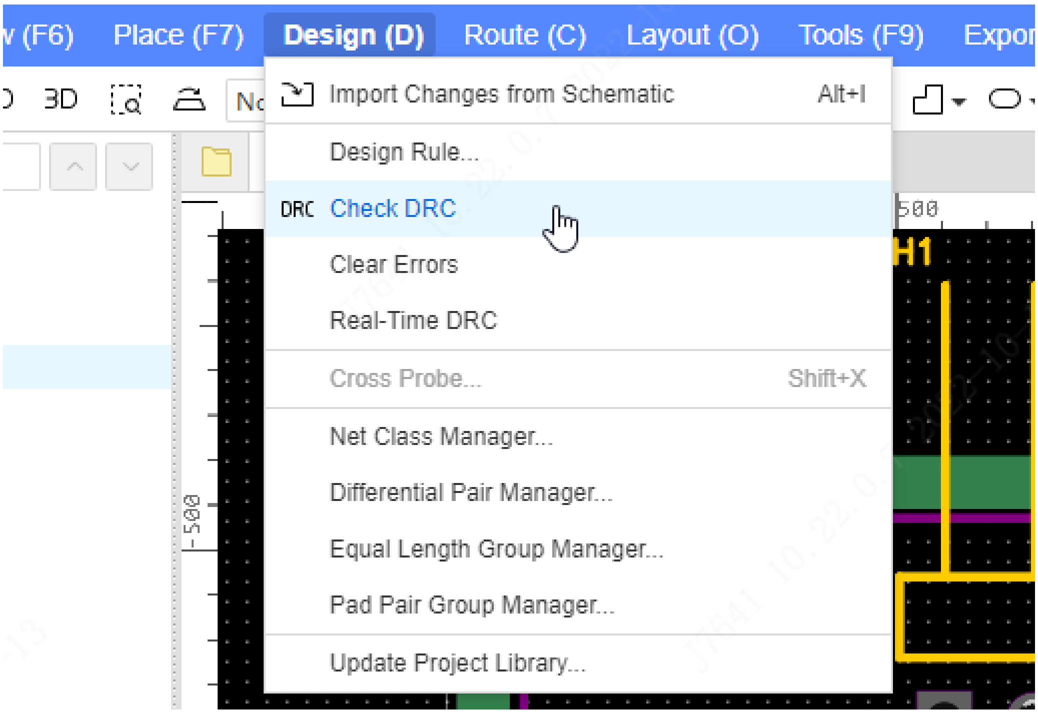
Check DRC

The purpose of the DRC inspection is for a general inspection after all PCBs are drawn. After designing a PCB, you need to check the DRC rules on the PCB. The DRC check is carried out according to the rules set by yourself. For example, the minimum spacing set by yourself is 8mil, then in the actual PCB, if the spacing is less than 6mil, an error will be reported.

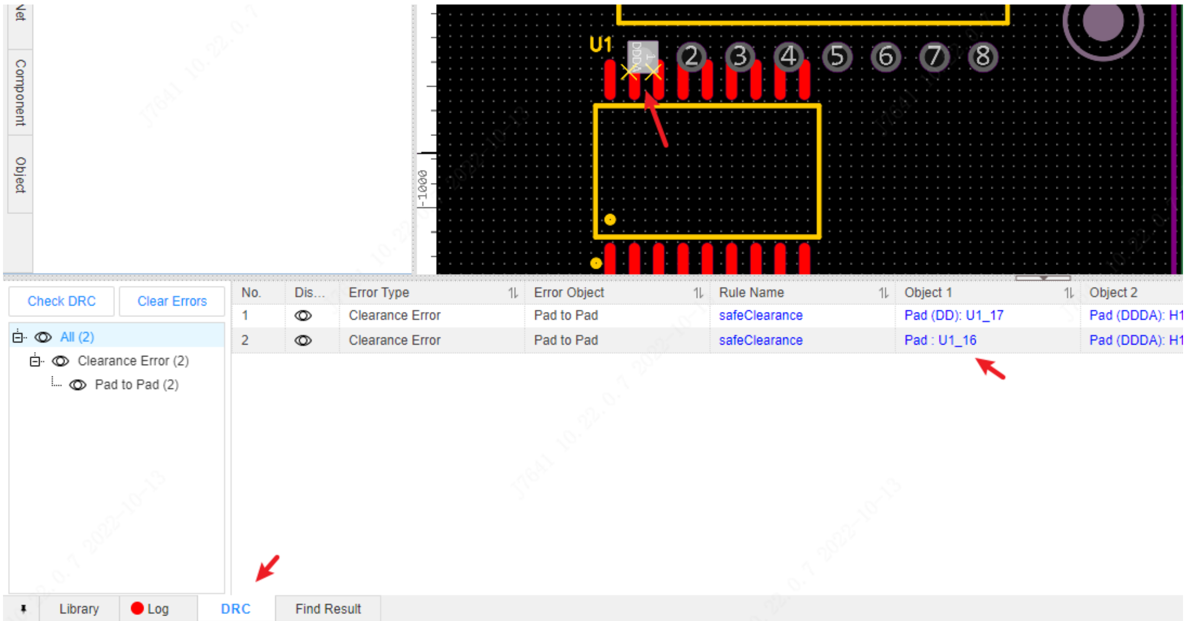
It is not that boards with errors in the DRC cannot be used. Some rules can be ignored. For example, errors in silk screen printing will not affect electrical properties.

Steps for usage:

- Top Menu - Design - Check DRC



The checked DRC results are displayed on the DRC panel at the bottom, and the corresponding PCB will also have a X symbol. Select and click the wrong point with the mouse to highlight it on the PCB, double-click to zoom in and locate the wrong point.

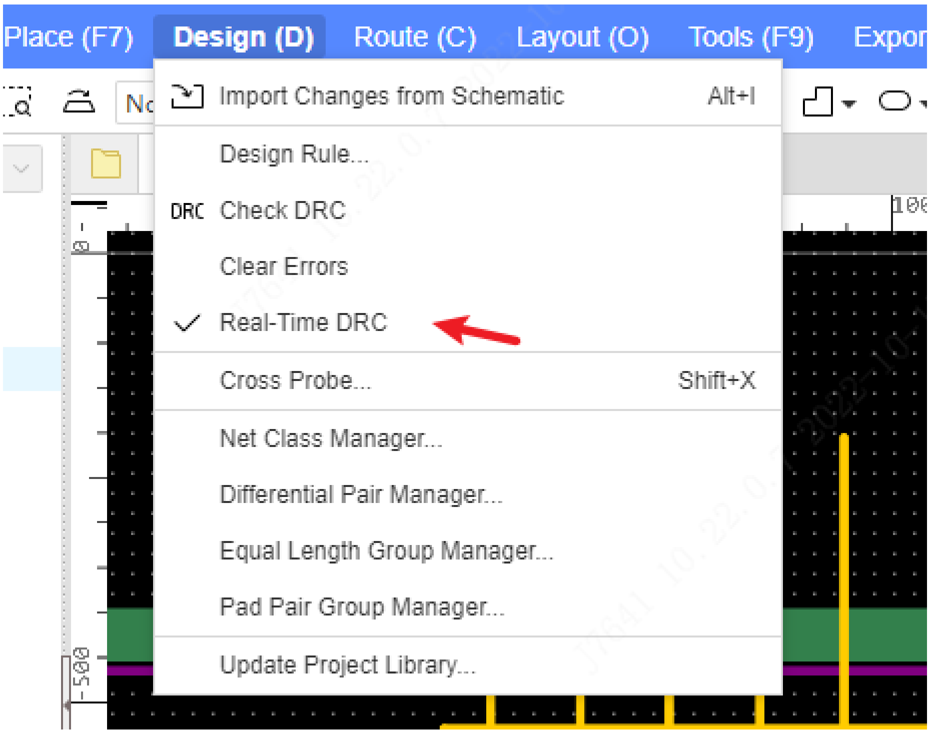


Real-time DRC

When real-time DRC is turned on, errors can be checked while drawing the PCB.

Steps:

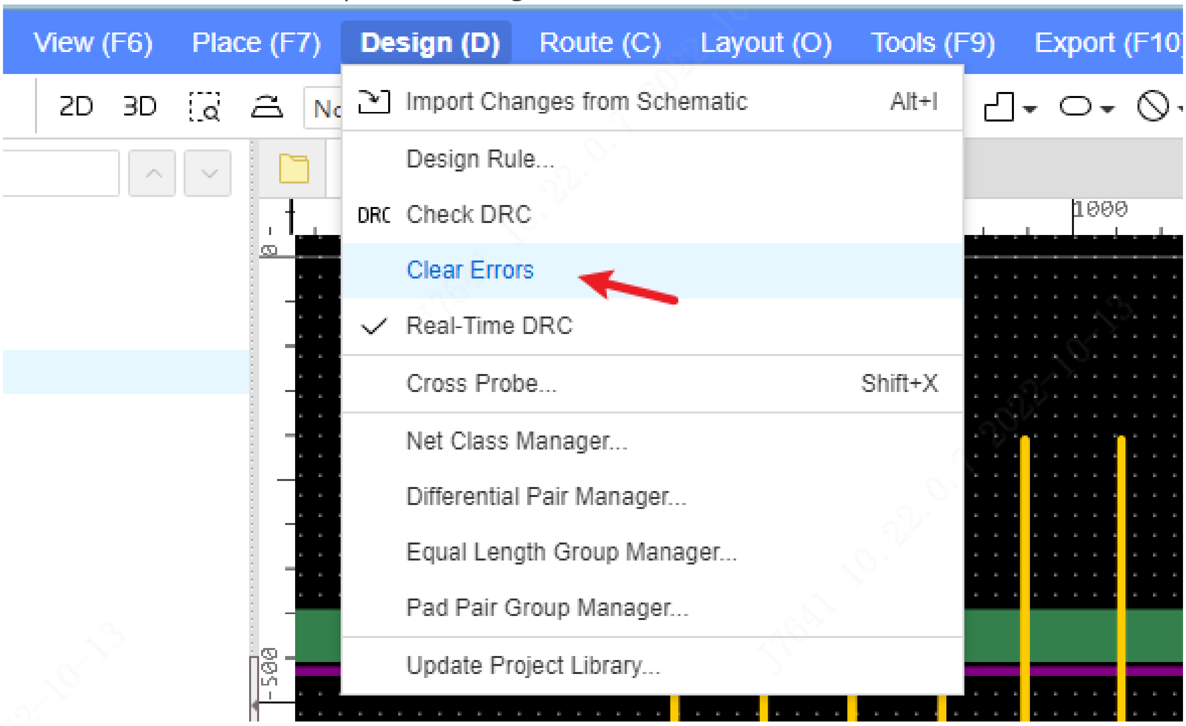
- Top Menu - Design - Real-Time DRC



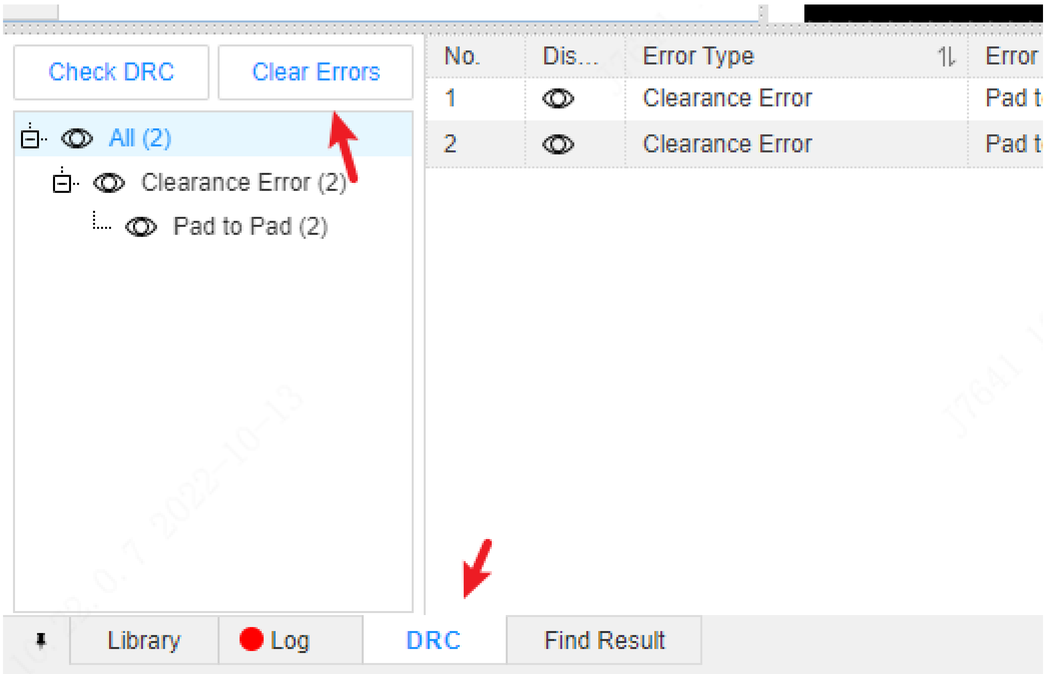
When drawing the PCB in violation of the rules, the real-time DRC will prompt an error in the PCB.

Reset DRC Error

Reset the DRC check. Via: Top Menu - Design - Clear Errors



or Bottom Panel - DRC - Clear Error resets the checked DRC.

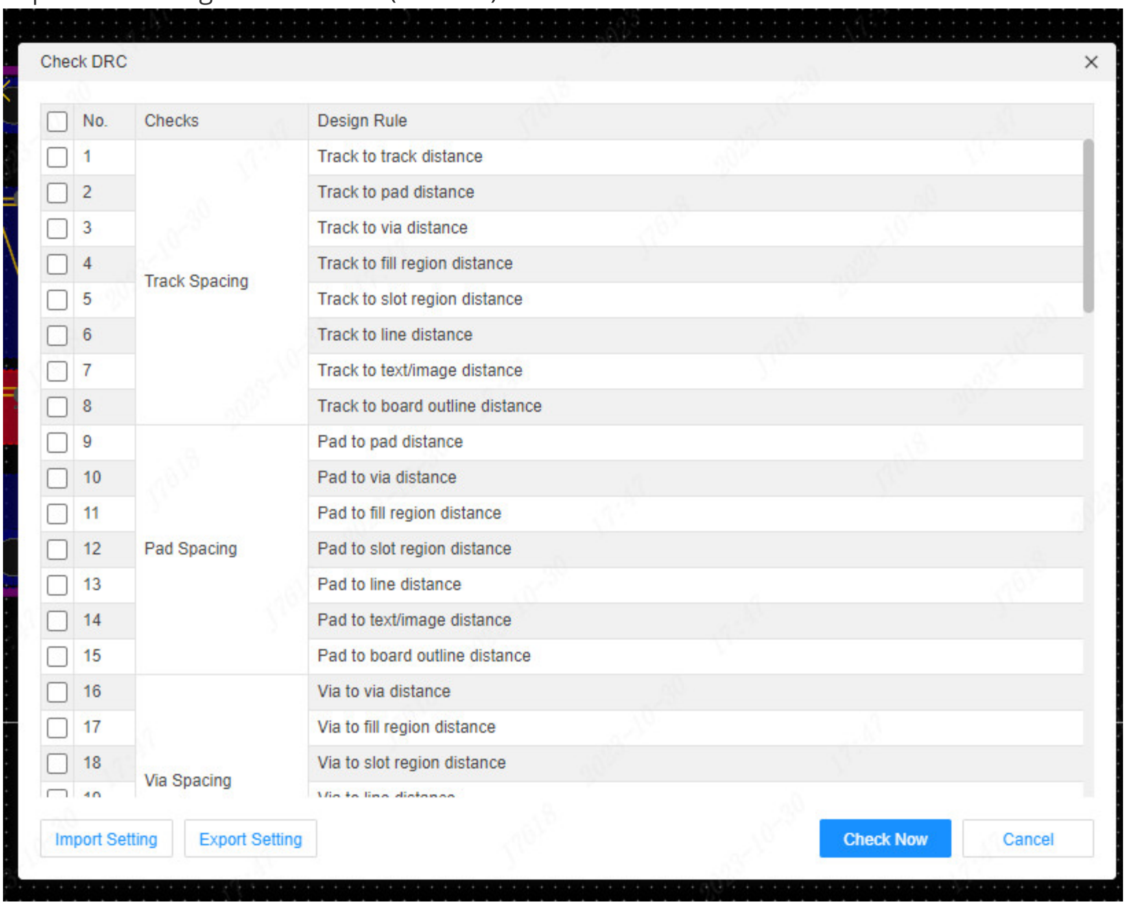


Check DRC(Custom)

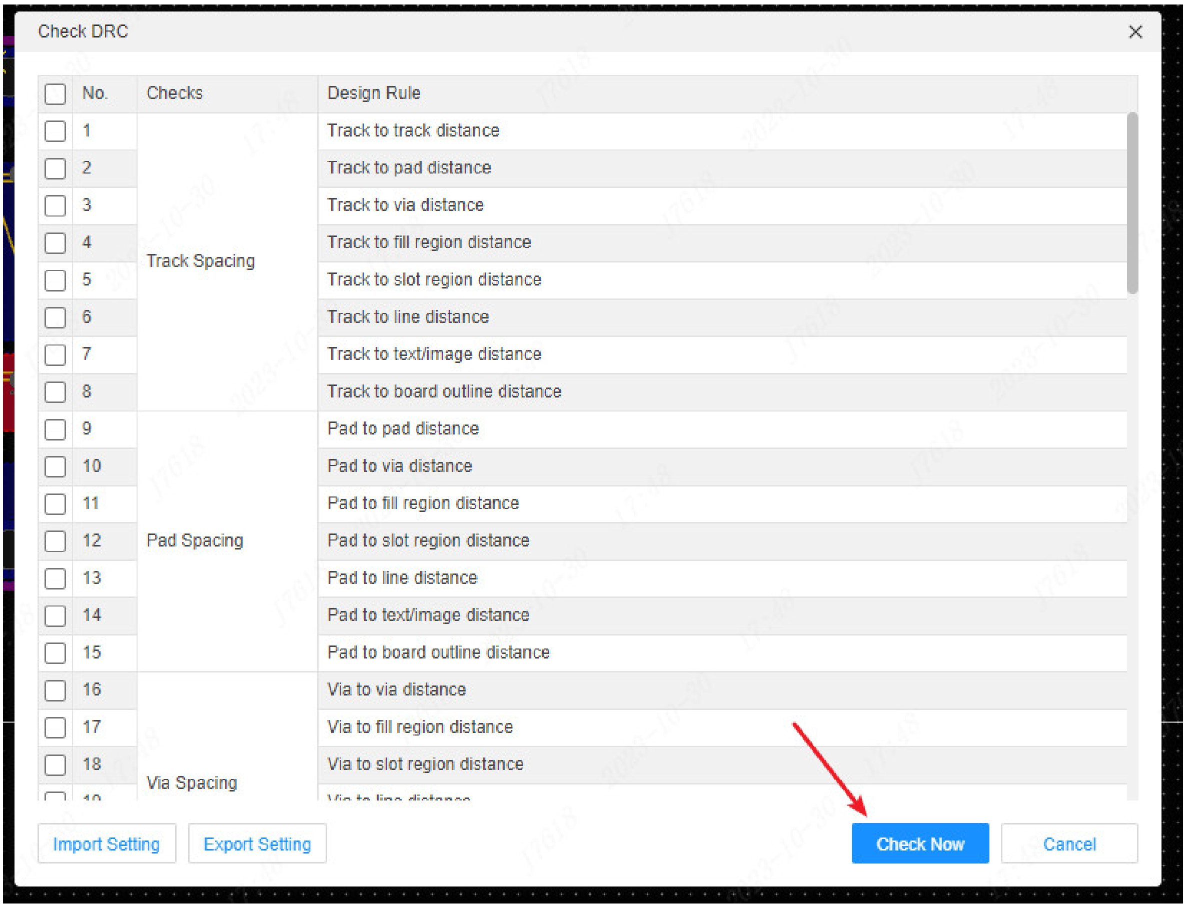
Check DRC (Custom) function to set the DRC project that needs to be checked by yourself, and check the DRC targeted manner.

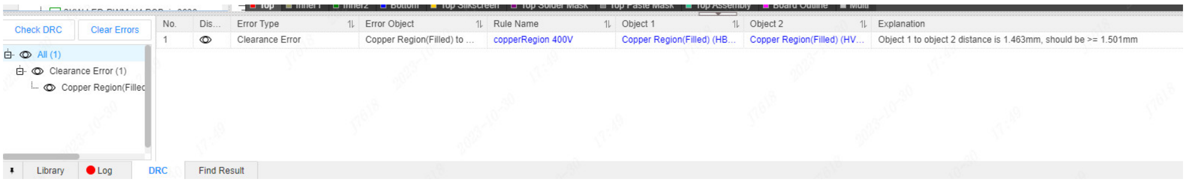
Steps:

- Top Menu - Design - Check DRC (Custom)

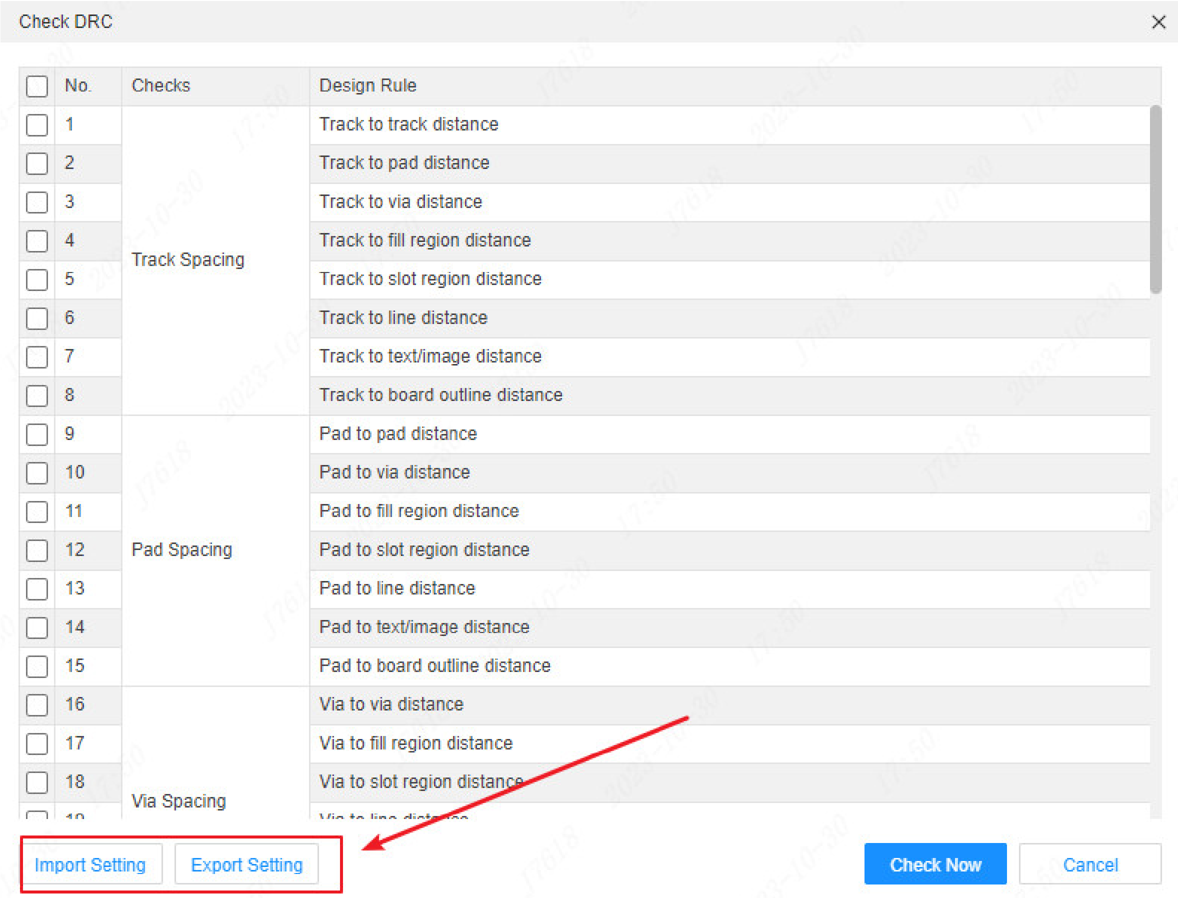


Check the items that need to be checked, click to Check Now, the bottom panel will show the DRC results of the custom project





You can point Export Setting to save the custom check item as a local file. Click Import Setting to introduce the configuration file.

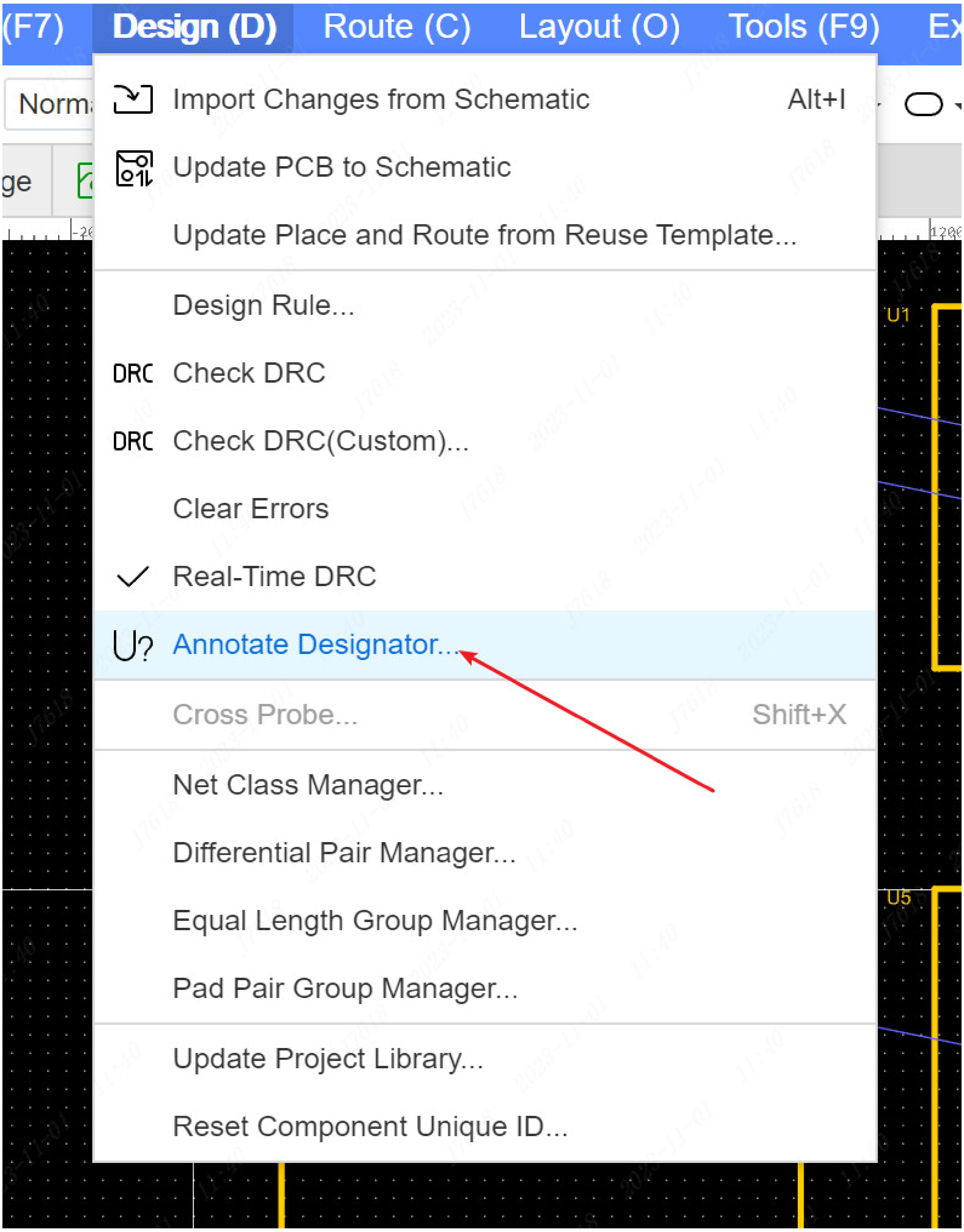


Annotate Designator

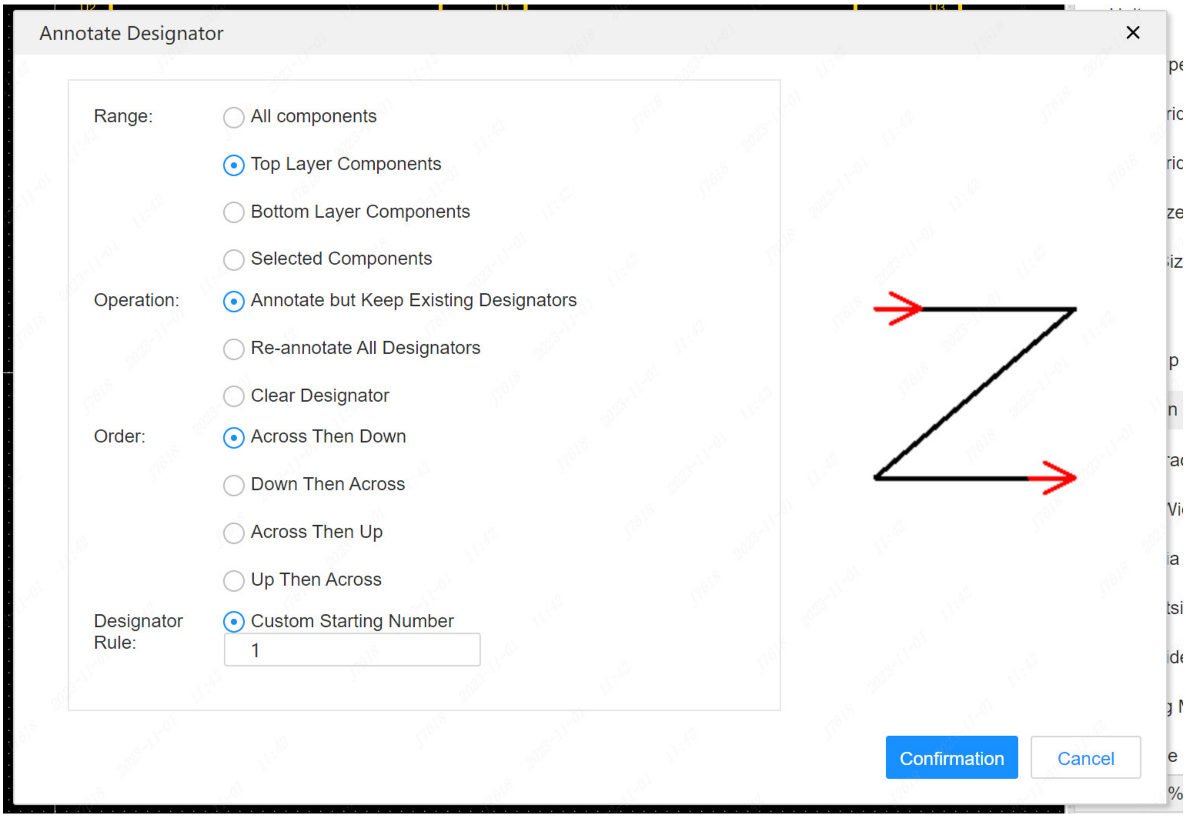
PCB can also be re -assigned by the bit number through the distribution number of the bit number. The position number can be reset through automation, which can greatly save the time of manual distribution.

Steps:

Top Menu - Design - Annotate Designator



Annotate Designator Dialog



Range

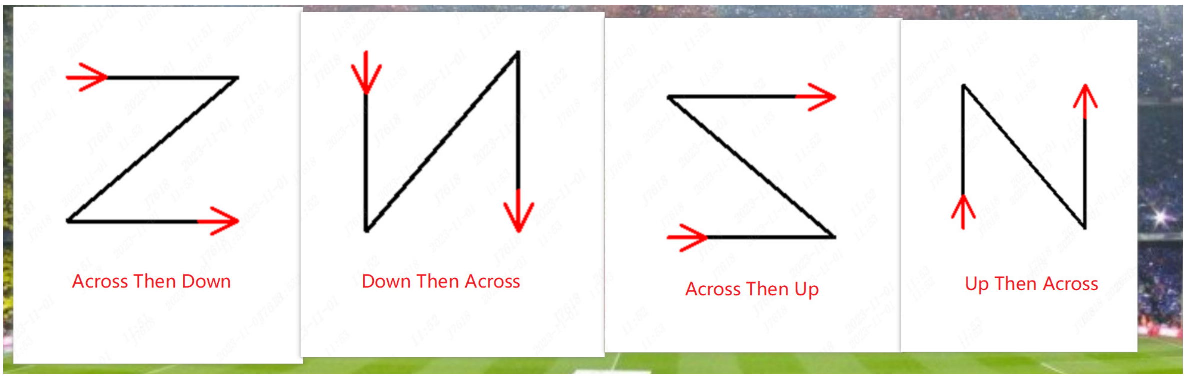
- All components:Annotate designator to all components in the canvas
- Top Layer Component:Annotate designator to all the top layer components in the canvas
- Bottom Layer Components:Annotate designator to all the bottom layer components in the canvas
- Selected components: allocate the distribution number of the element currently selected by the canvas

Operation

- Annotate but Keep Existing Designators: For the reservation of the bit number that already has suffix, only the bit number with the bit number is empty.
- Re-annotate All Designators: All bit numbers are redistributed
- Clear Designator: All bit numbers are reset to the suffix as "?"

Order

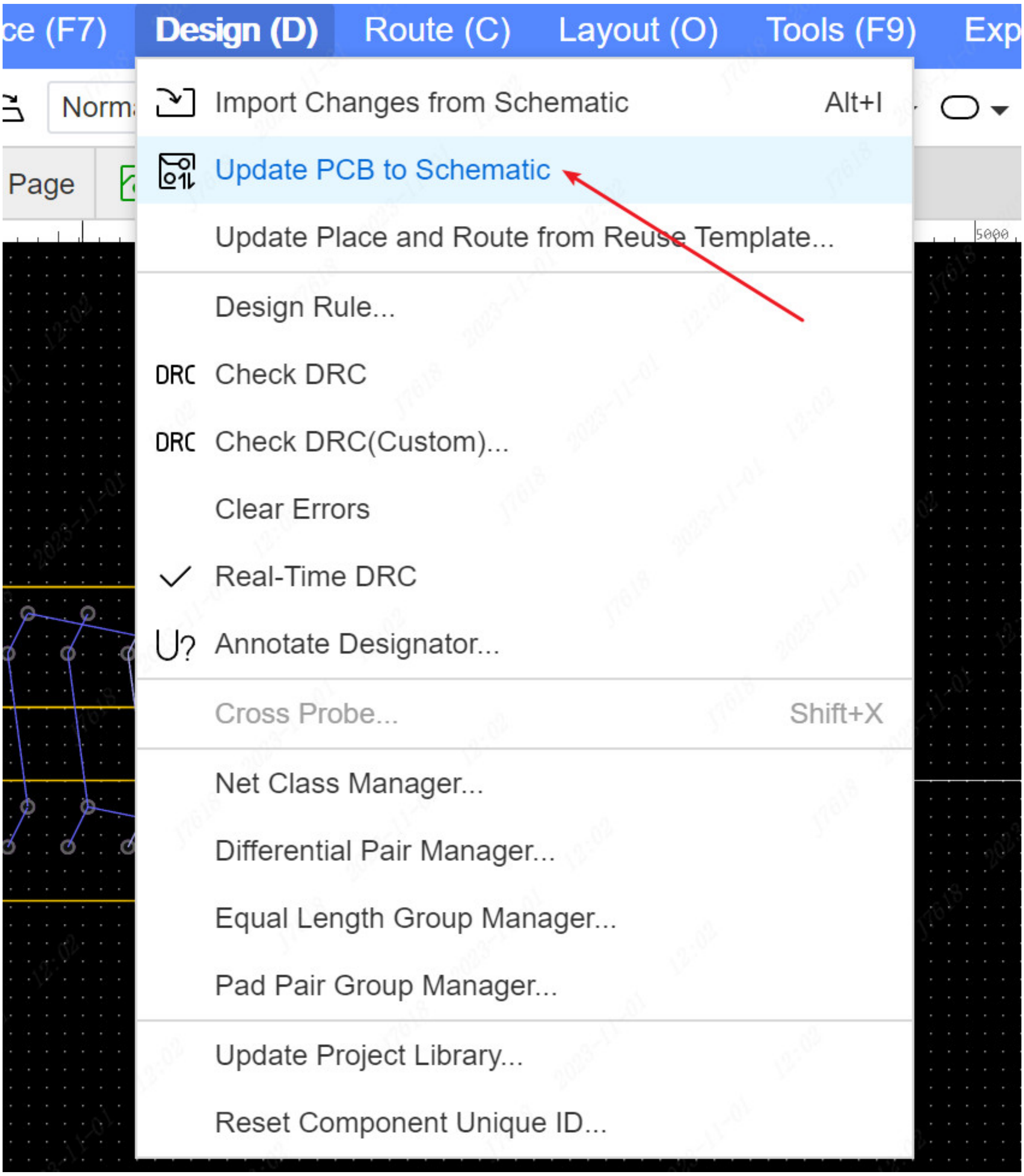
The distribution number number is performed in different order



Designator Rule

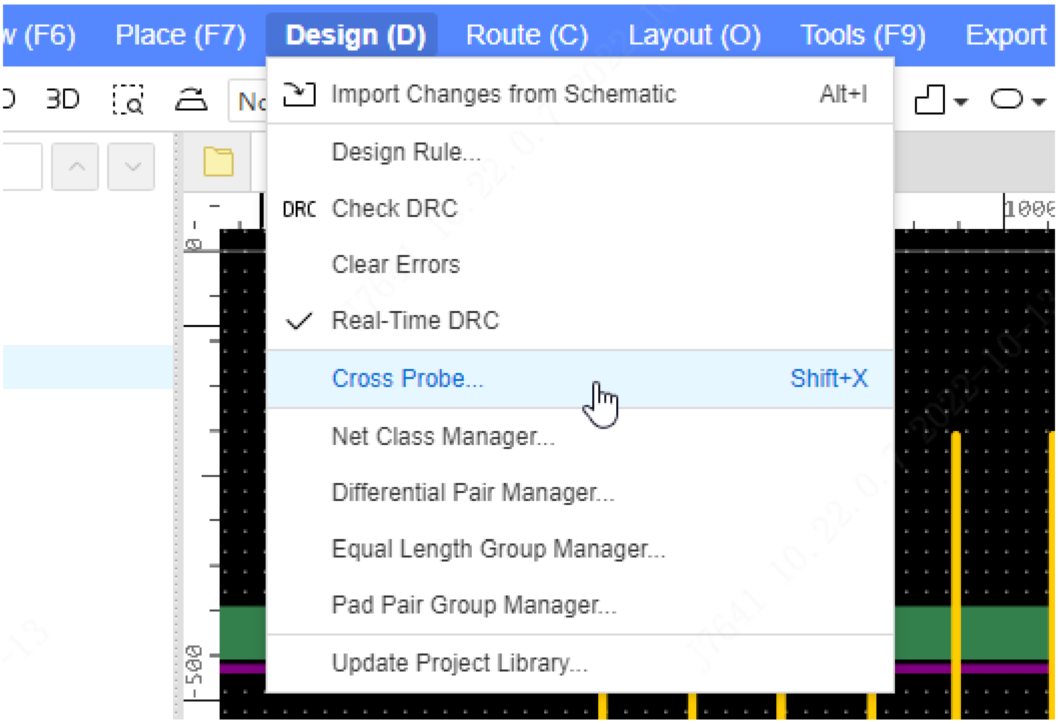
You can choose to start allocating bit numbers from which numbers, and order by incrementing according to this number. If a bit number has been allocated, the next bit number is allocated.

Tips: After the PCB is assigned the bit number, the schematic and PCB mesh will have a difference. At this time, the new bit number can be passed to the principle diagram through the PCB to the original diagram function.

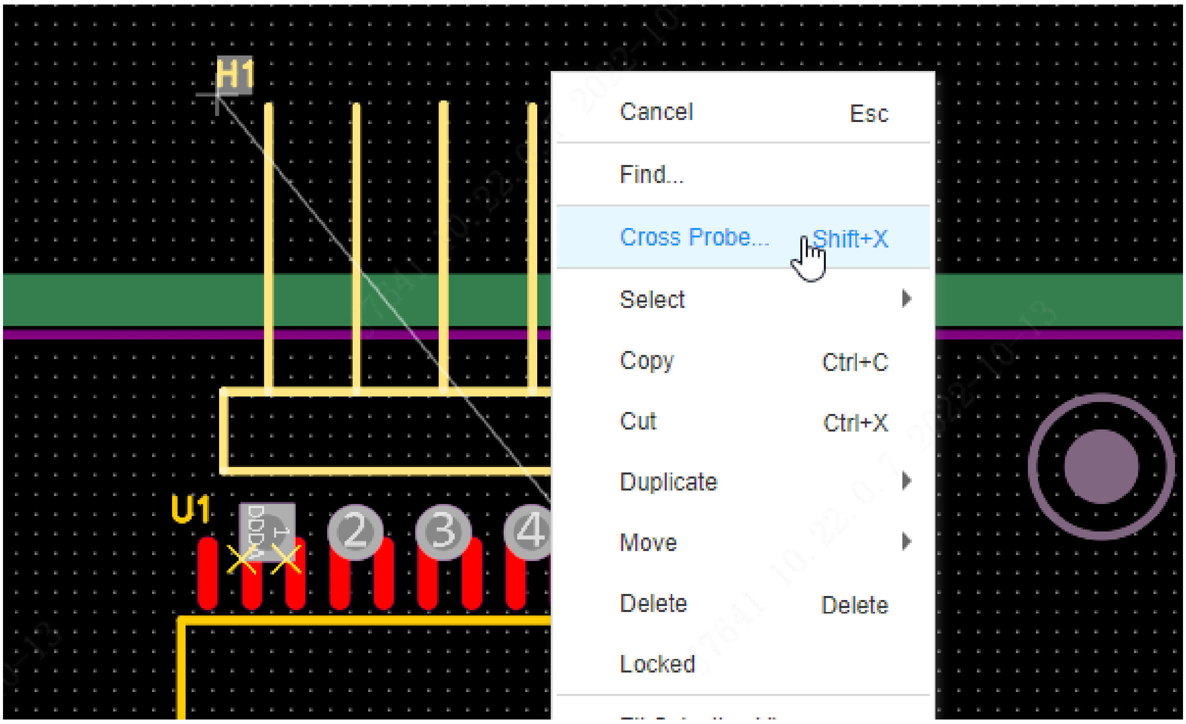


Cross Probe

When a component or component pad is selected, you can use the cross-select function to locate the component position in the schematic



Shortcut key SHIFT+X, or component right-click menu for cross selection



This can also be used to cross-select when both the schematic and PCB have separate windows.

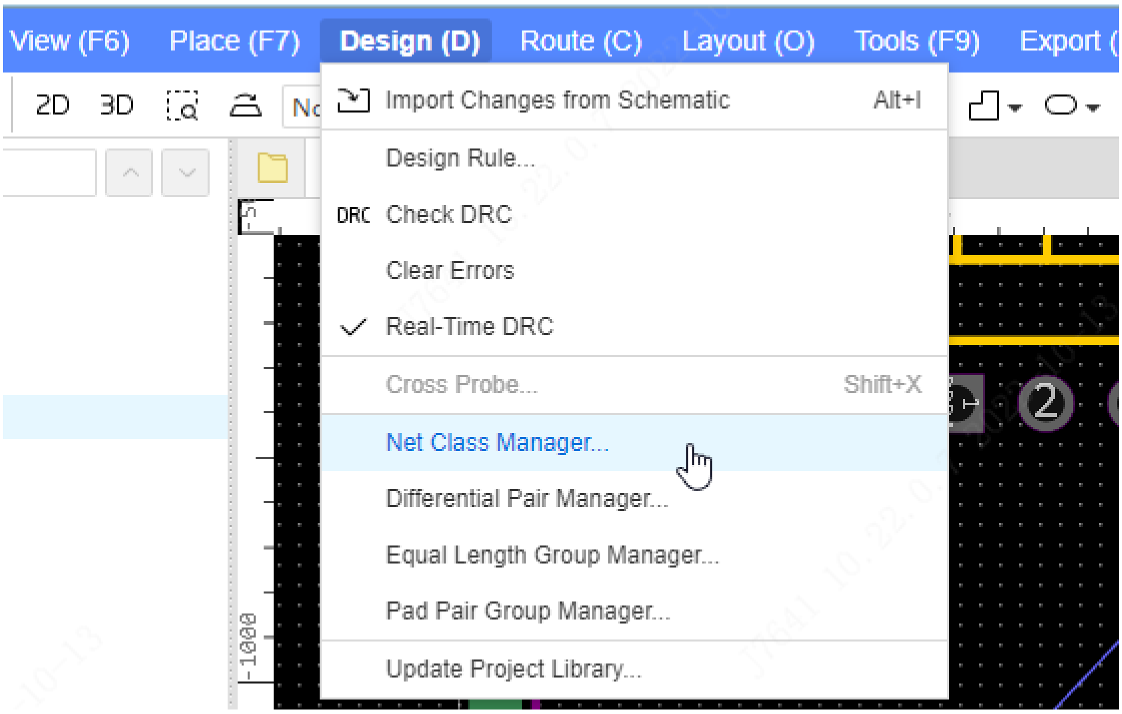
If you directly click the PCB component with the mouse, the components in the schematic window will also be positioned, but the canvas will not be moved. Using the shortcut key to cross-select can automatically move the canvas so that the component is in the center of the canvas.

Net Class Manager

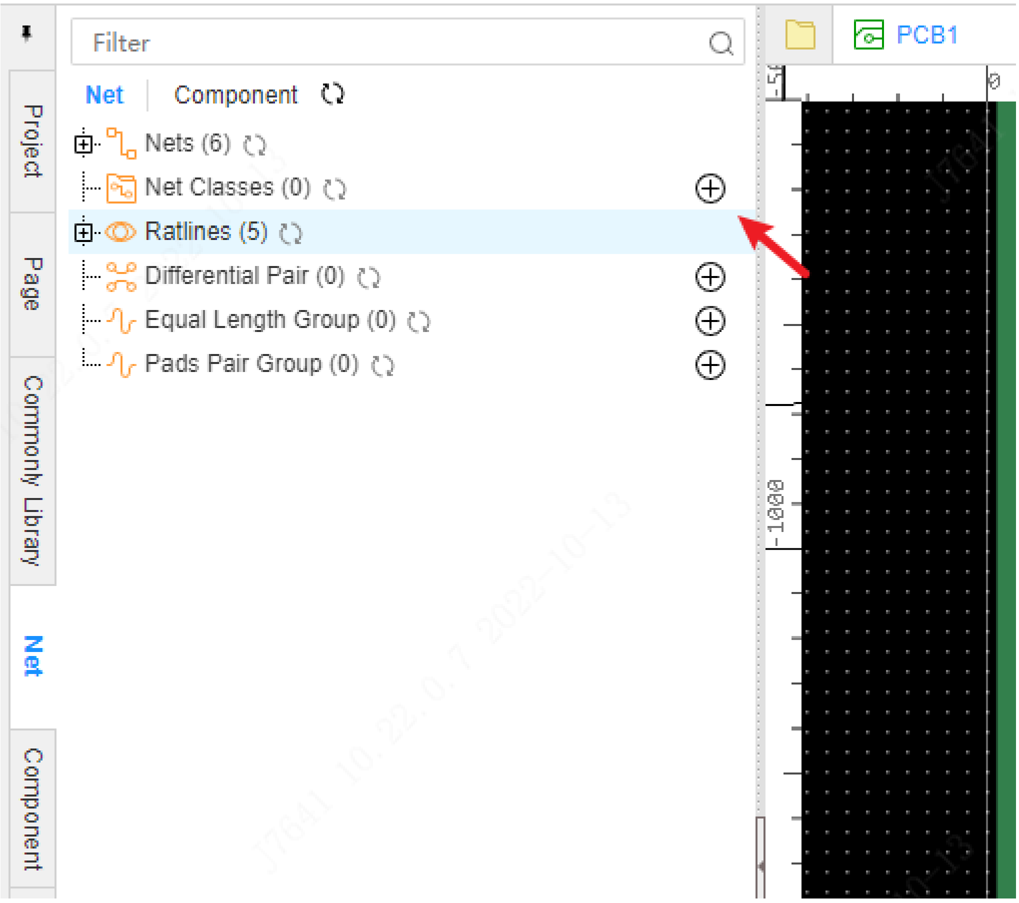
network class

The network class is generally used when the same type of network needs to set design rules together.

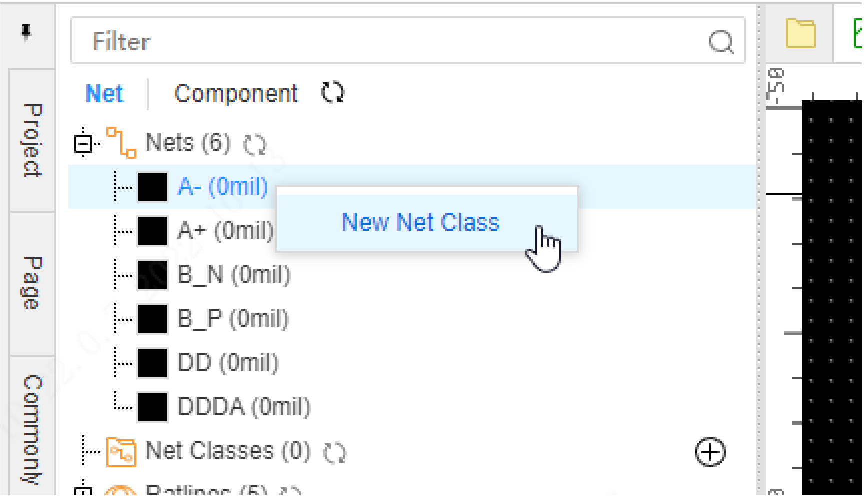
Entry: Design - Network Class Manager, or the left Network Tree - Network Class.



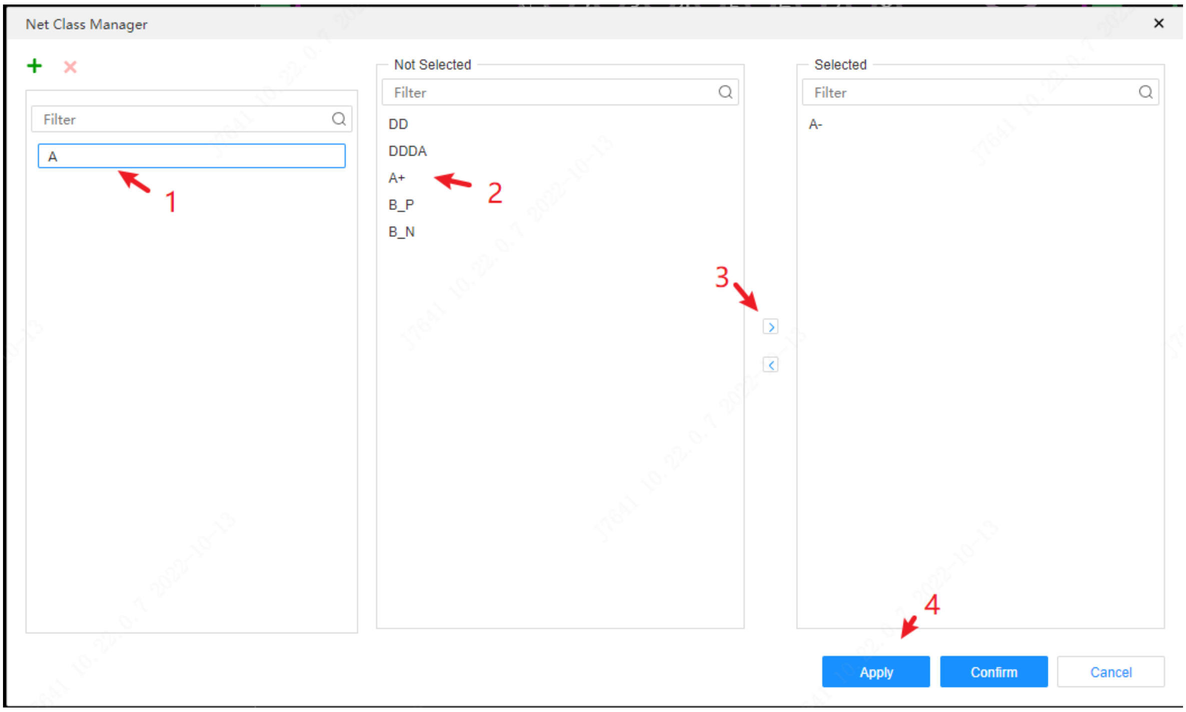
Right-click under the network branch - create a new category or click the new small icon



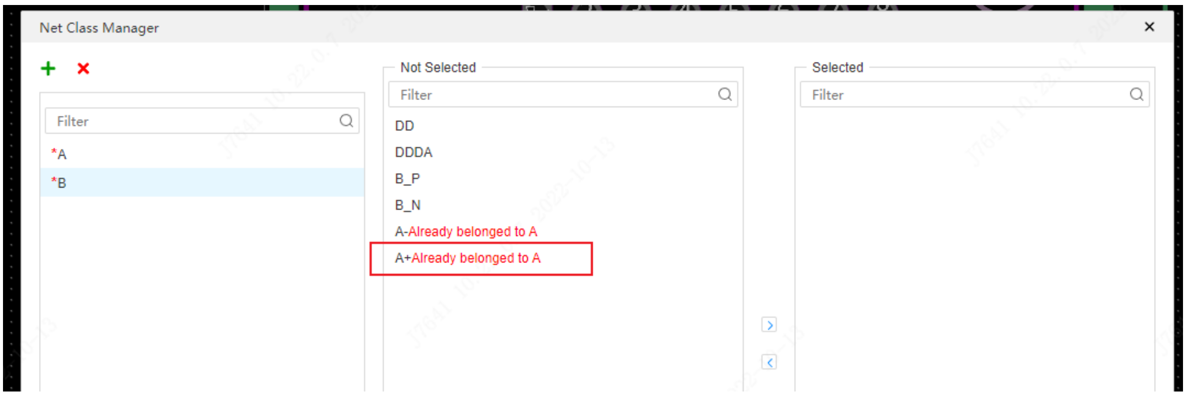
Or right-click directly on the network name in the "Network" list above to create a "Network Class".



Set the name in the network pop-up window, and set the corresponding network.



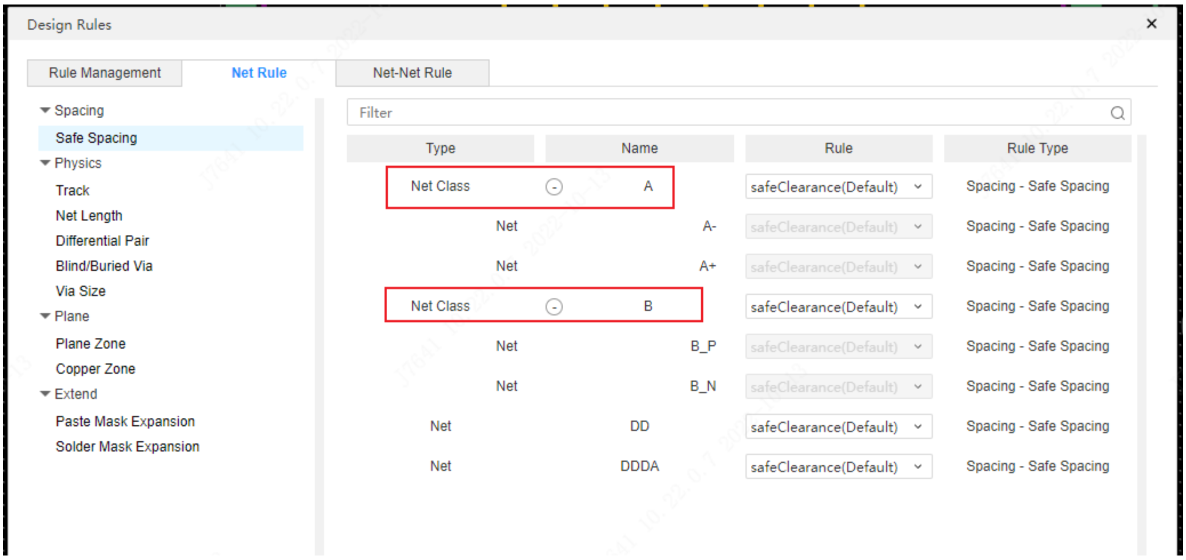
The network names that have been added to the network class will be displayed in the dialog box



After the creation is complete, refresh the network class on the left, and you can see the newly created network class.

Click the Sort button to sort the networks in the network class, so that you can view the networks by network class. Click to sort by network name and network length.

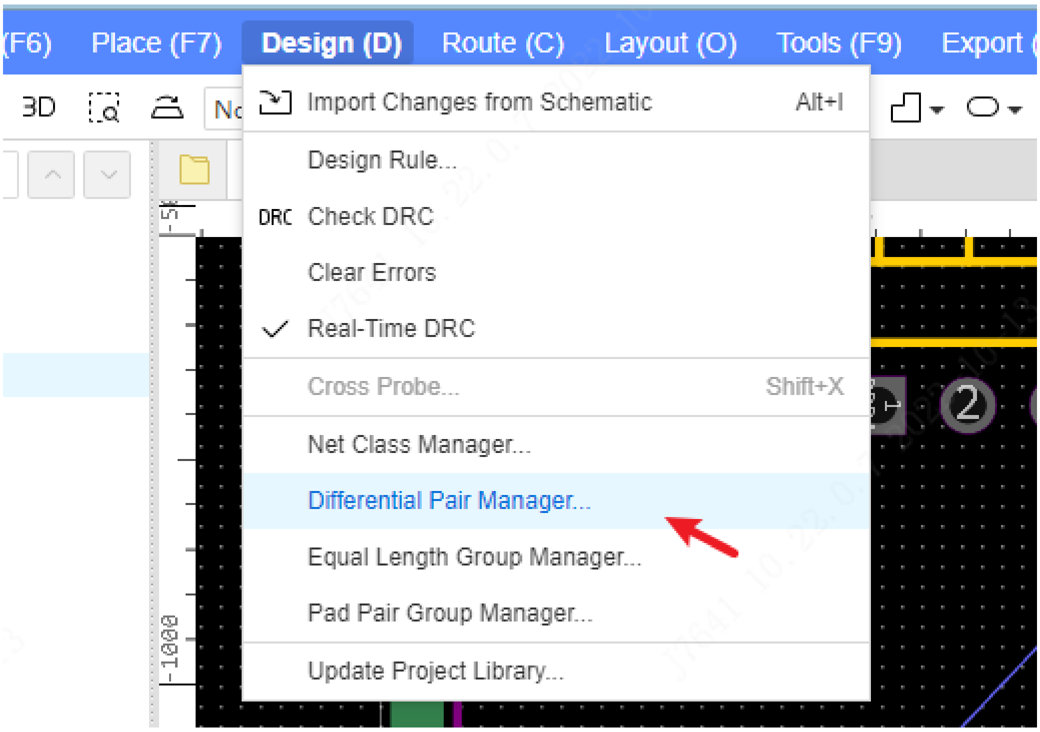
In the design rules, you can set rules for network classes to facilitate batch setting of rules.



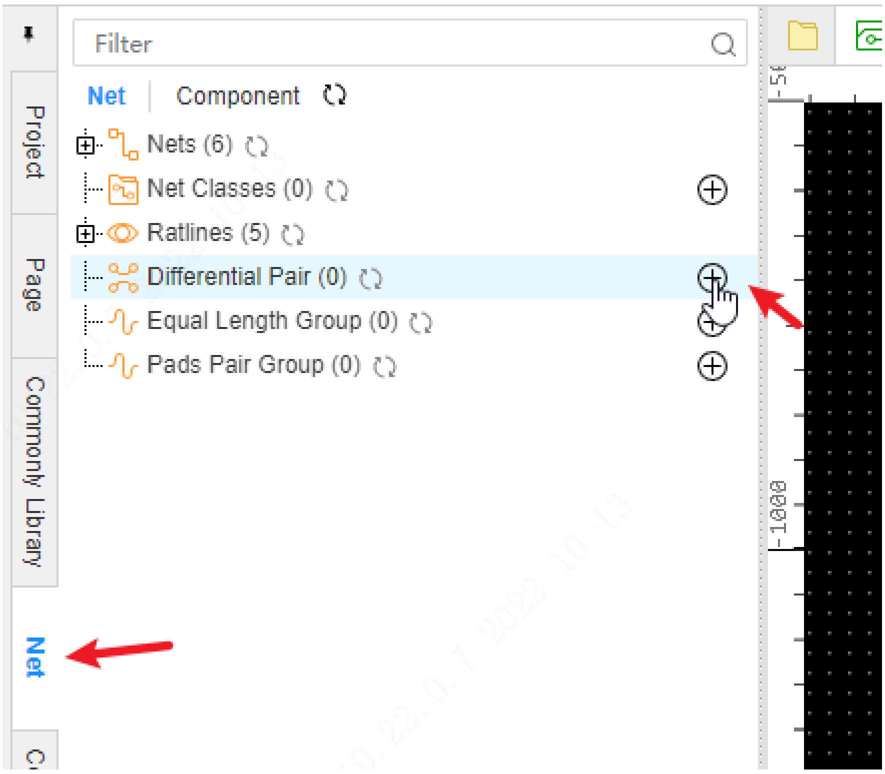
Differential Pair Manager

When differential pair routing is required, the differential pair needs to be created first and design rules are set.

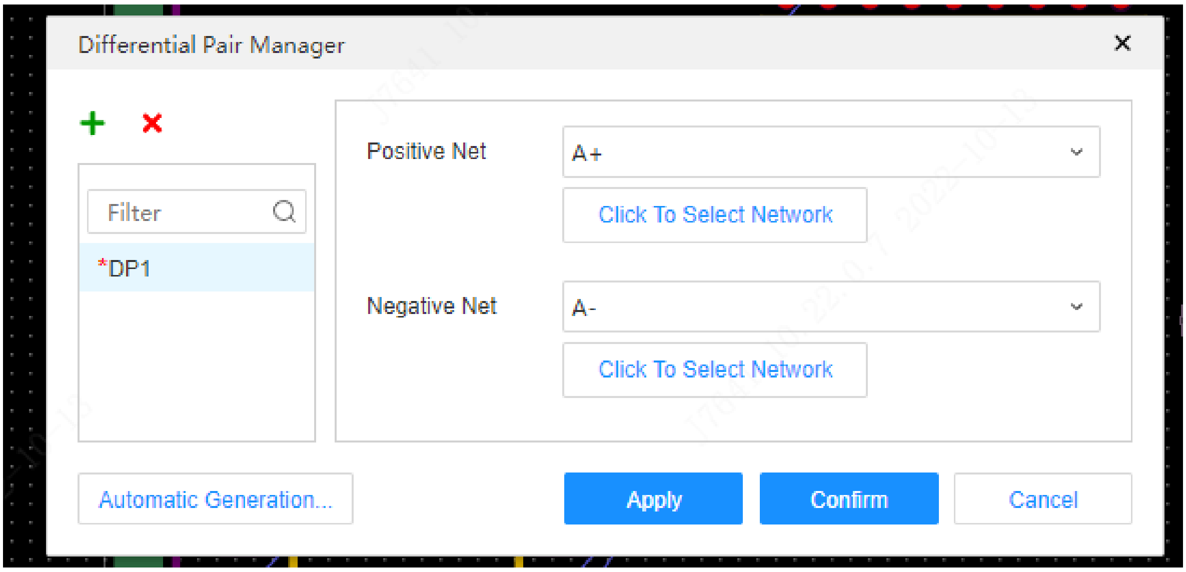
Entry: Design - Differential Pair Manager



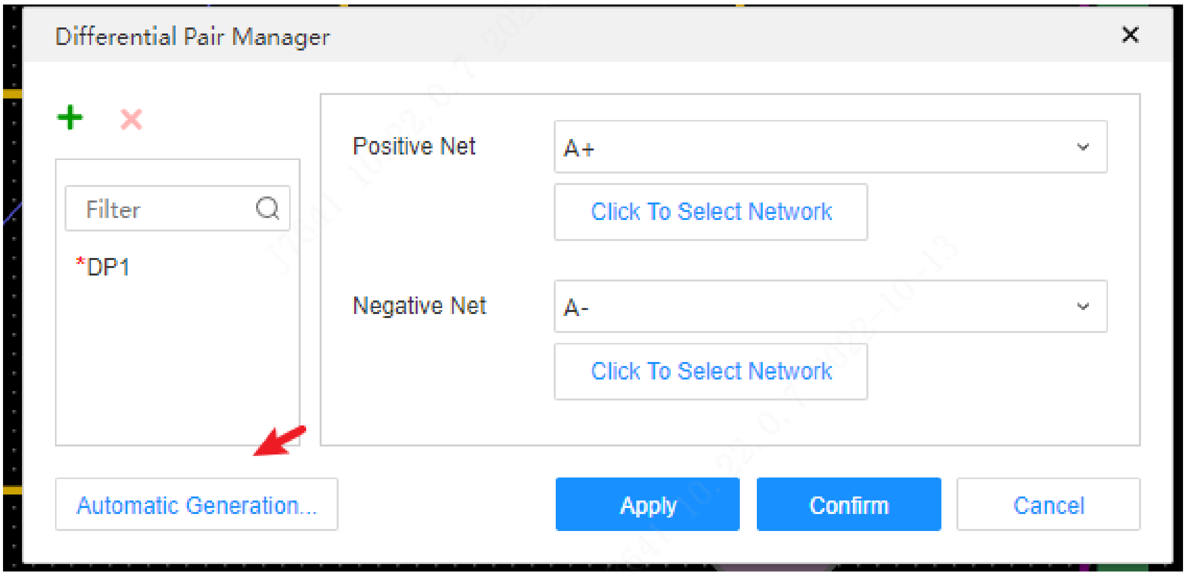
Or in the left network tree - differential pair group, click the new icon of differential pair to create a new differential pair.

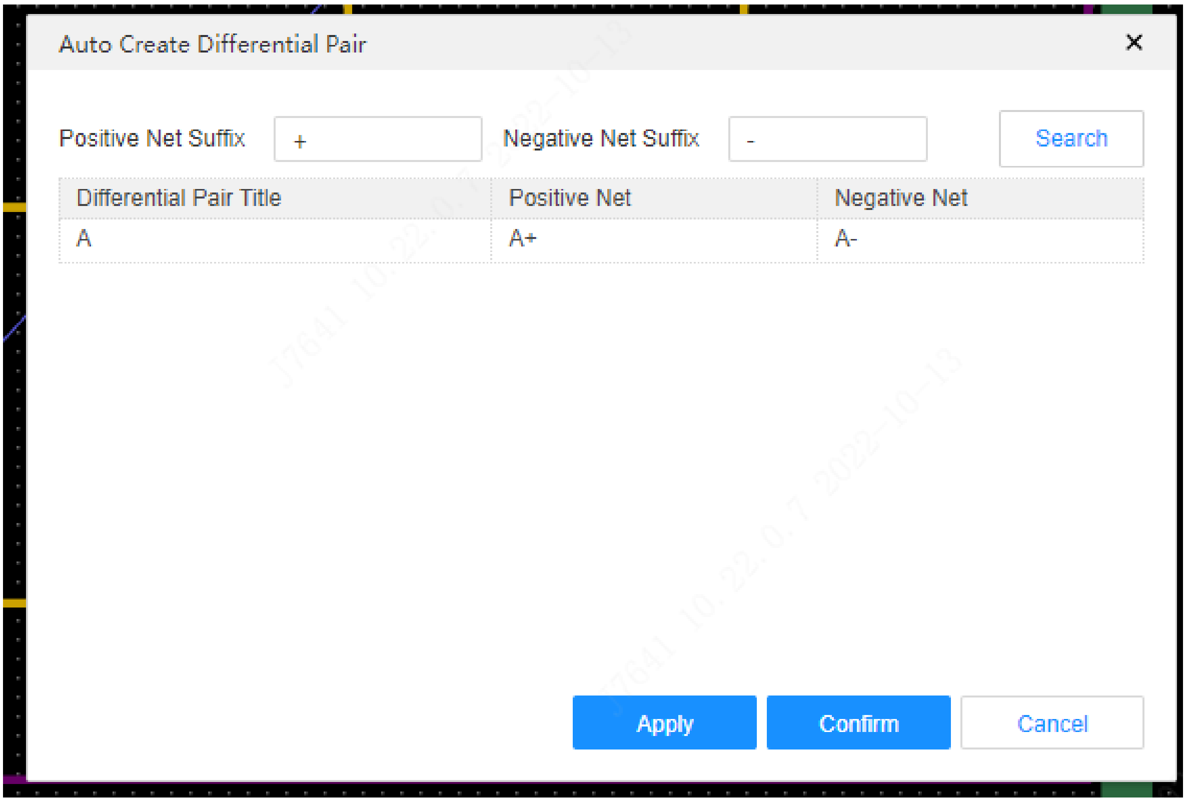


Set the name of the check-up pair, and apply it after adding the positive and negative networks.



You can also use the function of automatically generating differential pairs, which will be matched according to the network name with the same prefix but different suffixes



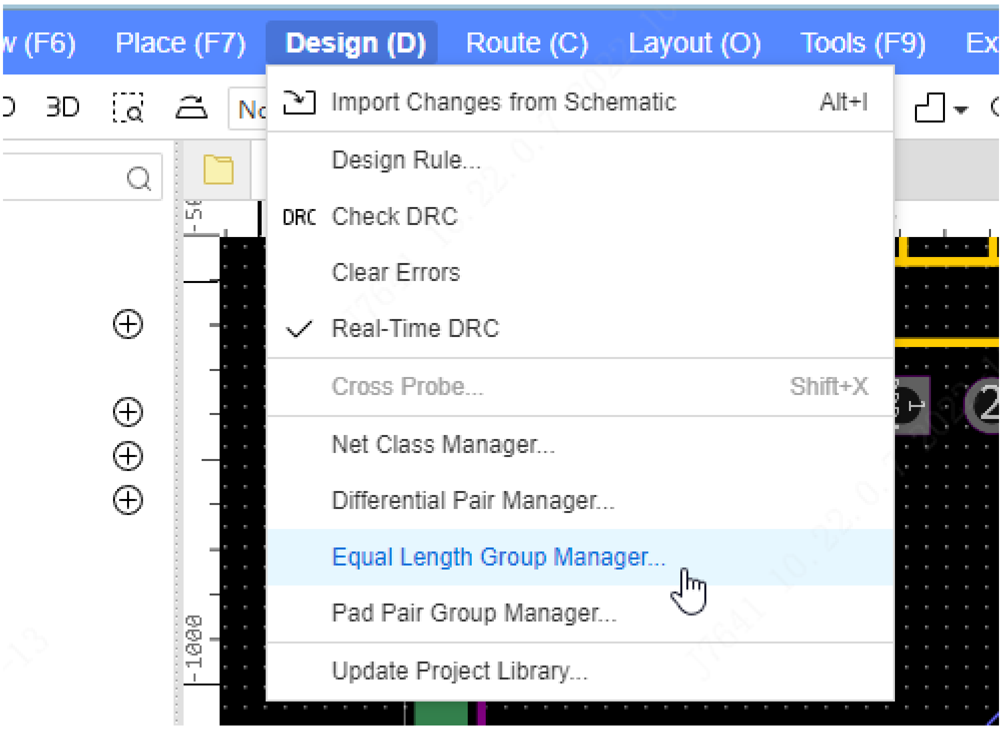


Then, when routing the differential pair, select the net in the differential pair group to start differential pair routing. For differential pair routing, see the Differential Pair Routing section.

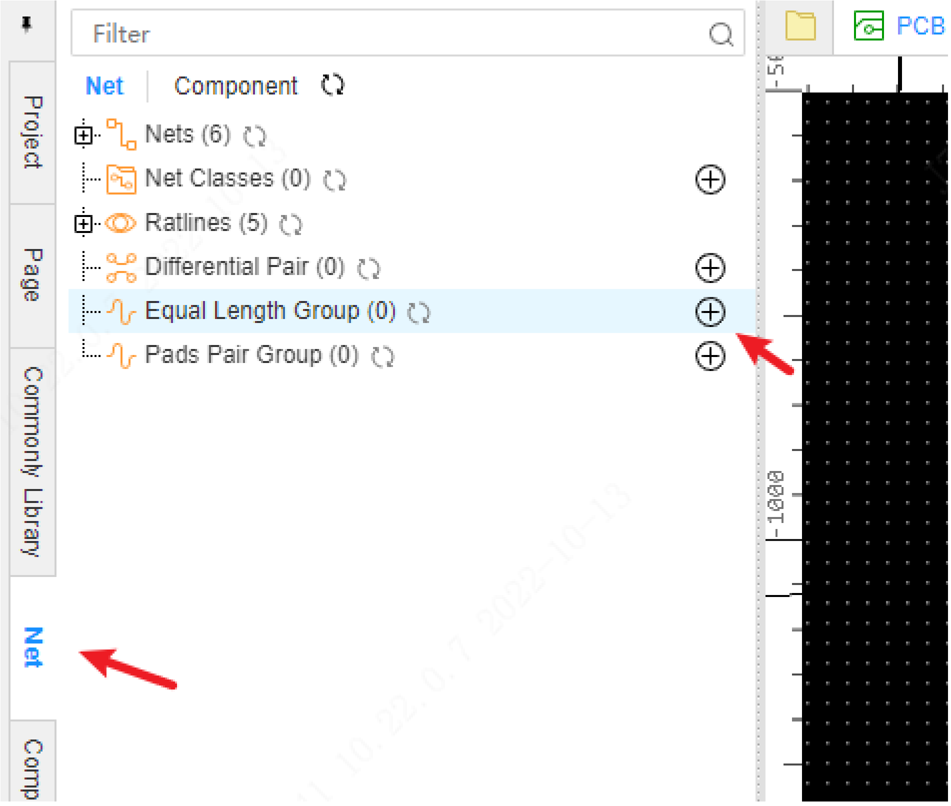
Equal Length Group Manager

When equal-length-controlled wiring is required, setting equal-length network groups can easily compare the network lengths between two networks or multiple networks, and it is convenient to set network rules for the entire network group in the design rules.

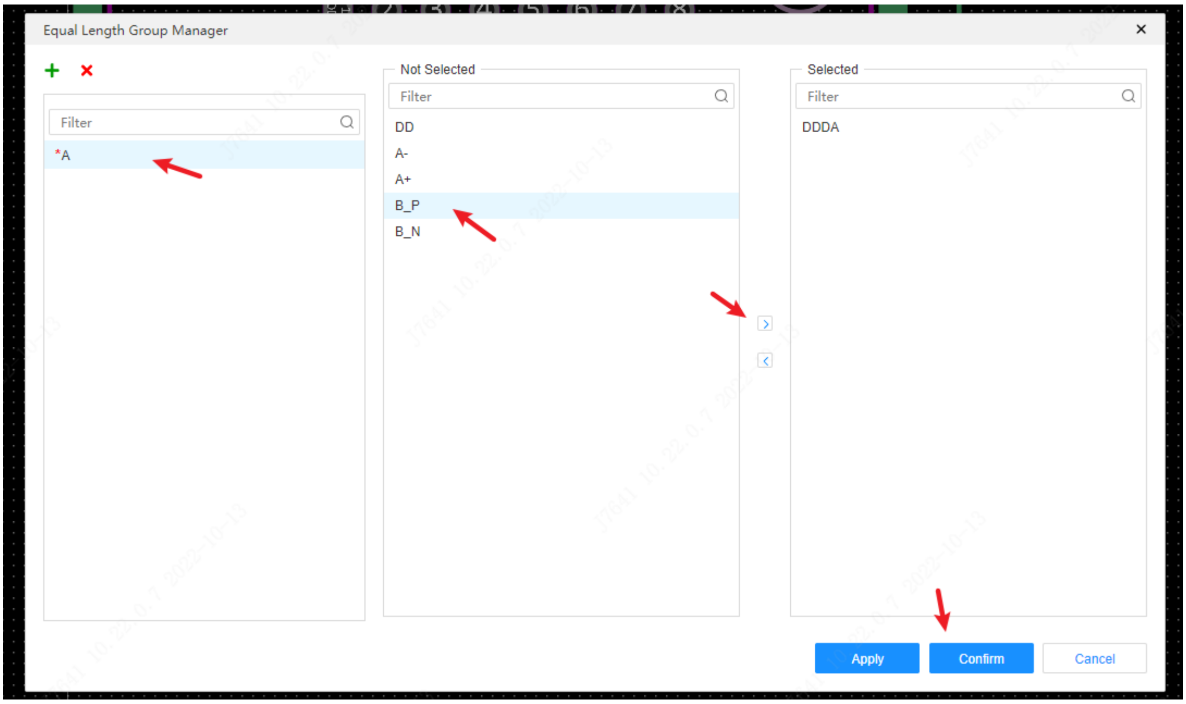
Set Entry: Design - Isometric Network Group Manager



Or created in the left network tree - equal length network group.



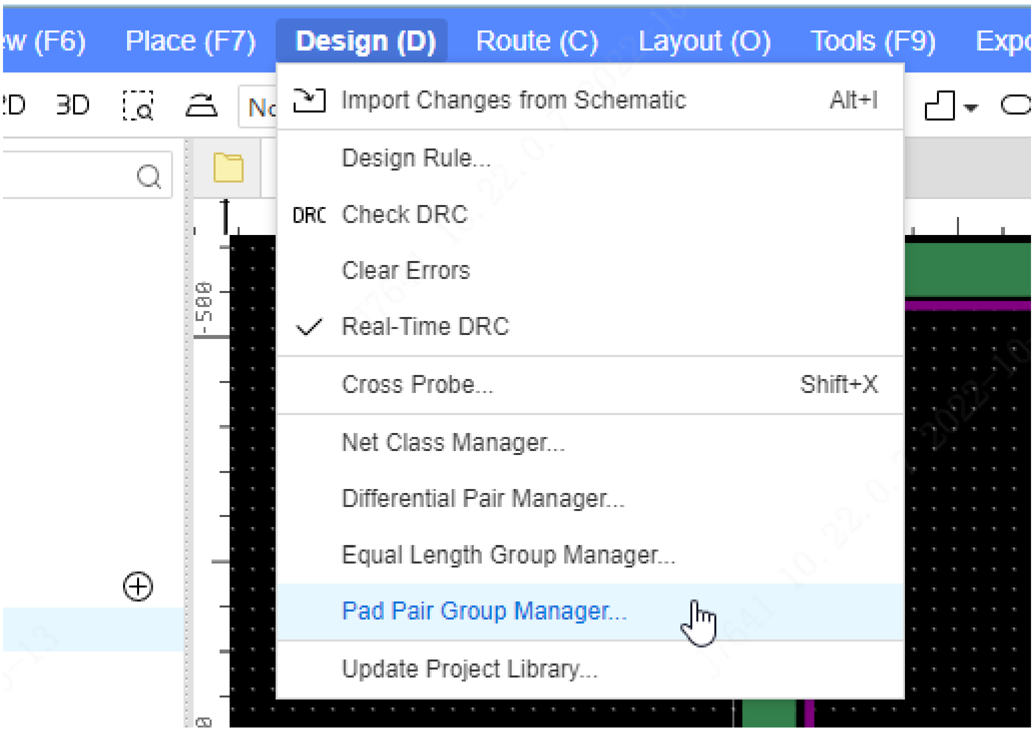
Similar to the previous new network class, click the New button to create it.



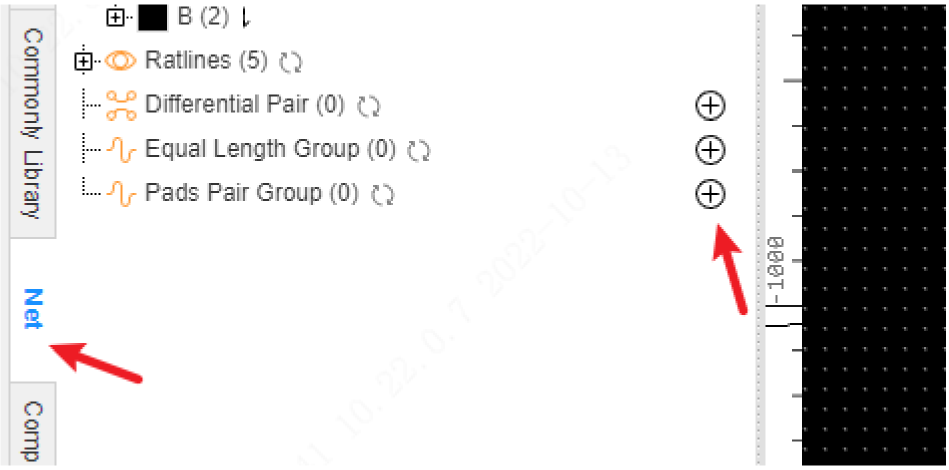
Pad Pair Group Manager

Creating a pad pair group can easily control the length of the differential pair, and you can visually check the length of the wire network between the two pads, which is convenient for equal-length adjustment.

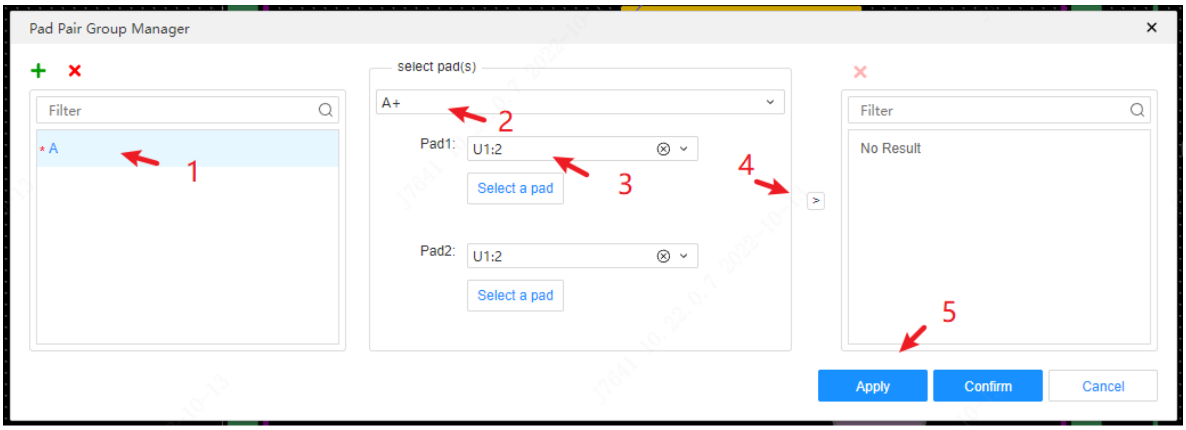
Entry: Design - Pad Pair Group Manager



Or the left network tree - pad pair group is created.



Set the name of the pad pair and associate the pad pair.



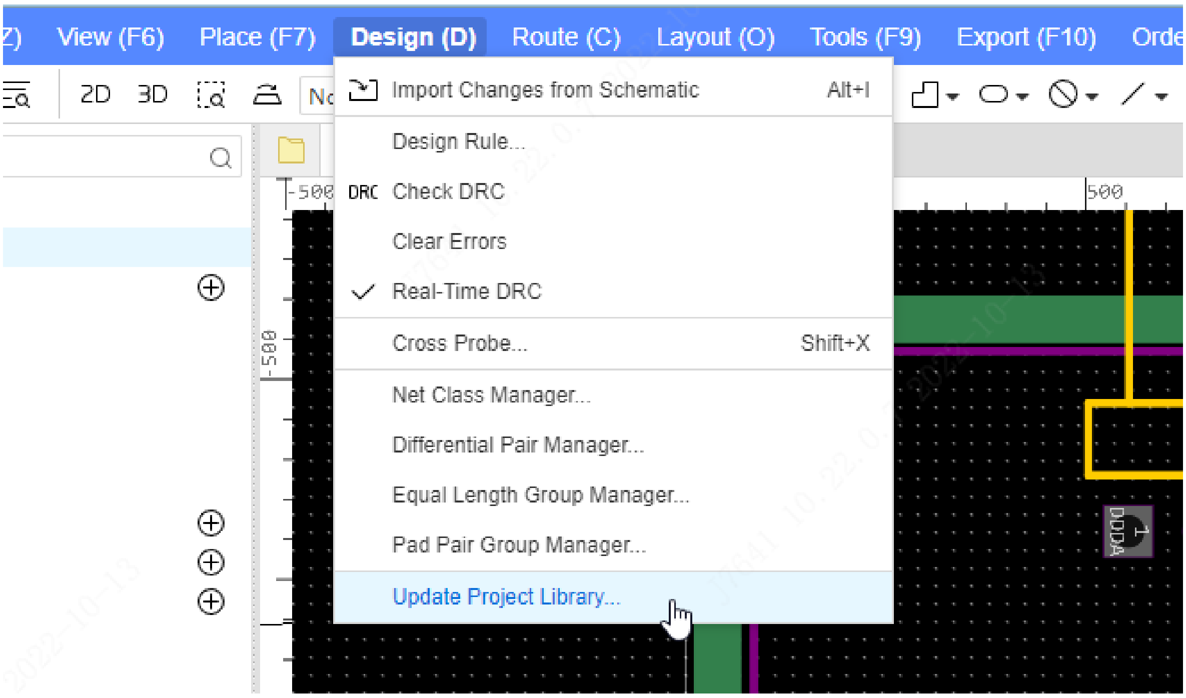
In the network tree on the left, it is easy to check the length of the wire between the two pads, which is convenient for comparison when adjusting the length of the wire.

Update Project library

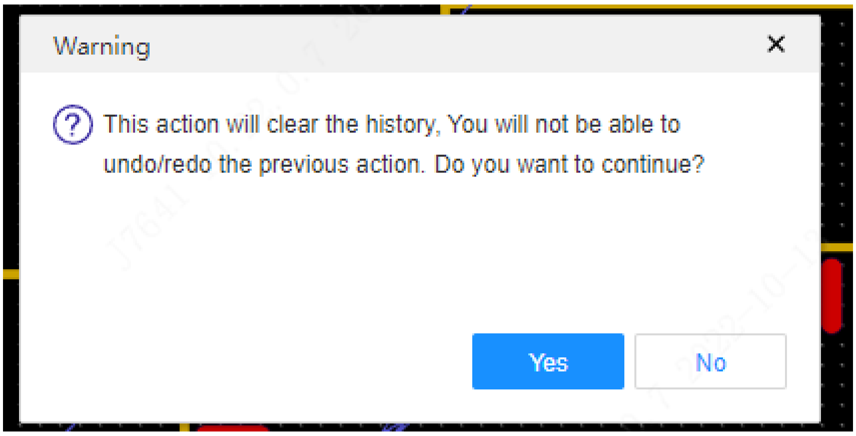
After modifying the footprint and symbols, there is no need to delete the footprint in the PCB, just update the modified footprint or device to the design drawing

How to operate:

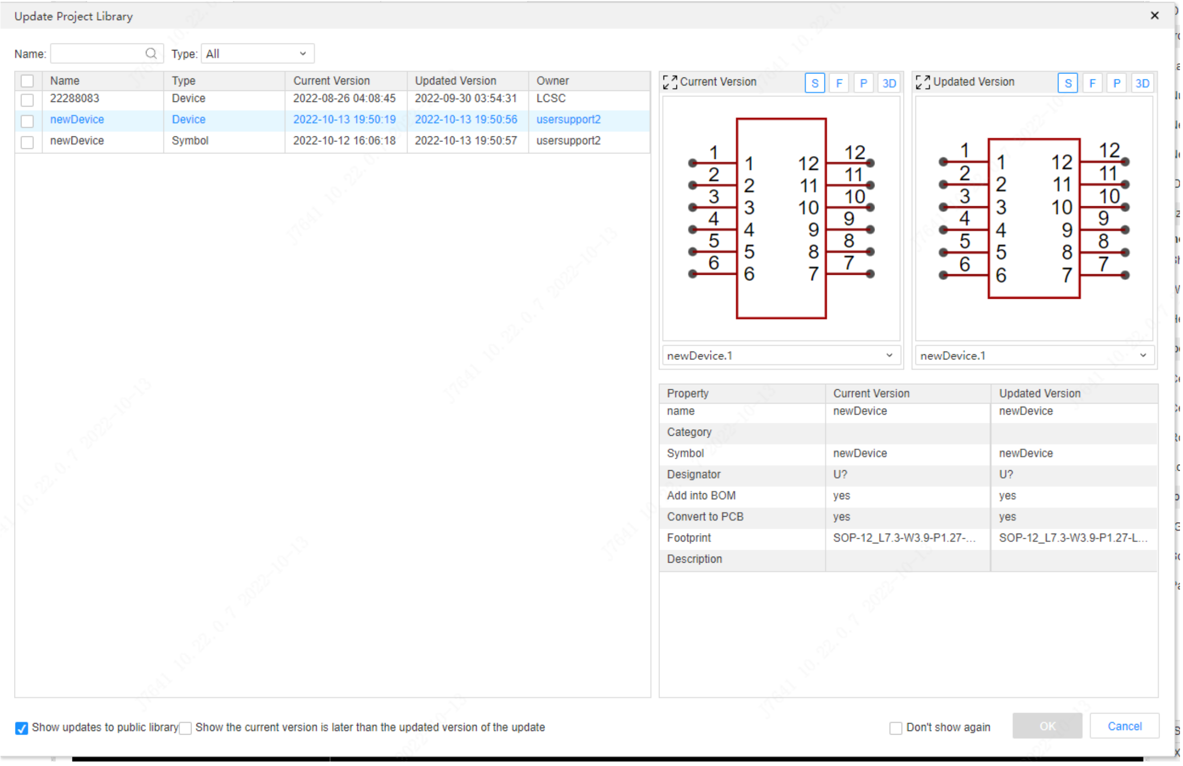
- Top Menu - Design - Update Project Library



will open the project library update dialog, it will list all libraries which can be updated



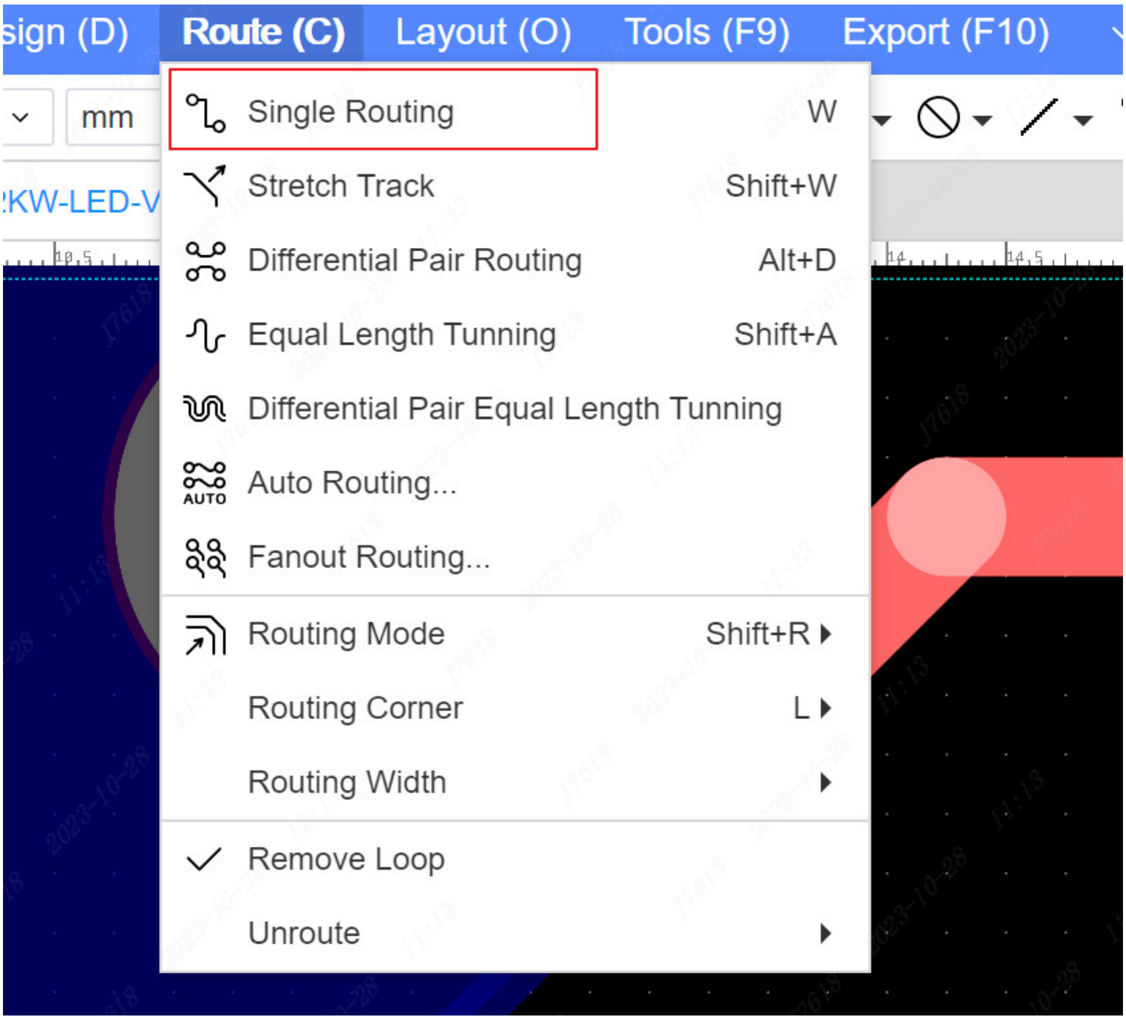
you can update what libraries as you want



Single Routing

Single routing method:

- Click the top menu - Routing - Single Routing to enter the Add Routing mode
- In addition to the top menu, you can also use the top toolbar - one-way routing
- Shortcut W
- Double click on pad or via to enter routing mode



Add inflection point

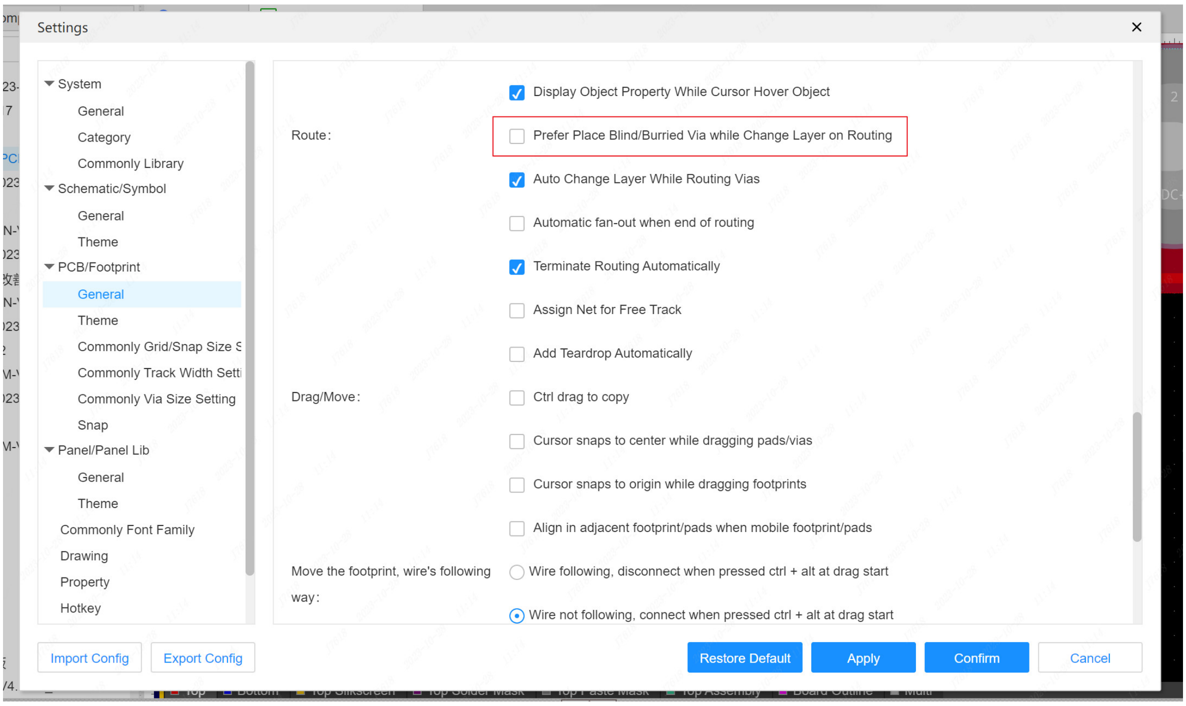
- In the process of routing, click the left mouse button to add an inflection point where you need to add an inflection point.
- Use the shortcut key `L` to switch the knee point during routing.

Add vias when routing

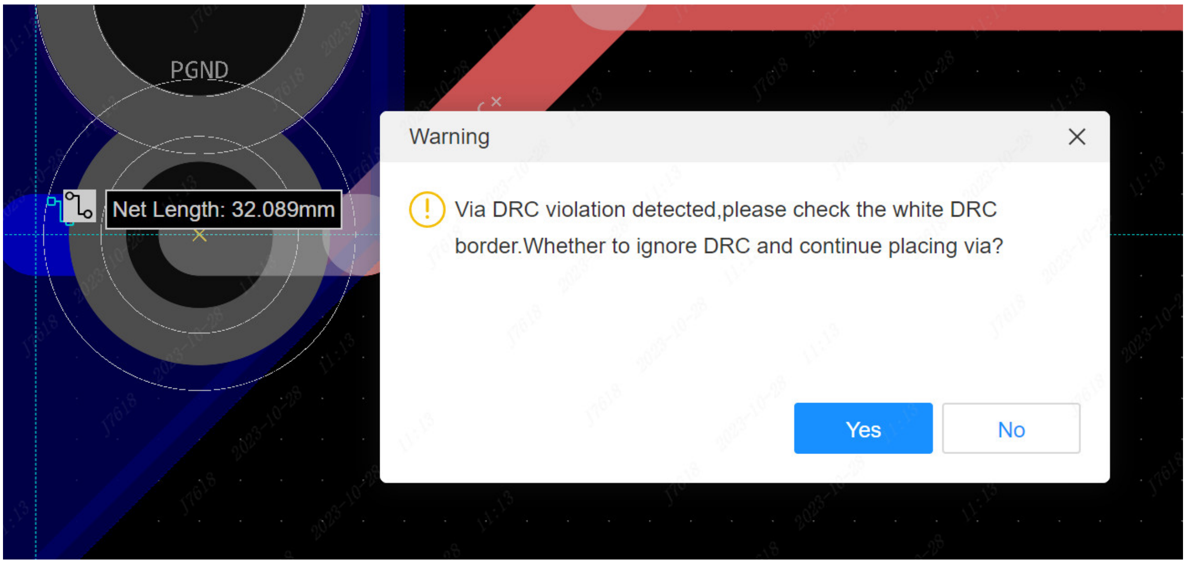
- Click the right mouse button during the routing process and select the function to add a via hole, and a via hole will be added at the next corner to be routed. Click the left mouse button on the canvas, and the via hole will be added successfully
- You can also directly add vias by entering the shortcut key `V` during the routing process
- Cut the layer to the signal layer during the routing process, and automatically add vias

If you want to add blind and buried vias, you need to add a list of blind and buried vias in the design rule first. Then click the right mouse button to place the blind via or buried via function during the routing process ** Select the blind and buried via you need to add.

- In the setting dialog box, if **Blind and buried vias are placed preferentially when routing layer cutting is checked,** When the signal layer is cut during the routing process, blind and buried vias that meet the conditions will be preferentially placed for Place



- In the block, around, and push mode, a hole is added during the wiring process. If the addition of the additional perforated violates the DRC, a warning prompt will appear. You can click on whether the DRC continues the wiring, or click to cancel the addition hole.



fallback routing

- During the routing process, click the right-click menu, click Back, or use the shortcut key backspace, you can manually back the routing to the previous inflection point
- During the routing process, if the cursor moves to the part that has already been routed, it will automatically return to the previous inflection point of the wire at the cursor position. At this time, if the mouse moves back to the part where no loop is generated, the previously hidden wire will be restored; after adding an inflection point, a new inflection point will be determined, and the previously automatically retracted wire will no longer be restored.

Switch via size when routing

- Follow the rules. The vias added next after clicking will become the default size in the design rules
- Rule settings. Click to pop up the Design Rules dialog box, where you can set the rules for via size

- List of commonly used via sizes. Quickly switch the commonly used via size through the right-click menu during routing, click the common size setting, and a system setting dialog box will pop up, where you can maintain a list of commonly used via sizes.
- Enter the via size. After clicking, a pop-up window will pop up, enter the outer diameter and inner diameter of the via hole to be modified, and then press Enter, the next via hole will be changed to this size

Stretch Track

stretch the track

- Routing via: Top Menu - Routing - Stretch Track
- Top Toolbar Stretch tracks
- Shortcut `shift`+`W`
- Click on a segment of track to start stretching. You can also click to select a section of track, and then click the function to stretch the track.
- Move the cursor to stretch the track, click the left mouse button again, and the track is stretched this time. When the track is stretched to merge with other tracks, it will stay at the fusion position and cannot continue to stretch

Drag track to stretch

Select a section of track, long press and drag to stretch it

When the mouse moves to the track head, and the mouse position changes to a hand icon, it means that it can be dragged

Drag right-angle inflection point

When the mouse moves to the right-angle inflection point, and the mouse position changes to a hand icon, it means that it can be dragged

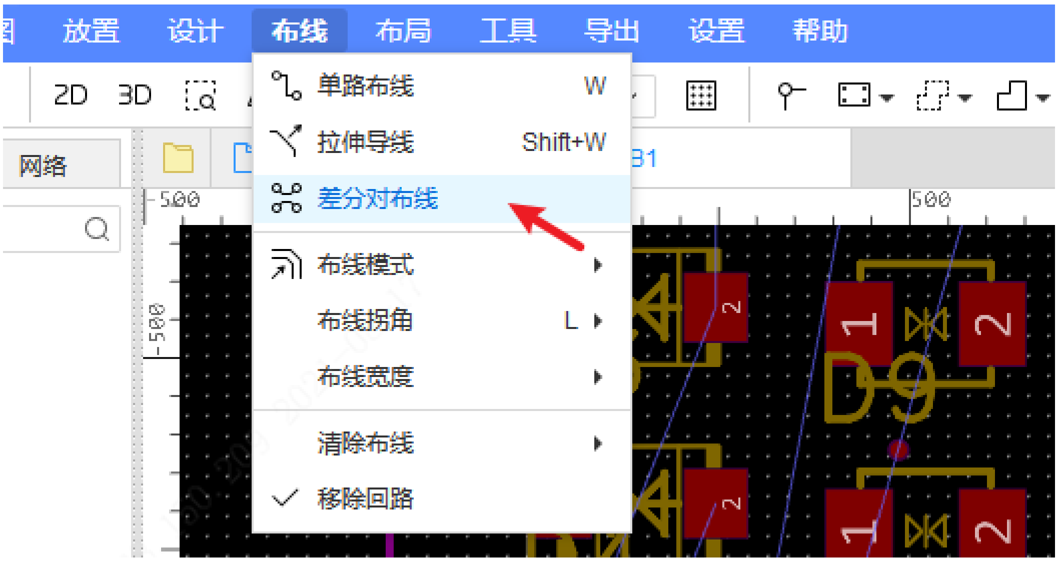
Differential Pair Routing

Differential pair routing

Differential pair routing is a technique that requires the creation of a transmission system on a printed circuit board that favors the balance of differential signals (equal and inverted signals). The differential line is generally connected to the external differential signal system. The differential signal system uses twisted pairs for signal transmission. One signal line in the twisted pair transmits the original signal, and the other transmits the signal with the opposite phase of the original signal. Differential signal is a method used to solve the problem that there is no good reference ground connection between the signal source and the load, and it has an inherent inhibitory effect on the interference of electronic products. Another advantage of differential signaling is that it reduces the electromagnetic interference (EMI) generated by the signal lines.

Set differential trace steps:

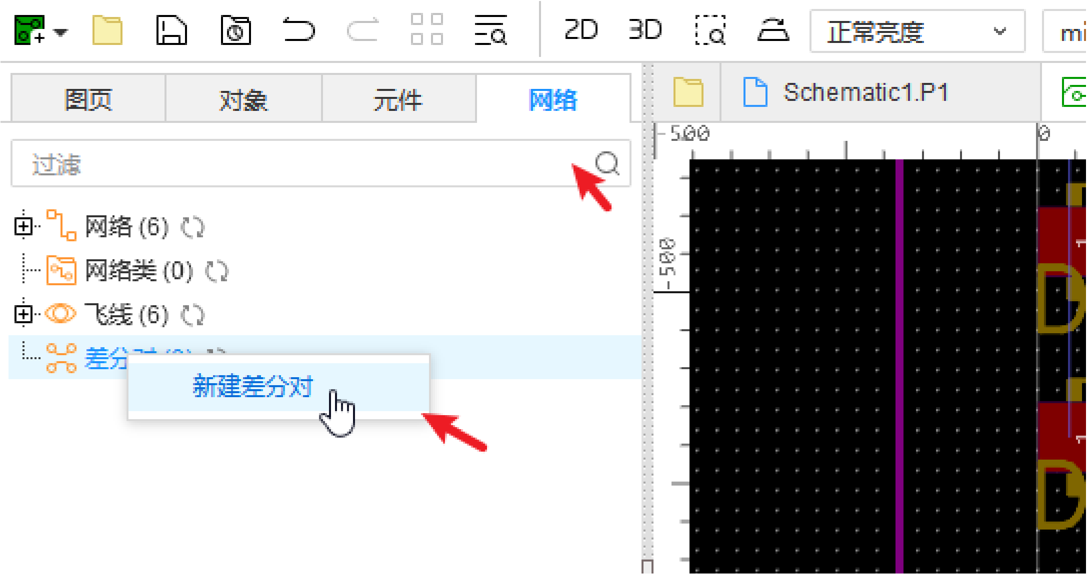
- Top Menu - Routing - Differential Pair Routing.



Create differential pair network class

Before performing differential pair wiring, you need to set up the differential pair network class.

1) You can right-click on the differential pair category on the left network tab and create a new differential pair.



2) You can also directly click the wiring menu - differential pair wiring. After clicking the pad, if it is detected that no differential pair has been established, a pop-up window will prompt you to create a differential pair.

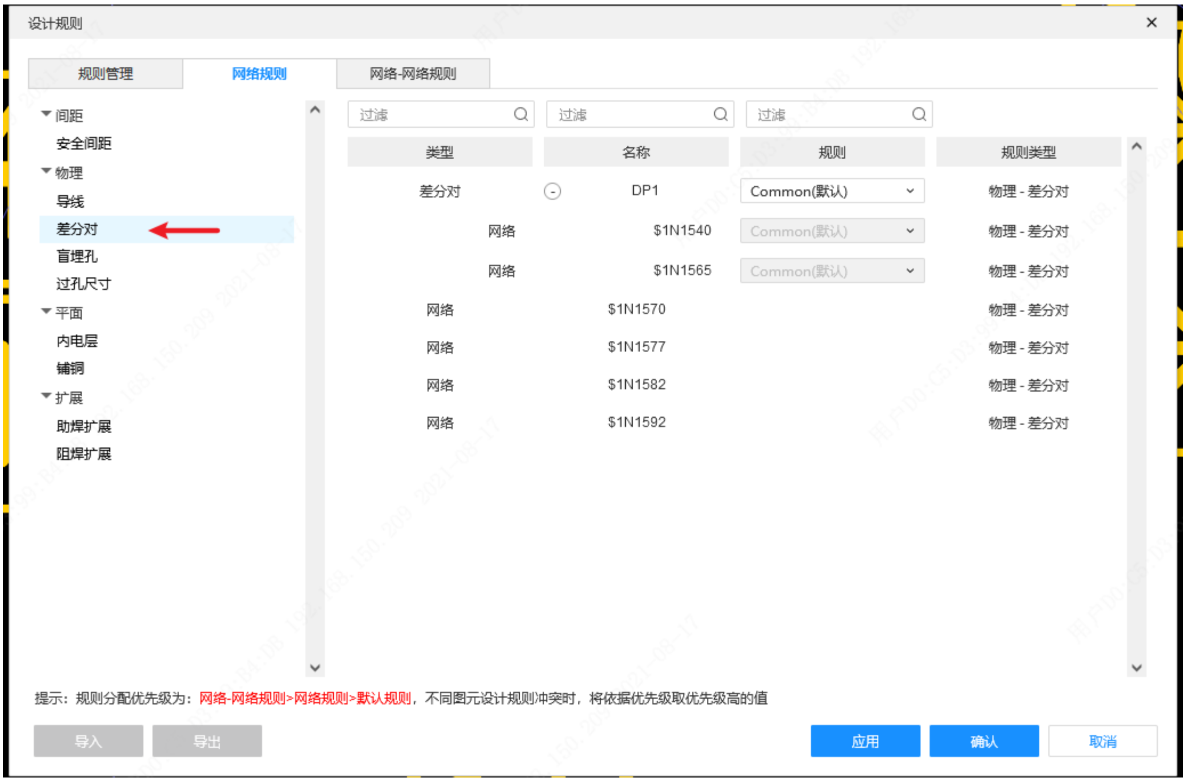
3) It is very convenient to create differential pairs

Differential pairs can be automatically generated, and the query generation is performed according to the positive and negative signs of the network name:

After creating the differential pair, you can see it on the left network tab. And you can see the total wire length of the network.

Create differential pair rule

Set the net rules for differential pairs in the Design menu - Design Rules.



plot differential pairs

When the rules are set, you can click the differential pair routing menu to perform differential pair routing, and you can also view the routing length and differences in real time during the routing process.

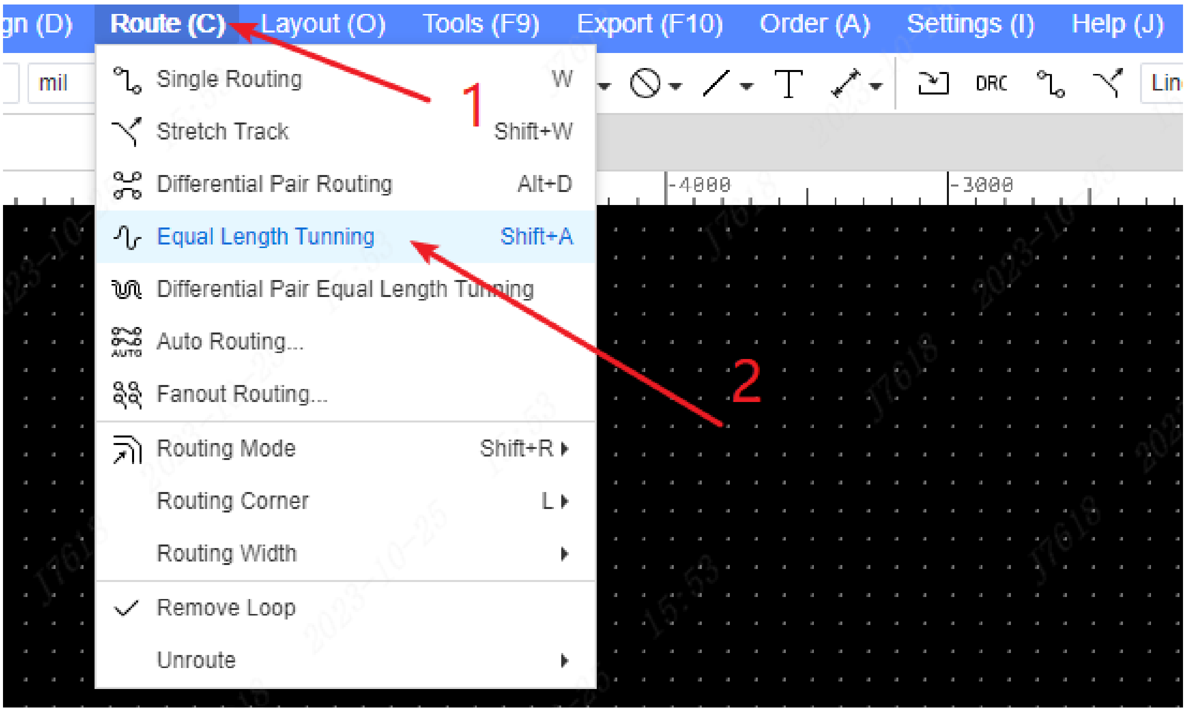
Differential Pair Routing

Equal Length Tunning

Equal-length adjustment, also known as equal-length tunning and delay line, is a special form of routing in PCB design. Relative delay between group signals to avoid timing problems.

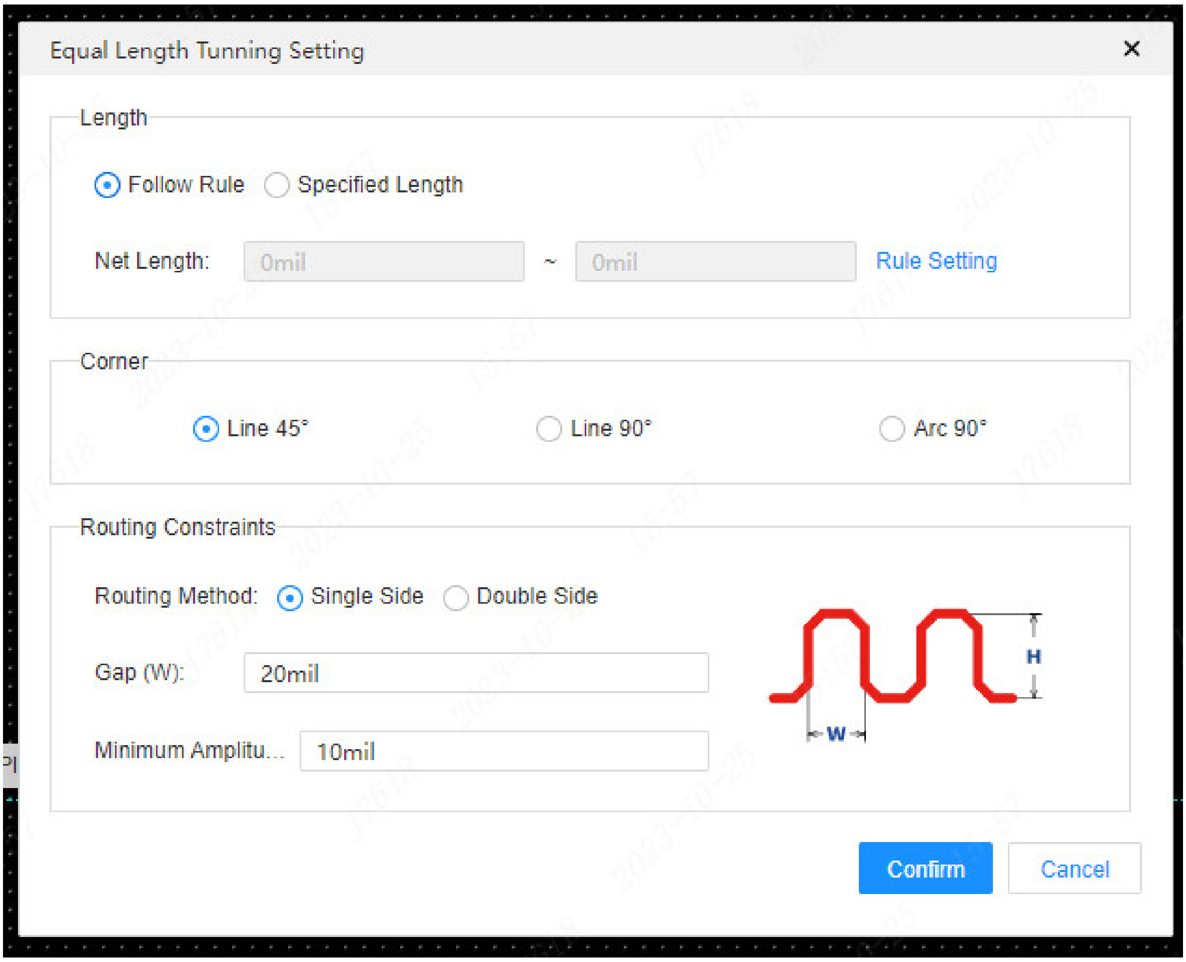
Steps:

- Top Menu - Routing - Equal Length Tunning



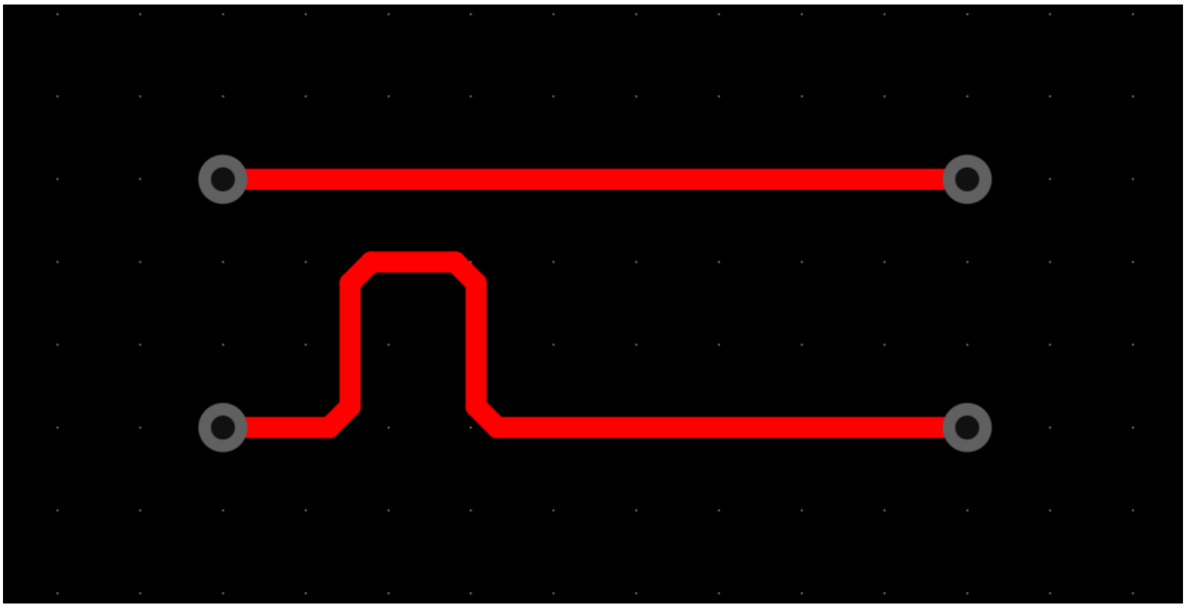
equal length tunning settings

After activating the equal length tunning, click TAB to bring up the equal length tunning property box



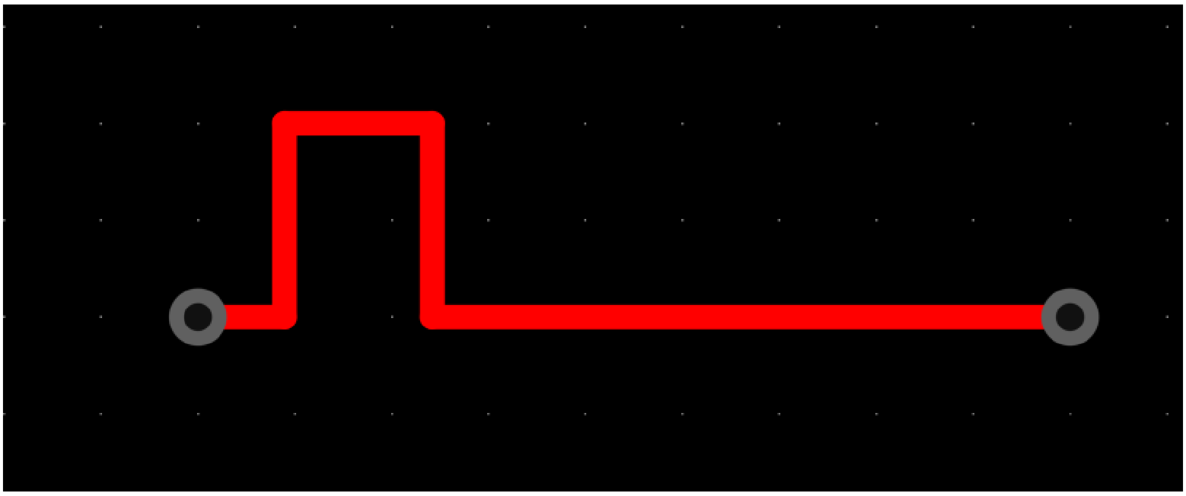
corner

- **Line 45°**: Set the corner of the equal-length tunning to an angle of 45°

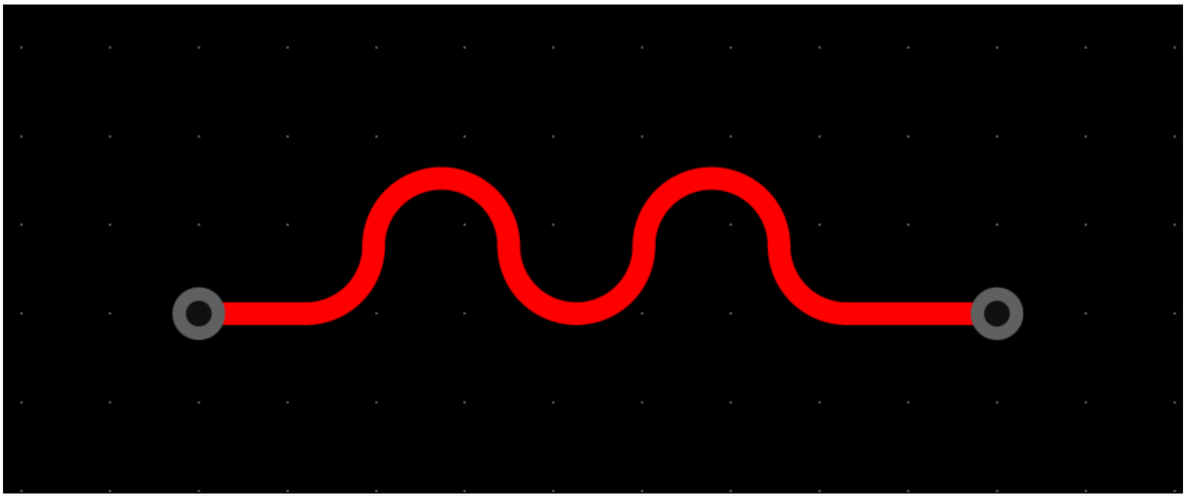


- **Lines 90 degrees**:

Set the corners of the equal-length tunnings to an angle of 90°



- **Arc 90 degrees**: The most commonly used equal length tunning setting, this angle setting is also called snake equal length.



routing constraints

Routing method: When the unilateral routing is of equal length, it will only be equal in one direction. Bilateral means that both sides of the line can be adjusted to the same length.

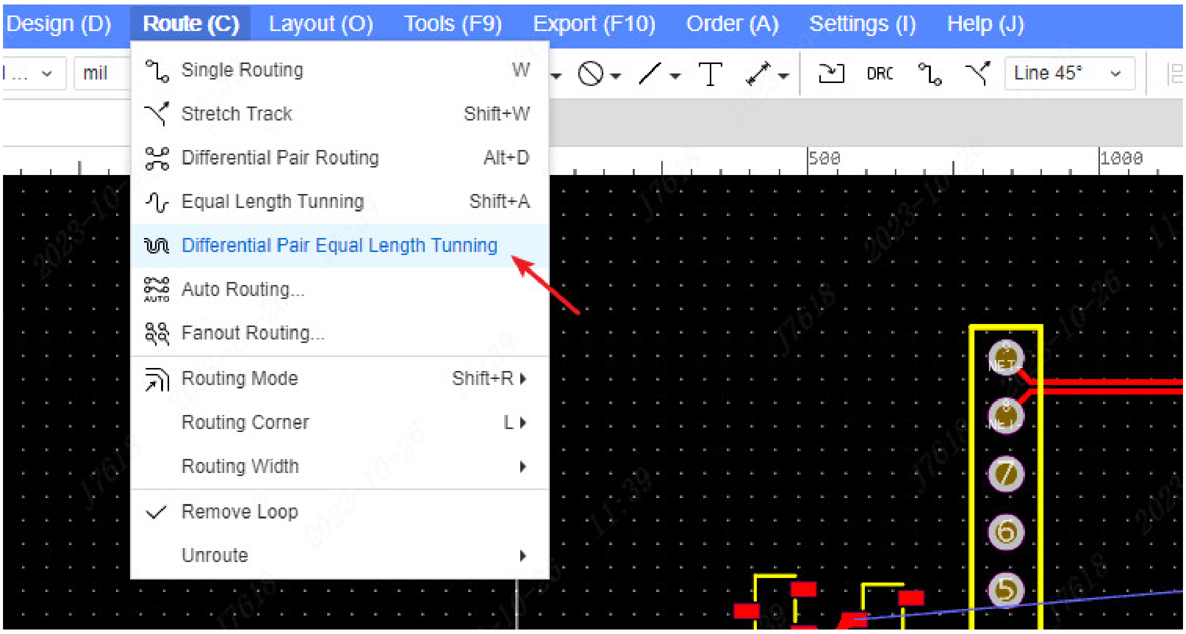
Introduction (W): equal length tunning of the direct width setting of the trace.

Minimum Amplitude (H): The amplitude setting of the equal-length trace, which can also be said to be the height setting.

Differential Pair Equal Length Tuning

Differential Pair Equal Length Tuning has important application value in high -speed signal design. It is to ensure the two signals in the differential pavage.performance.The specific principle of differential equivalence is to adjust the two network lengths of the differential signal together to make the internal error of the differential signal unchanged. At the same time, the overall adjustment of the length of the difference with other signals to achieve the purpose of multiple sets of signals.

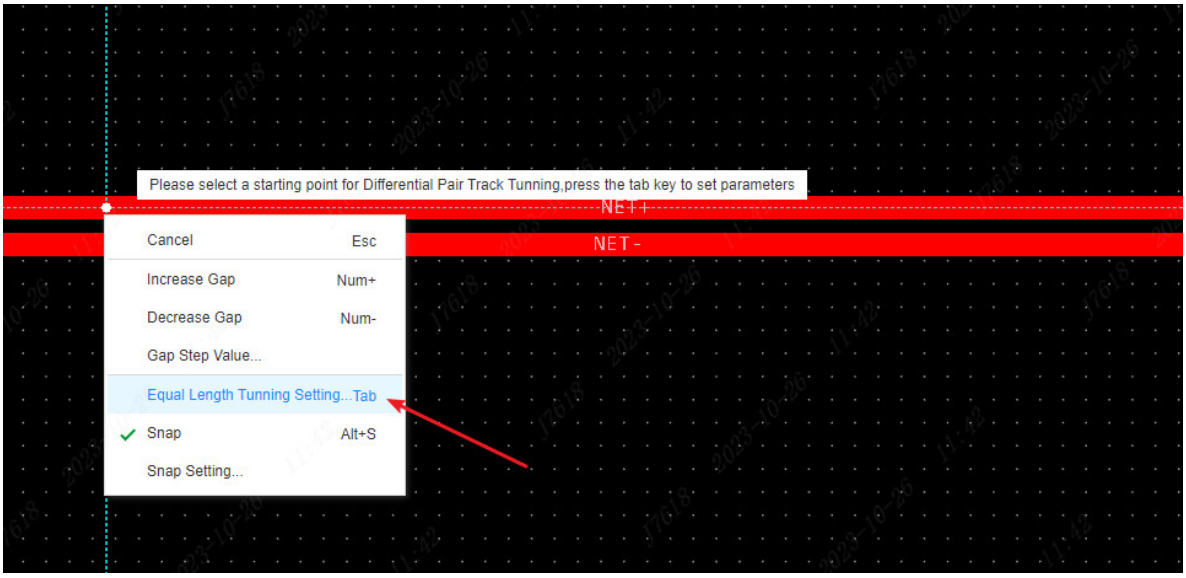
Tips:-On Menu - Route - Differential Pair Equal Length Tuning

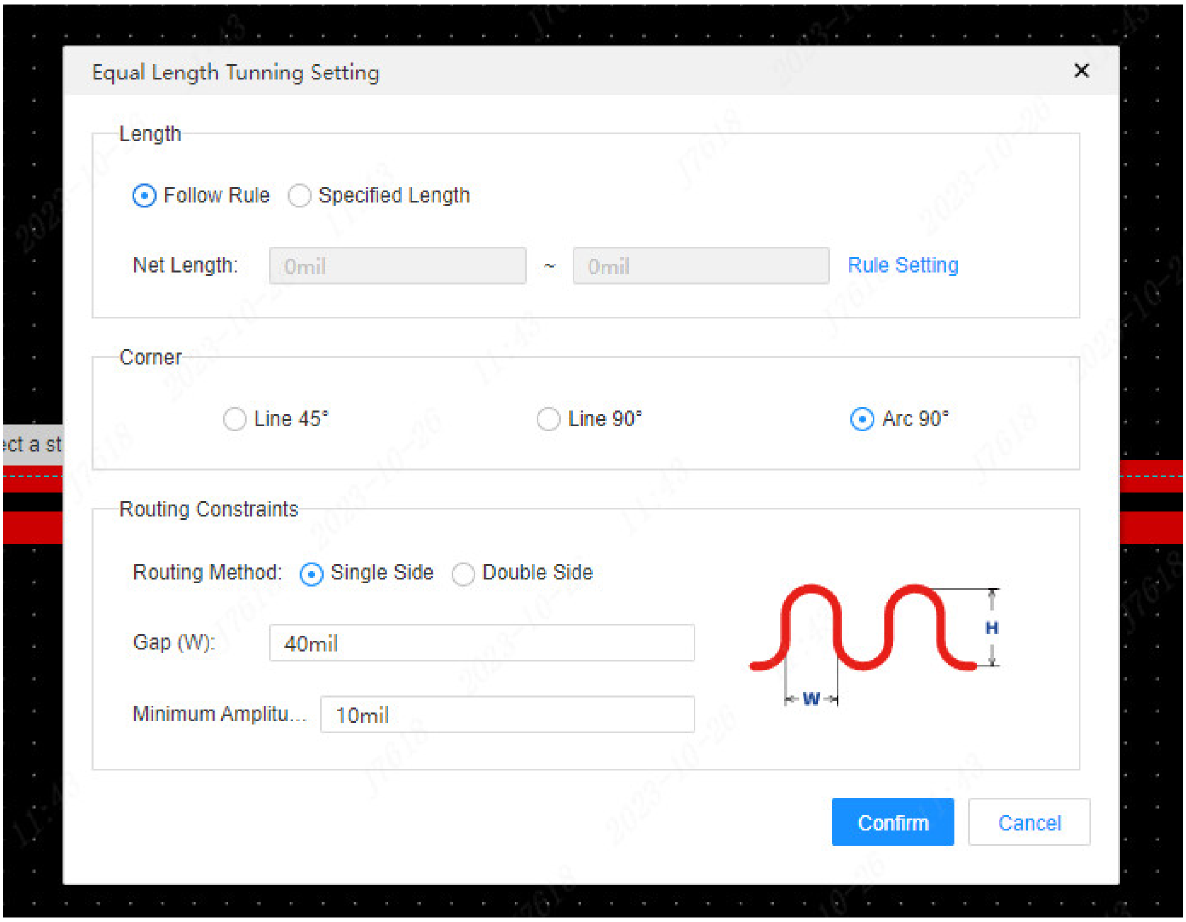


Equal length tuning setting

After activating the differential pair equal length tuning, the mouse becomes a cross cursor, waiting for the starting point of the selection of the difference between the difference

Click TAB or you can adjust the long adjustment attribute box with the long adjustment settings through the right -click menu.

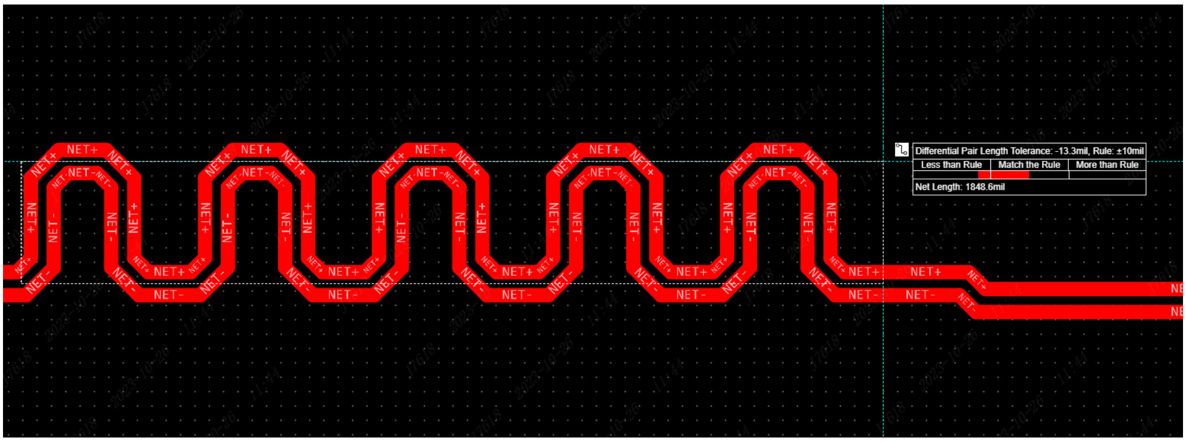




** Tips **: Increase the length input box to quickly calculate the required values by entering four operations expressions.For example, enter 5+2 Enter, it will automatically calculate 7.

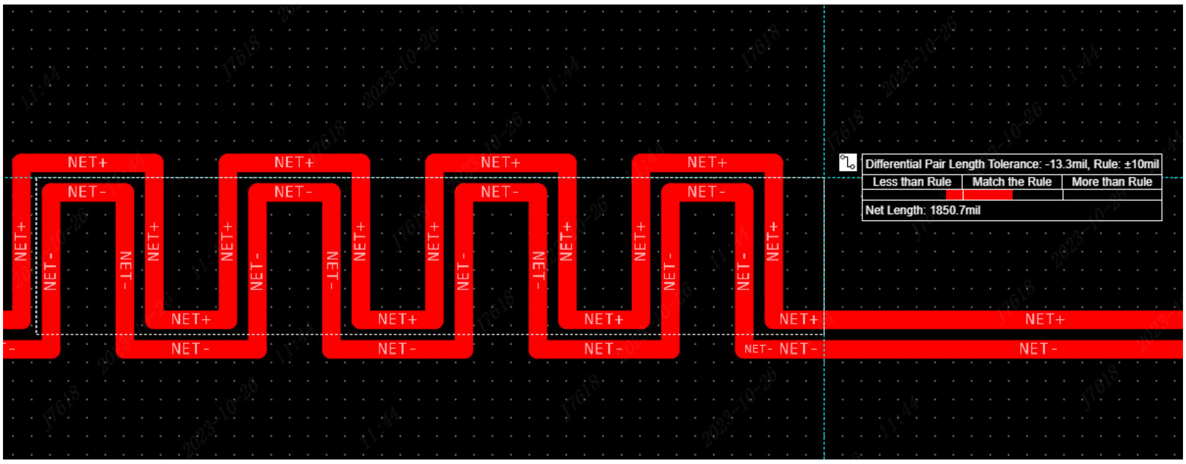
corner

- ** Line 45 degrees **: Set the angle of the differential long winding of the differential pair is 45 °

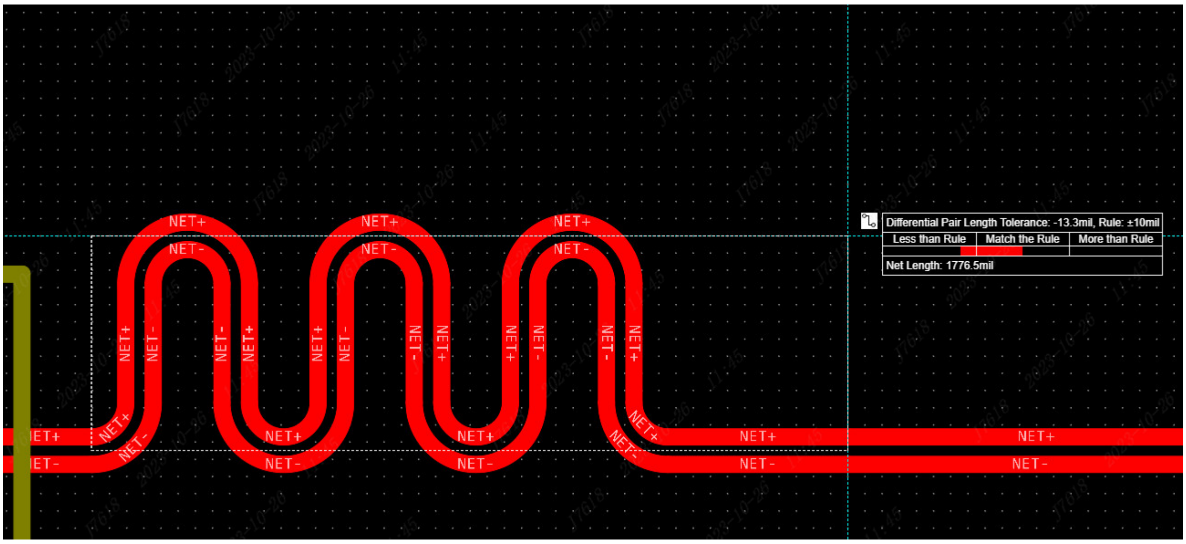


- **Line 90 degrees**:

Set the corner of the differential long winding of the differential long winding, the angle is 90 °



- **Arc 90 degrees**: The most commonly used long adjustment settings. This angle settings are also called snake -shaped equal lengths.

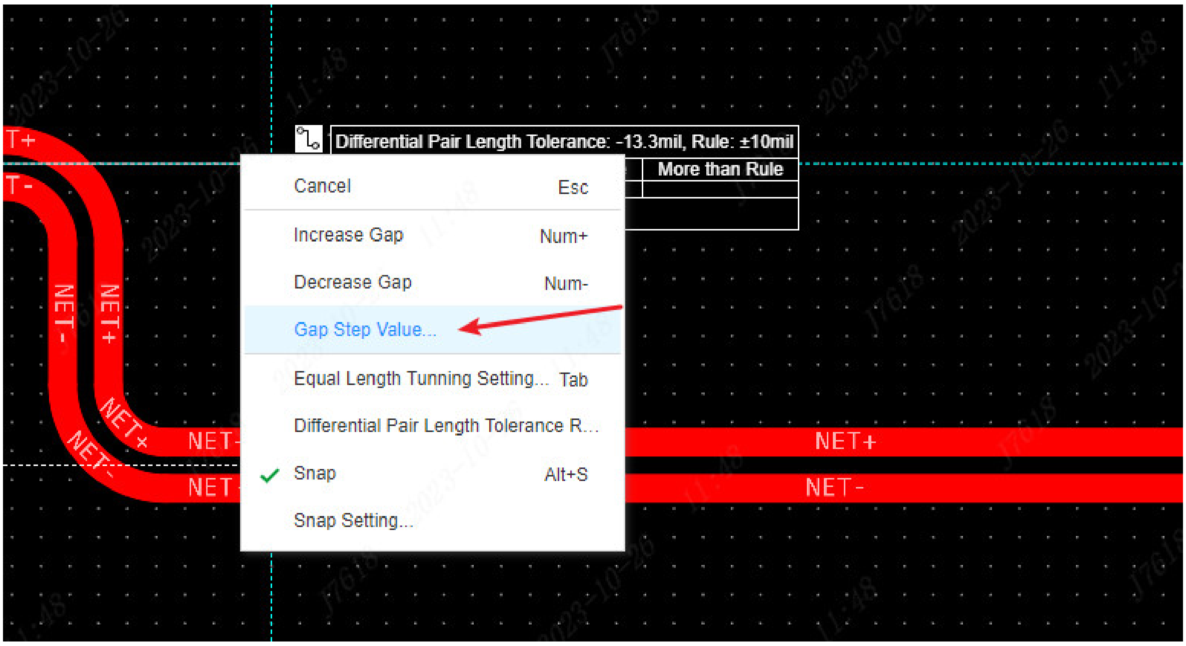


Route constraint

- **Routine method**: When the unilateral routing is waiting, it will only be equivalent in one direction.Both sides are equal adjustment on both sides of the lines.

- **Spacing (W)**: Wide -to -width settings between long adjustment.
- **minimum amplitude (H)**: The amplitude settings of the long wiring can also be said to be a high degree of settings.

Tips: When the equal adjustment is performed, you can fine-tune the spacing through the shortcut key Num+, Num-, and the gap step can be adjusted by right-click the menu.

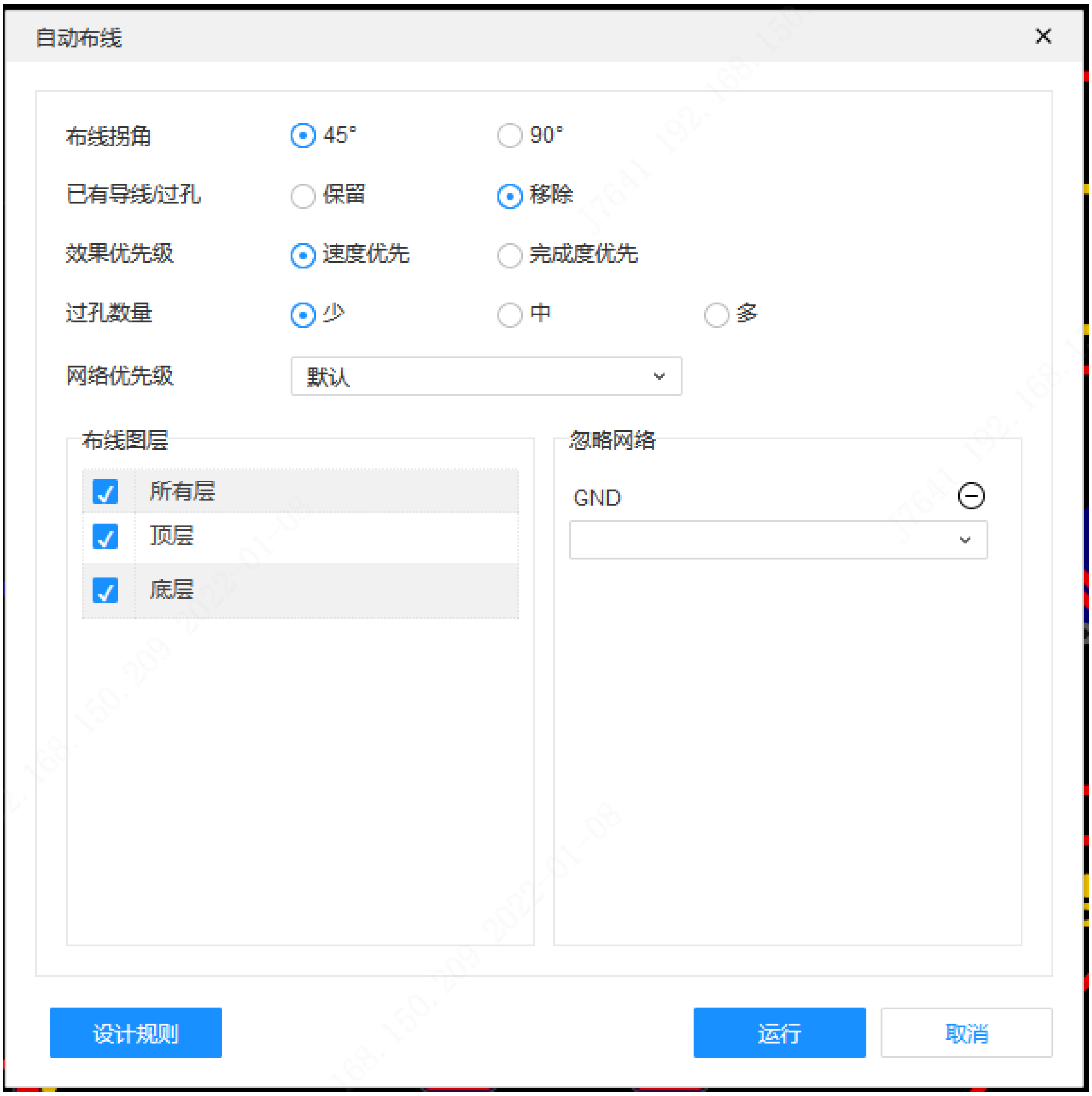


Auto Routing

EasyEDA Pro supports automatic routing. The current automatic routing effect is general and needs to be manually adjusted again. We will continue to optimize the automatic routing function in the future.

Entrance: Routing - Auto Routing

You can set some auto-routing related parameters:



Routing Corner: 45 degrees or 90 degrees. Generally, the corners of the wiring are 45 degrees. If the board requirements are not high, it will not affect the use of 90-degree wiring.

Existing Tracks/vias: keep or remove. When starting routing, existing wires or vias can be retained or removed. Reserved by default. Selecting Remove will automatically clear all tracks or vias for routing.

Effect Priority: Speed priority or completion priority. If you want to route quickly, choose speed priority, and automatically stop within a certain trial time, and some parts may not be routed; if you are not in a hurry, you can choose completion priority, and changing the option will try to complete the wiring.

Number of Vias: Low, Medium or More. This determines the number of vias that are automatically placed. The higher the number of vias, the higher the success rate of routing. Choose according to your level of acceptance. auto routing generates vias on its own.

Net Priority: Auto-routing from beginning to end according to the set net order of the nets. Different net ordering will affect the routing success rate and the final routing effect.

- Default: The order of the PCB nets directly read by the editor program, not sorted.
- Random: Randomly generate network sorting for routing.
- User-defined network name priority: Network names that do not start with $ have priority. Sort by first letter in natural increasing order.
- Pads from most to least: Sorts nets from most to least according to the number of pads the net contains.
- Pads from few to many: Sort the nets from least to most based on the number of pads the net contains.
- Pads Farthest to Nearest: Sorts nets from farthest to nearest based on the total distance of the mutual spacing of the pads the nets contain.
- Pads Nearest to Farthest: Sorts nets from near to farthest based on the total distance of the mutual spacing of the pads that the net contains.

Route Layer: Set the layer that needs to be routed.

Ignore Network: Set the network that does not need to be automatically routed, click the drop-down list to select the network to add, and click the Remove button to remove the network. For example, GND does not require automatic wiring, and copper connection is generally performed at the end.

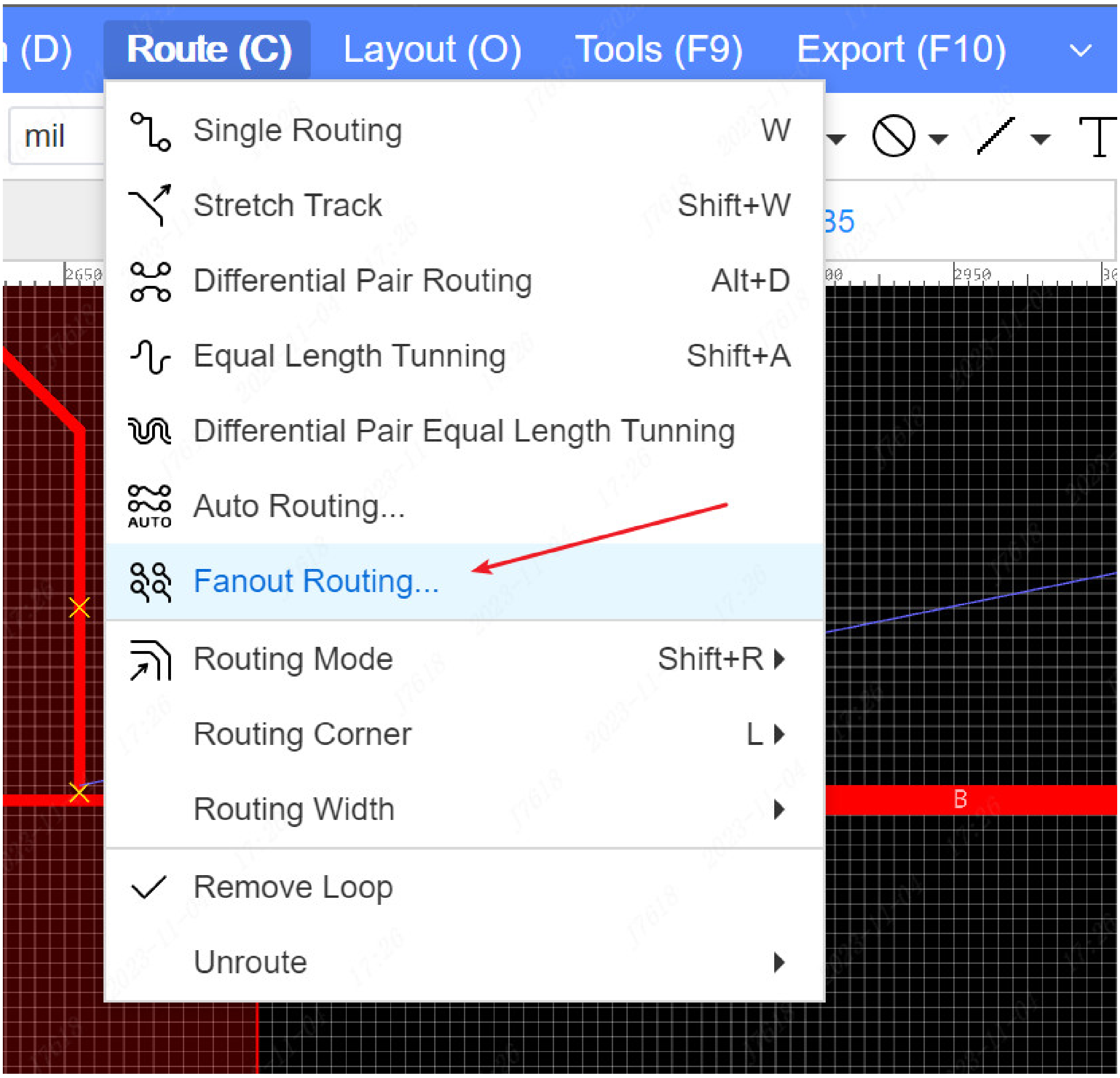
Design Rules: auto routing routes and places vias according to design rules. You can set the setting rules first and then perform wiring.

After clicking the "Run" button, automatic routing starts, and the routing process can be previewed in real time. If the over-routing angle is 45 degrees, the routing is carried out at a 90-degree corner, and it will be optimized to 45-degree routing at the end of the last surface.
Stop routing during the routing process, and the completed 90-degree routing and vias will be retained, and can be manually cleared if not needed.

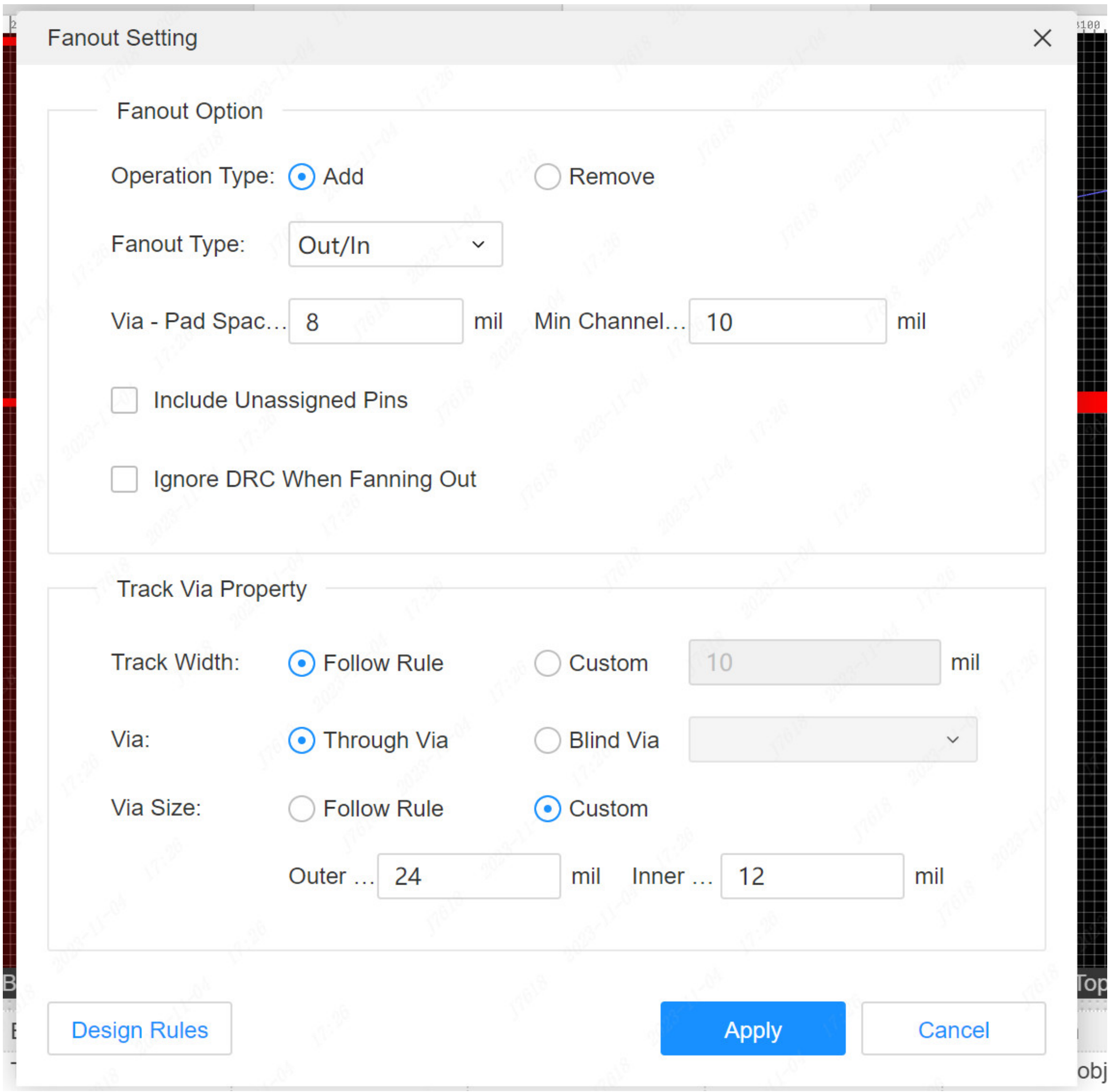
Fanout Routing

The fan is out or a fan. It refers to a short wire and punching from the chip pad. The purpose of this is mainly to give up the necessary space for different networks to facilitate subsequent inner wiring operations.For small IC devices of capacitors and resistors, it can be quickly fanned by manually copying and paste. For packaging with a large number of pins and small distances such as BGA, it takes a lot of time to manually fan out.The professional version provides automatic fan -out wiring function, and quickly batch fan outs for such packaging.

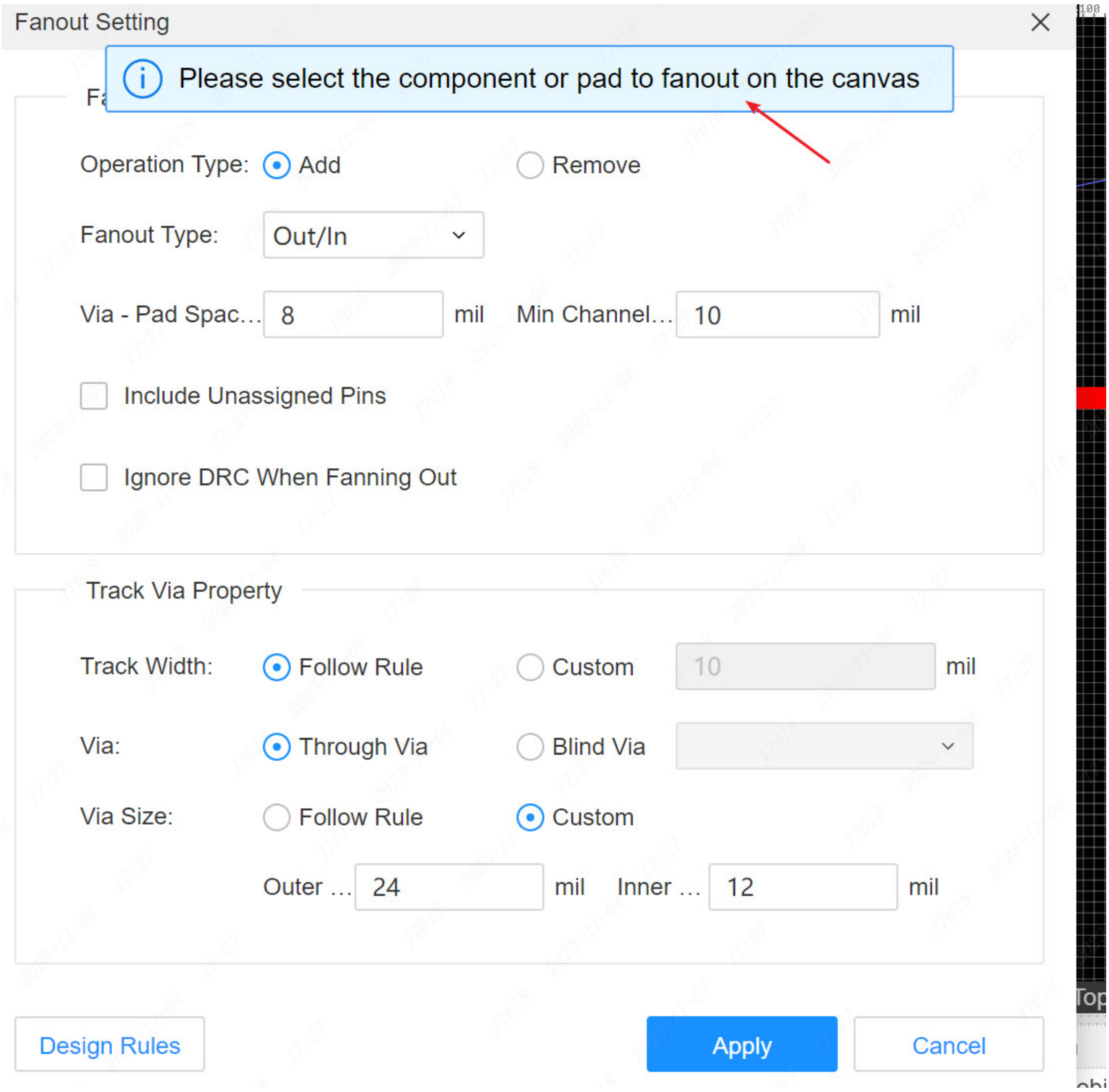
Steps: Top Menu-Route-Fanout Routing



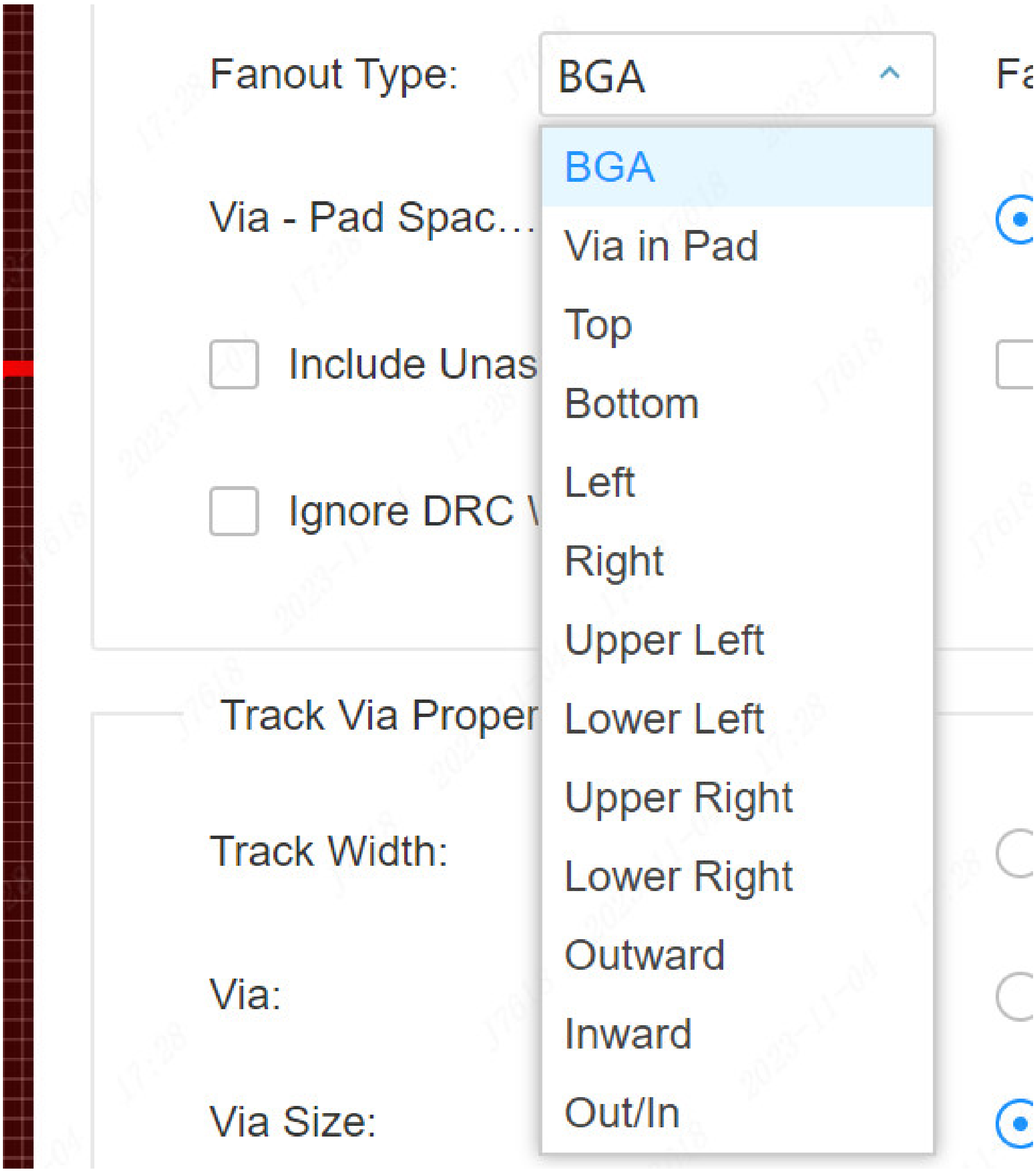
After clicking the menu, a fan settings dialog box appears.There are many fan settings in the dialog box, which is convenient for different types of packaging and requirements for fan.



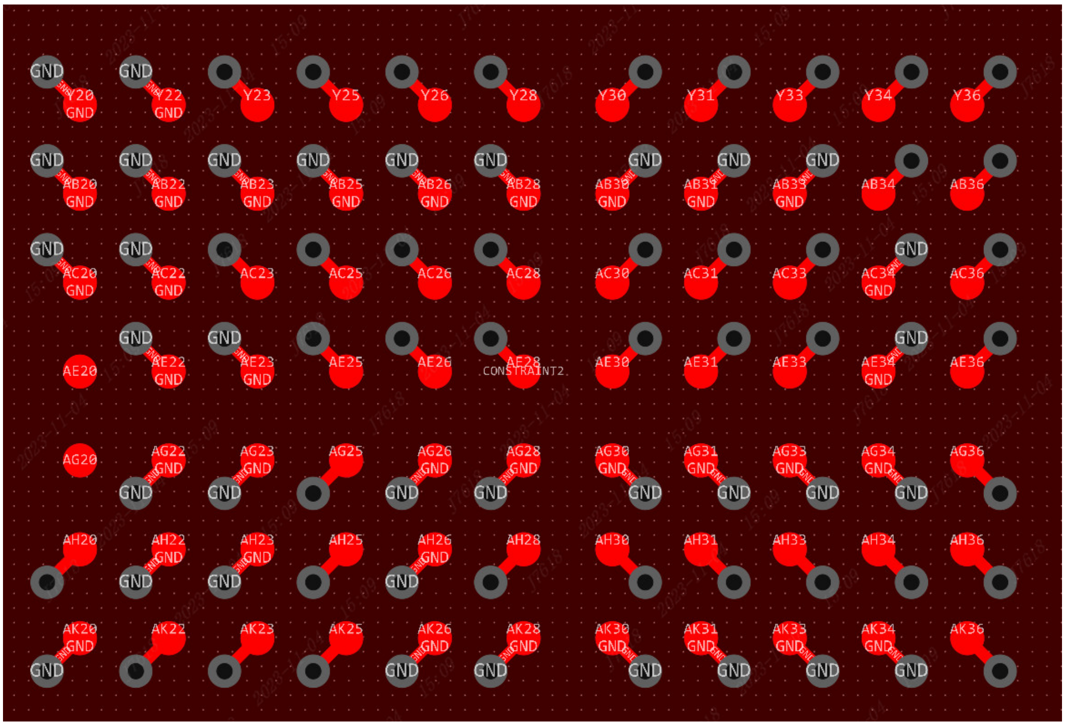
Steps: When the dialog box pops up, you can also drag and select the canvas.The fan can take effect on the components and pads. Before clicking on the application, you need to select the component or pile that needs to be made in the canvas, otherwise it will not be able to fan out.



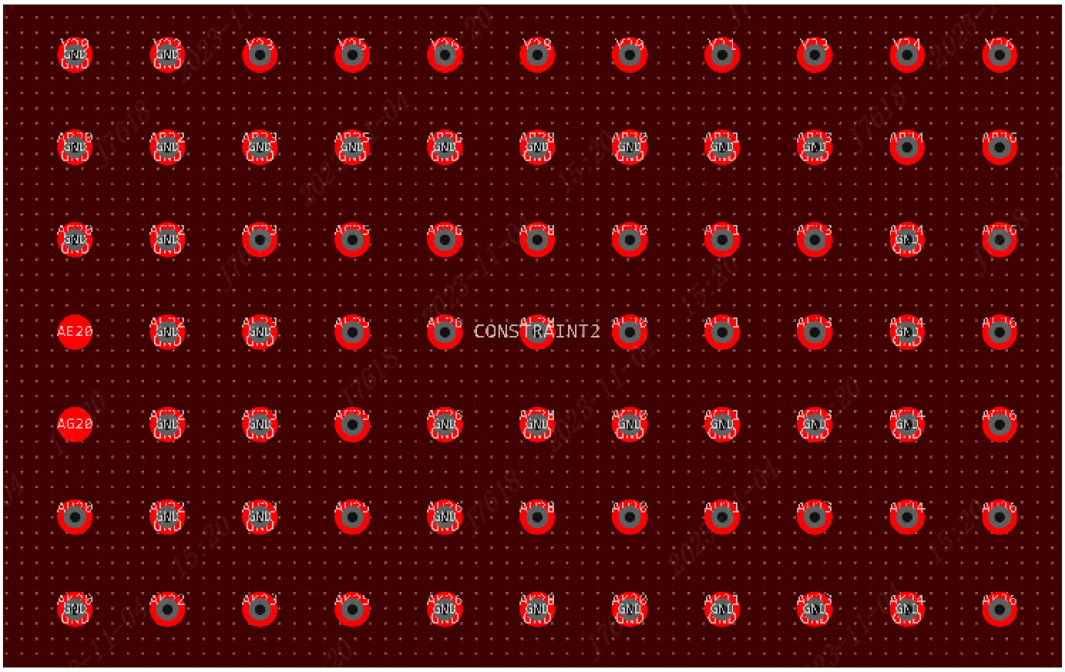
- **Fanout Option**
  - **Operation Type**: Divided into new and removal.If the wires or pores are connected on a certain pad, the fan needs to be reopened, and the removal operation needs to be performed first.
  - **Fanout Type**: easyEDA pro version provides a rich fan type option.



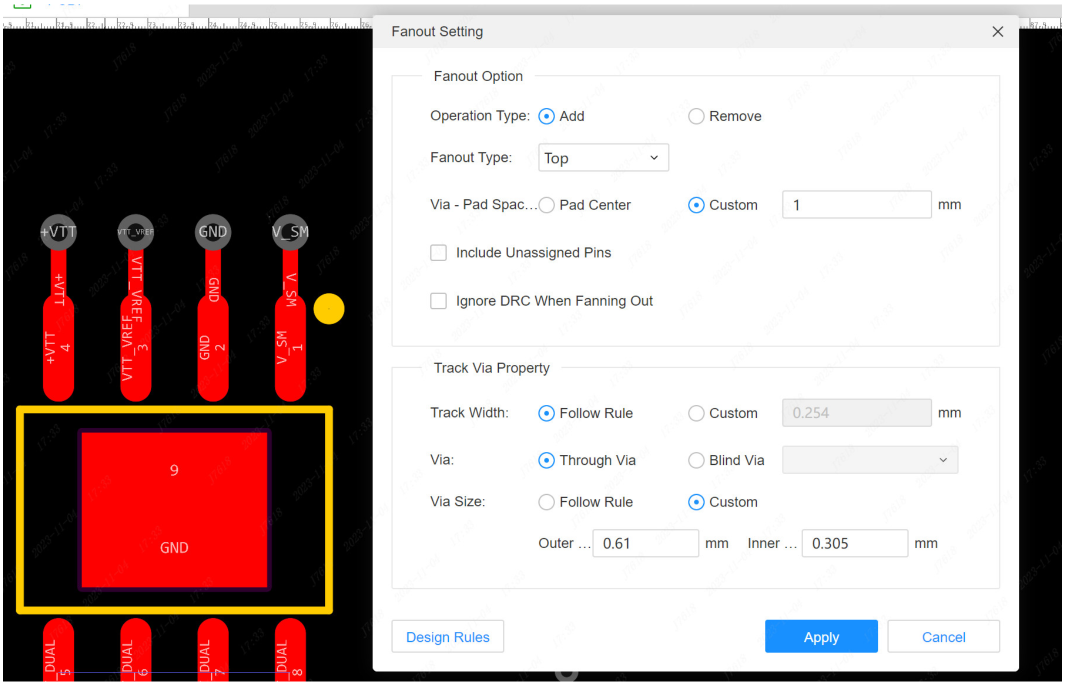
- **BGA**: This is the most commonly used fan, mainly for BGA packaging.Based on the BGA central line, the pads in the four regions in the upper left, lower left, upper right, and lower right are different directions.



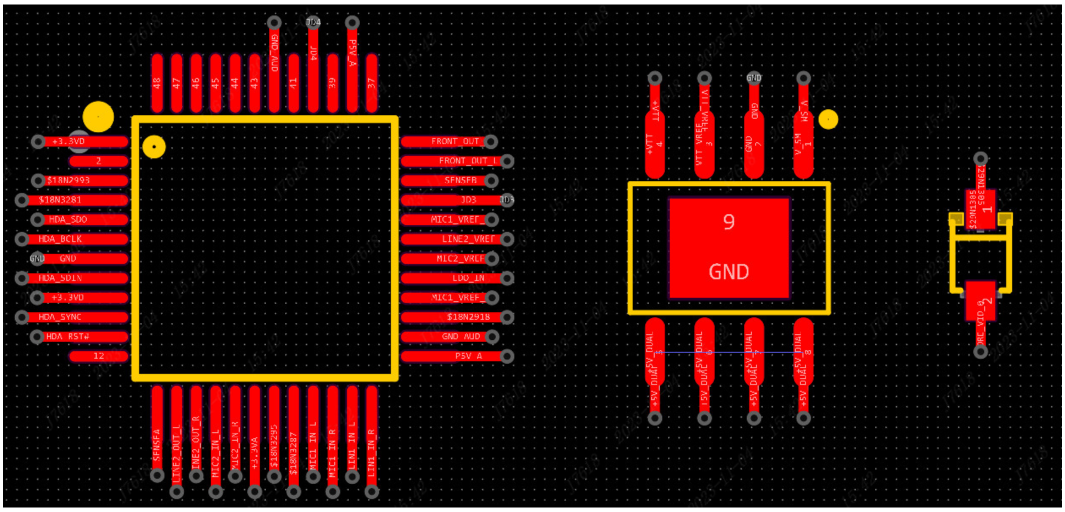
- **Via in Pad**: The hole in the disk is to play a perforated directly in the center of the pad, without adding a wire, the "resin plug hole+electroplating cap" provided by Jia Lichuang can make the pads not see the holes at all, and the performance spike spikes."Covering oil" and "over -pore oil".



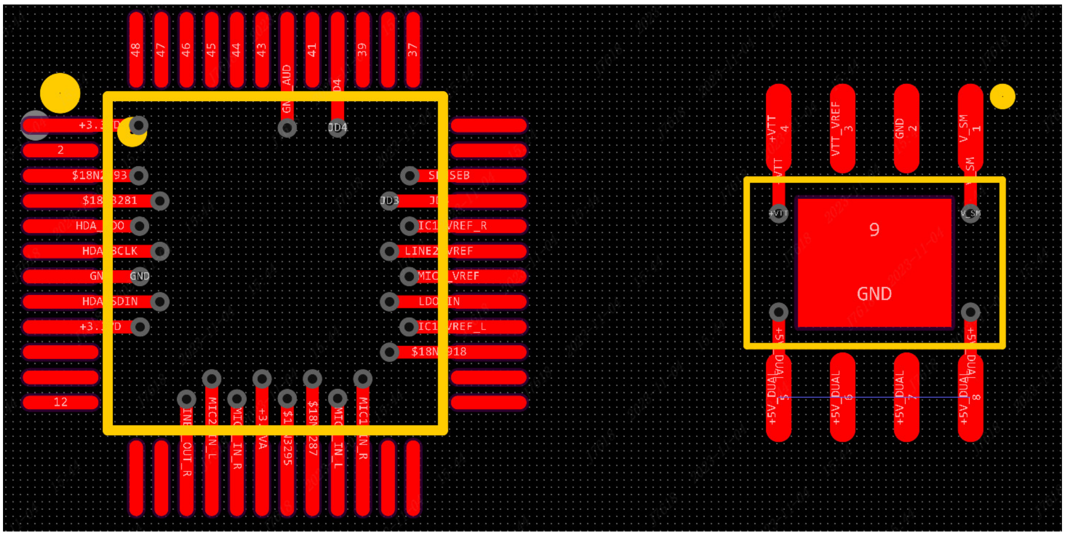
- **Top**: The direction of the fixing fan is up.



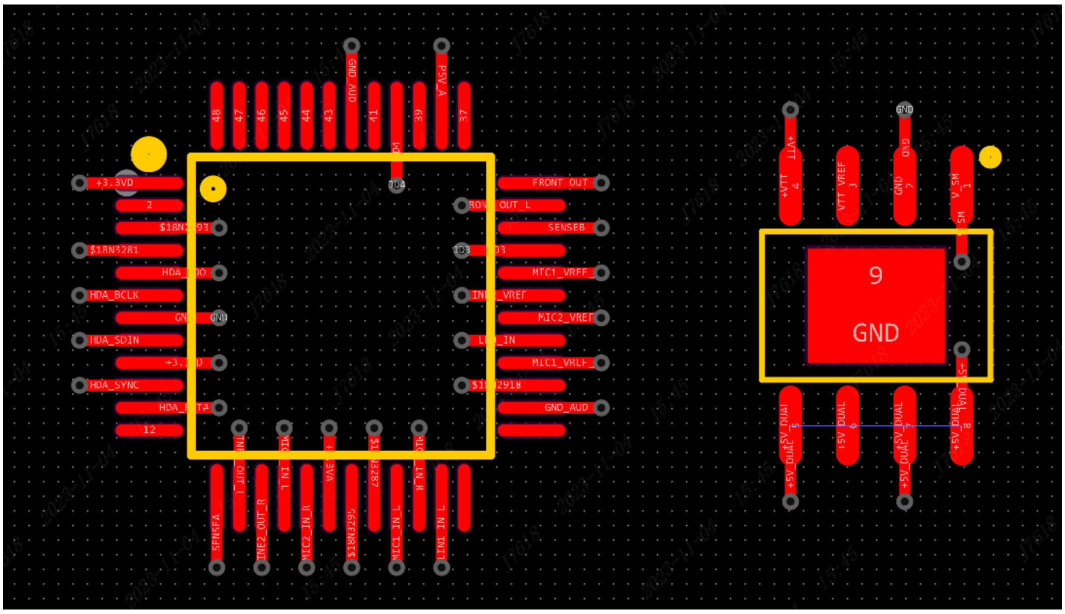
- **Bottom**: The direction of the fixing fan is down.
- **left**: The direction of the fixed fan is to the left.
- **right**: The direction of the fixed fan is to the right.
- **Upper left**: The direction of the fixed fan is upper left.
- **Lower left**: The direction of the fixed fan is the bottom left.
- **Upper right**: The direction of the fixing fan is upper right.
- **Lower right**: The direction of the fixing fan is the bottom right.
- **outward**: It is mainly aimed at the packaging of rectangular or bilateral pads in the three rules as shown below. For example, if LQFP, the direction of the pads will be automatically detected when it is fanned, and it will be fanned to the outside of its location.



- **Inward**: Like the packaging of the outward type, the direction of its fan is inside the position where the pad is located.



- **Out/In**: Like the packaging of the outward type, the direction of its fan is one inward in order.



```
- **Fanout direction**: This option will appear when the type of fan is out is BGA. If the BGA packaging rotates 45 degrees, the direction of the fan can be modified to 90 °.

![图 5](../_images/route-fanout-routing_20231104_151417.jpg)
```

```
- **Via - Pad Spacing**

   That is, the length of the fan out of the wire is set, and the pores here are the air gap distance.

    ![图 13](../_images/route-fanout-routing_20231104_155115.jpg)
```

- Pad center: Except for outward, inward, outward, and inward types, other fan-out types can be set to the pile-pile spacing center as the pad.When the fan is made, the software will automatically detect the minimum gap of the adjacent pads, and the fixed fan is in the middle of the two pads.

```
    ![图 14](../_images/route-fanout-routing_20231104_155605.jpg)

    - Custom distance: Set a fixed fan length.

    - Mini channel space: three types of fans, outward, inward, outward, and inward, this option will appear. Set up the pores of the two adjacent channels.The fan of the adjacent pad.

    ![图 15](../_images/route-fanout-routing_20231104_160032.jpg)

- **Include Unassigned Pins**: If this option is checked, the pads of the empty network will also be fanned.

- **Fanout Outer 2 Rows of Pads**: This option is only displayed when the fan is selected for BGA. If this option is checked, the two -line pads of the outermost line of BGA packaging will also be fanned, otherwise it will not be fanned.Essence

![图 16](../_images/route-fanout-routing_20231104_161139.jpg)

- **Ignore DRC When Fanning Out**: After checking all the pads of the pads will ignore the DRC to fan out. You can check the effect of the fan first. After adjusting the relevant parameters, then cancel the checked fan.Those fans and perforated fans that violate the DRC will be removed.
```

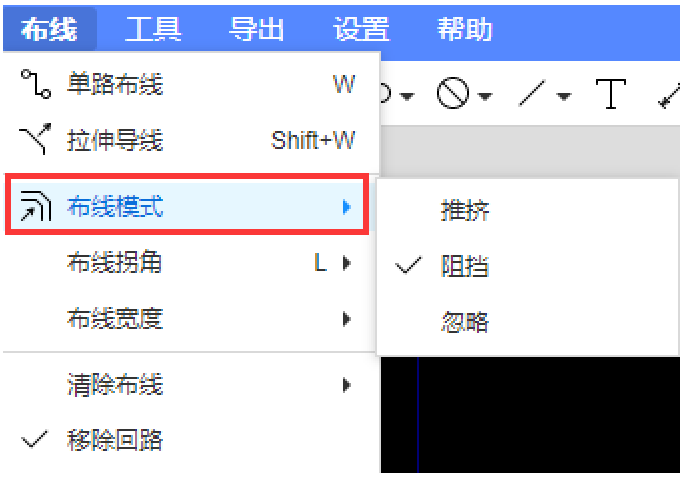
- **Track Via Property**
  - **Track Width**: Divided into follow -up rules and customs, set to follow the rules, the software will be qualified according to the default wire width of the wire in the design rules.
  - **Via**: You can set the type of punching a punching hole or blindly buried holes. If the blind burial hole is set, the blind buried hole type needs to be set in advance in the design rules.
  - **Via Size**: Divided into follow -up rules and customs, and set to follow the rules. The software will punch the default hole size of the wire in the design rules.

Routing Mode

Switch routing mode when routing

Steps:

- Routing via right-click menu or top menu - Routing mode switching



Push: When turned on, other line positions can be pushed under the wiring

routing-push

Block: When enabled, the wire will be blocked when it encounters a line in routing mode

routingblock

Surround: When enabled, when the PCB trace encounters obstacles, it will automatically bypass the obstructed element routing

routing wrapping

Ignore: When enabled, ignore routing rules

wiring ignore

Unroute

Click Cancel in the right-click menu, or the shortcut key ESC, to exit this routing, and the completed routing will not be deleted

Finish Routing

- Click Finish in the right-click menu, or enter the shortcut key to complete the wiring, add an inflection point according to the current cursor position and complete the wiring
- `ctrl` +click with the right mouse button or the left mouse button, you can also directly complete the wiring, add an inflection point according to the current cursor position and complete the wiring

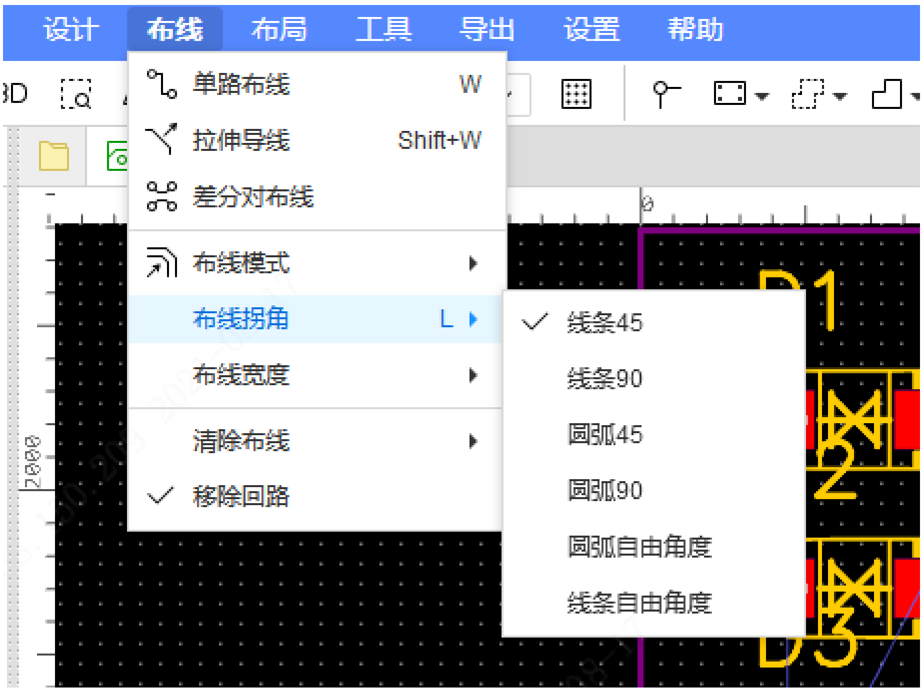
Routing Corner

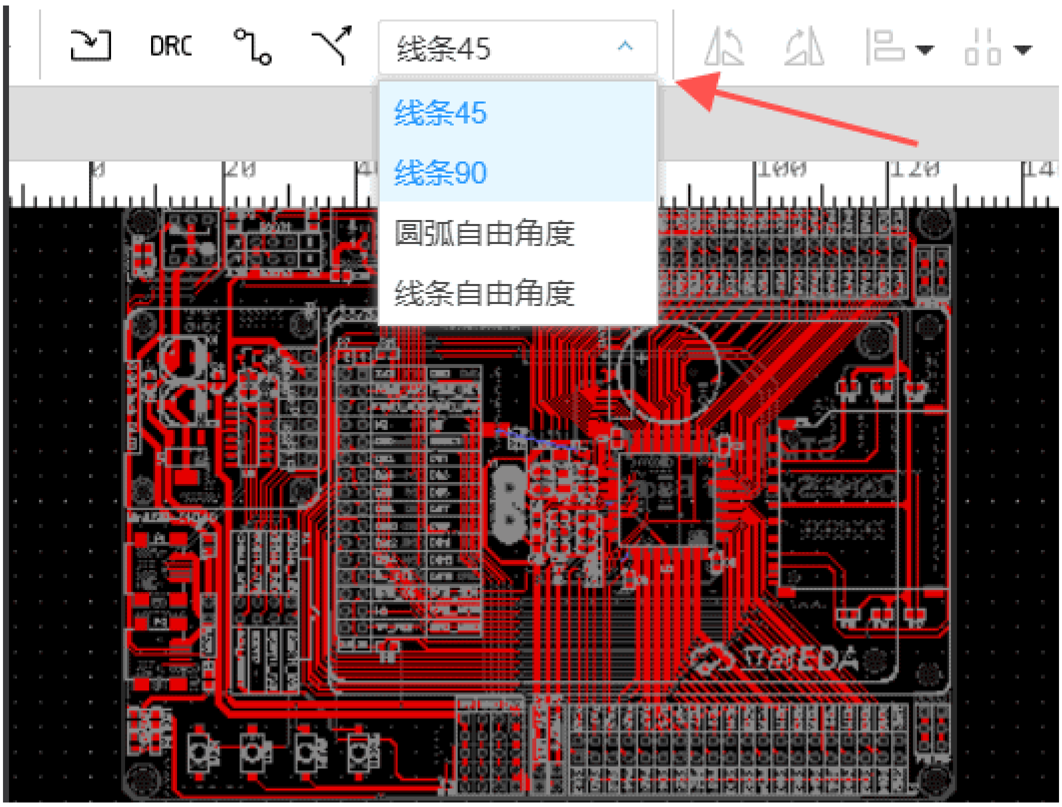
switch routing corners

Switch the angle of the trace during wiring, support 45°, 90°, arc 45°, arc 90°, arc free angle, and line free angle.

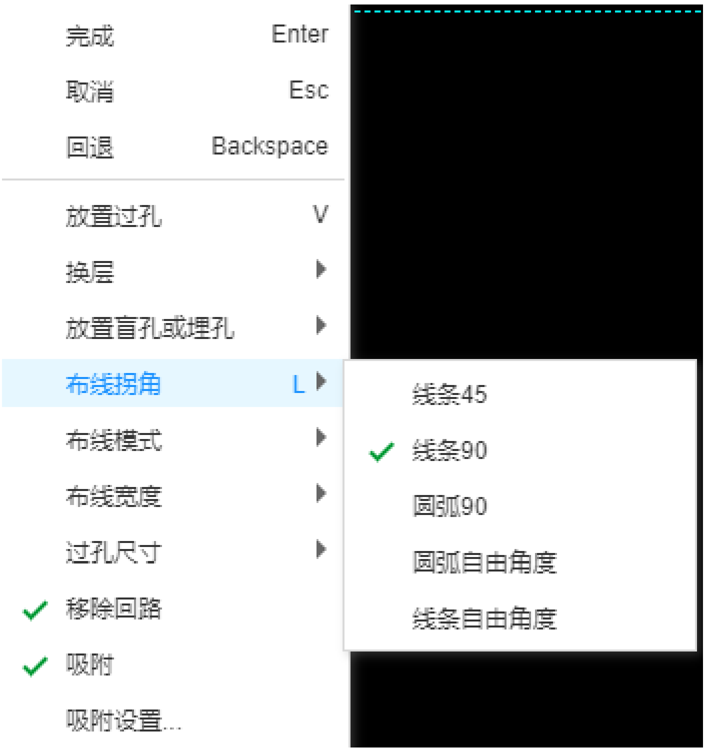
How to operate:

- Top Menu - Routing - Routing Corners
- Routing corners on the top toolbar to quickly switch corner modes
- Change the angle of the wiring by pressing the shortcut key L in the wiring mode





In the process of routing, click the right-click menu to quickly switch the corner mode through the routing corner menu



Routing Width

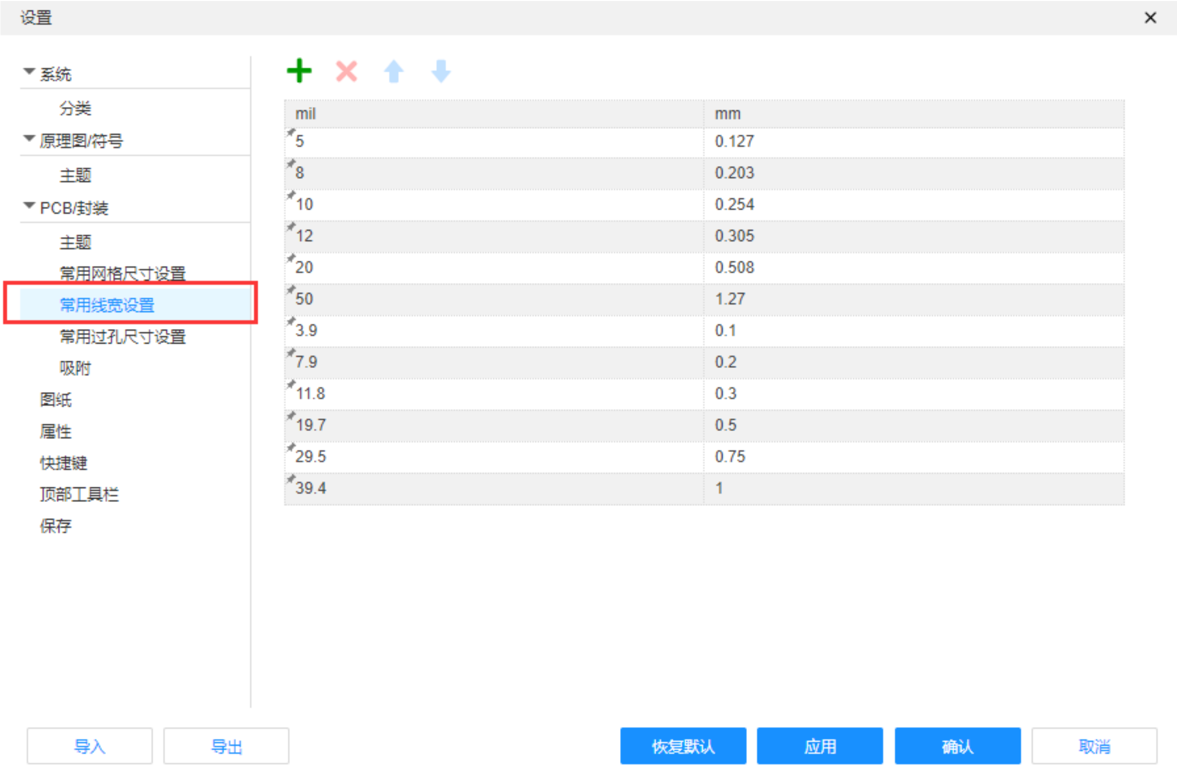
Toggles the set wire width during PCB routing

Routing width setting method:

- Top Menu - Route Width or Right Mouse Button Menu - Route Width
- Shortcut `shift` + `W`


布线
工具
导出
设置
帮助
单路布线 W
拉伸导线 Shift+W
布线模式
布线拐角 L
布线宽度
清除布线
移除回路
跟随规则
规则设置...
5mil/0.127mm
8mil/0.203mm
10mil/0.254mm
12mil/0.305mm
20mil/0.508mm
50mil/1.27mm
3.9mil/0.1mm
7.9mil/0.2mm
11.8mil/0.3mm
19.7mil/0.5mm
29.5mil/0.75mm
39.4mil/1mm
常用线宽设置...
输入宽度...

- Rule settings. Click to pop up the Design Rules dialog box, where you can set the rules of the wire
- List of common line weights. When wiring, use the right-click menu to quickly switch the common line width, click the common line width setting, and the system setting dialog box will pop up, where you can maintain the list of common line widths
- Enter the width. After clicking, a pop-up window will pop up, enter the line width to be modified and press Enter, the next wire will be changed to this line width
- Use the command "W+" to quickly switch the line width to the specified value when routing
- Use shortcut keys `Shift` + `W` to switch common wire widths when routing.



Unroute

Clear the track drawn in the PCB.

clear wire method

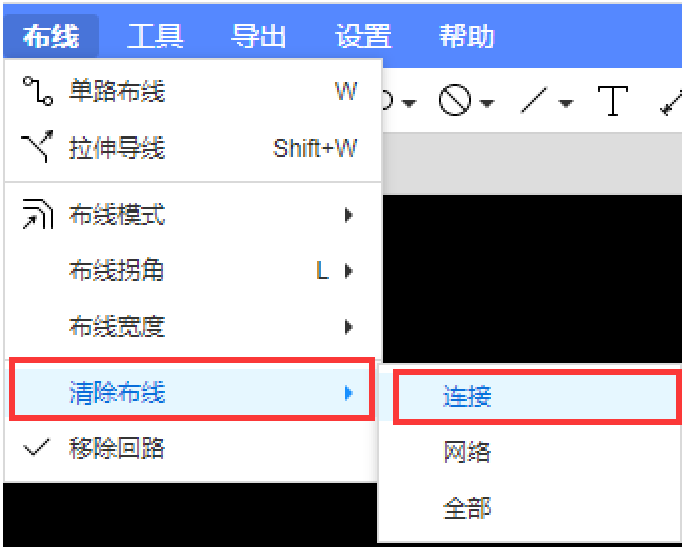
- Top Menu - Routing - Clear Routing
- Select wire - right mouse button - clear routing

Unroute - connect

Wire removal of the same pad

Steps:

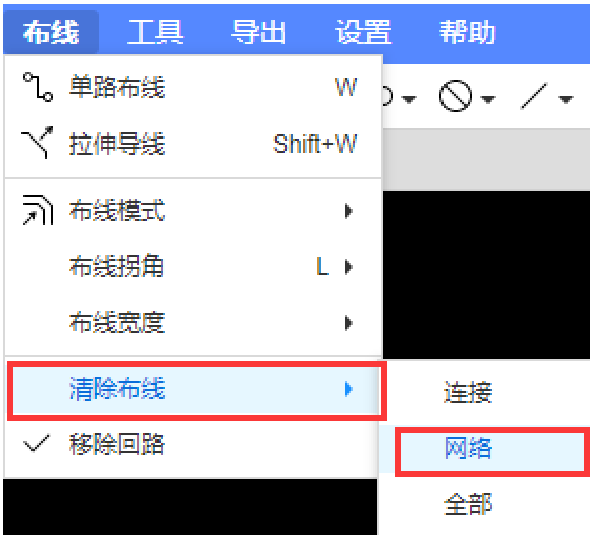
- Select the wire to clear - Top Menu - Routing - Clear Routing - Connect
- Select the wire to clear - Right mouse button - Clear Route - Connect



Unroute - net

Clear the track of the same net.

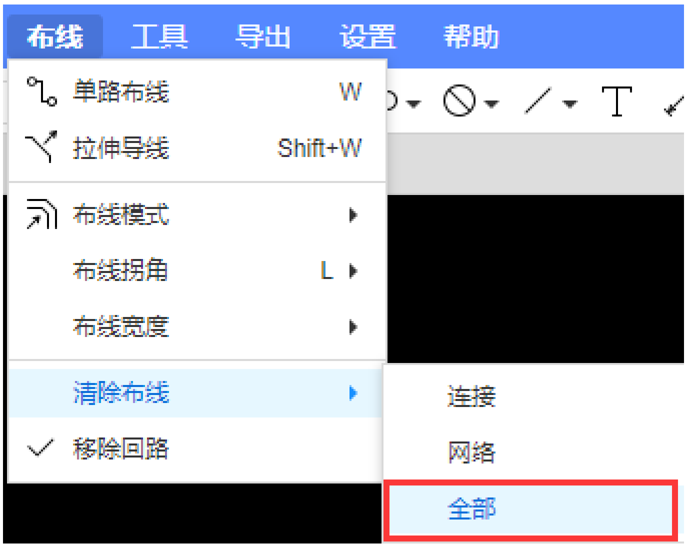
- Select track to clear - Top Menu - Routing - Clear Routing - Nets
- Select the wire to clear - right mouse button - clear routing - net



Unroute - all

Clear all the track drawn in the PCB.

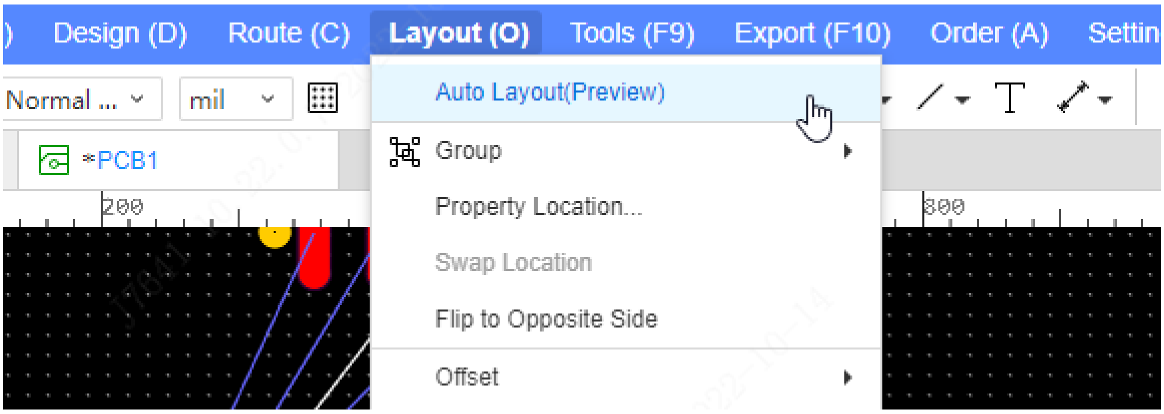
- Top Menu - Routing - Clear Routing - All
- select any wire - right mouse button - clear routing - all



Auto Layout

EasyEDA supports simple automatic layout function, currently only the experience stage (preview version), it will continue to iteratively update.

Entry: Top Menu - Layout - Auto Layout



When the subsequent development is completed, it will be automatically laid out according to the position of the components in the schematic diagram, the connection relationship between the components, the size of the PCB board, and the spacing rules.

If a board outline is drawn, the auto layout will automatically place the components according to the board frame. If the board frame is not drawn, the auto layout will automatically generate an optimal rectangular frame and automatically layout.

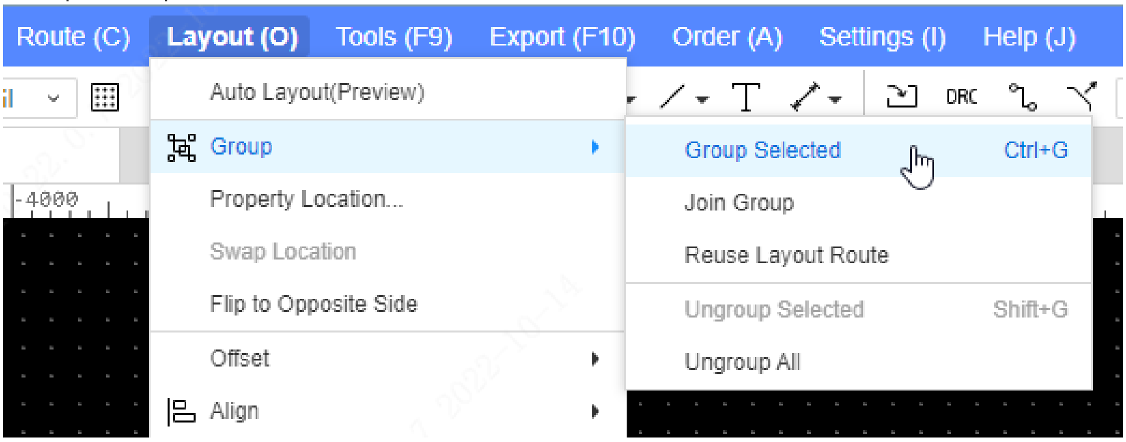
Note: The automatic layout will be laid out according to the built-in rules, and cannot completely replace the manual work. Please pay attention to check whether it meets the design requirements after the automatic layout, and may need to manually correct the layout.

Group

The combination function supports the combination of any image element as a whole. After the combination is combined as a whole, the relative position of each image element in the combination can be guaranteed to remain unchanged when dragging, rotating and other operations are carried out.

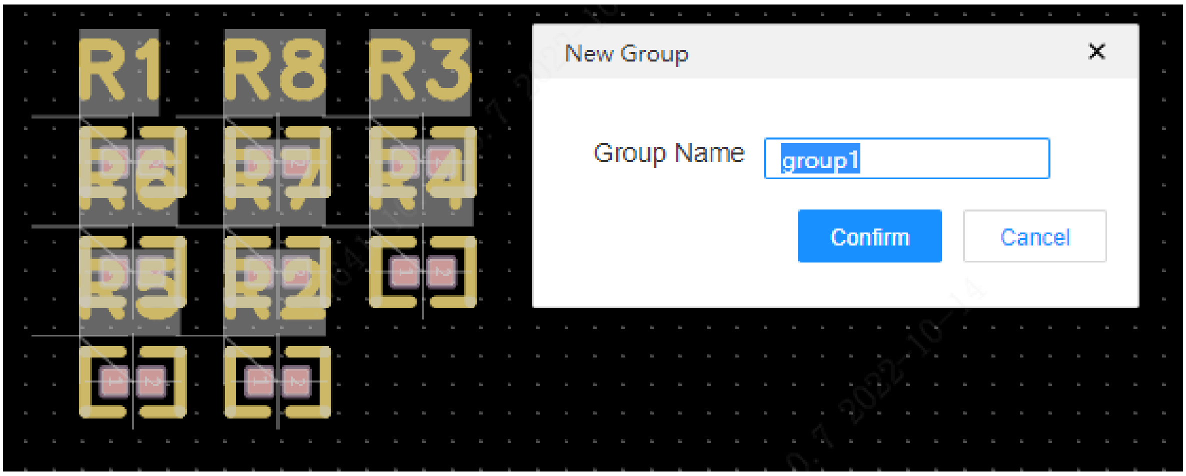
group selected

- Select the devices that need to be combined (must be two or more) - Top Menu - Layout - Group - Group selected



- Select the device to be combined - Right mouse button - Combine - Combine selected
- Select the device to be combined - Shortcut `Ctrl` + `G`

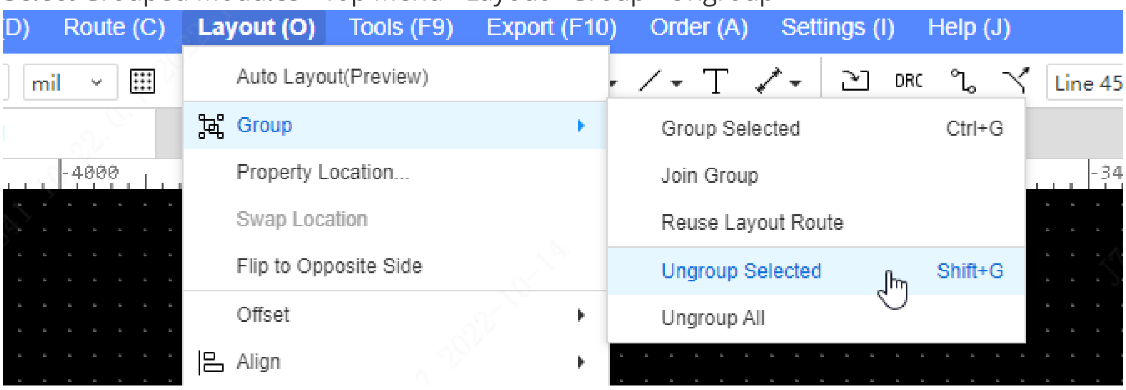
When you add a group, you can give the group a custom name in the window



Ungroup

This menu allows you to cancel the group after selecting all the primitives in the group

- Select Grouped Modules - Top Menu - Layout - Group - Ungroup

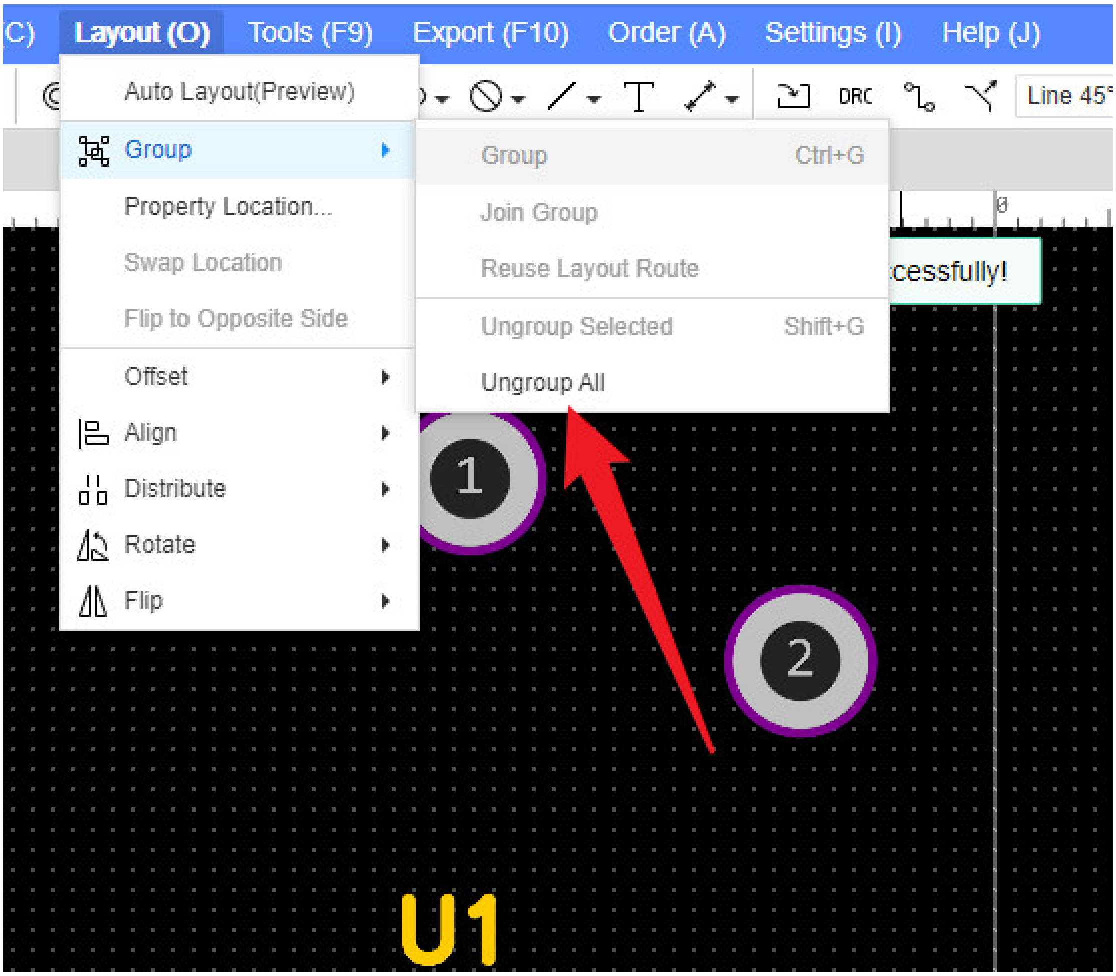


- select combined modules - right mouse button - ungroup
- Select combined modules - Shortcut `shift` + `G`

Ungroup All

Cancel all combined modules

- Top Menu - Group - Ungroup all



Join the group

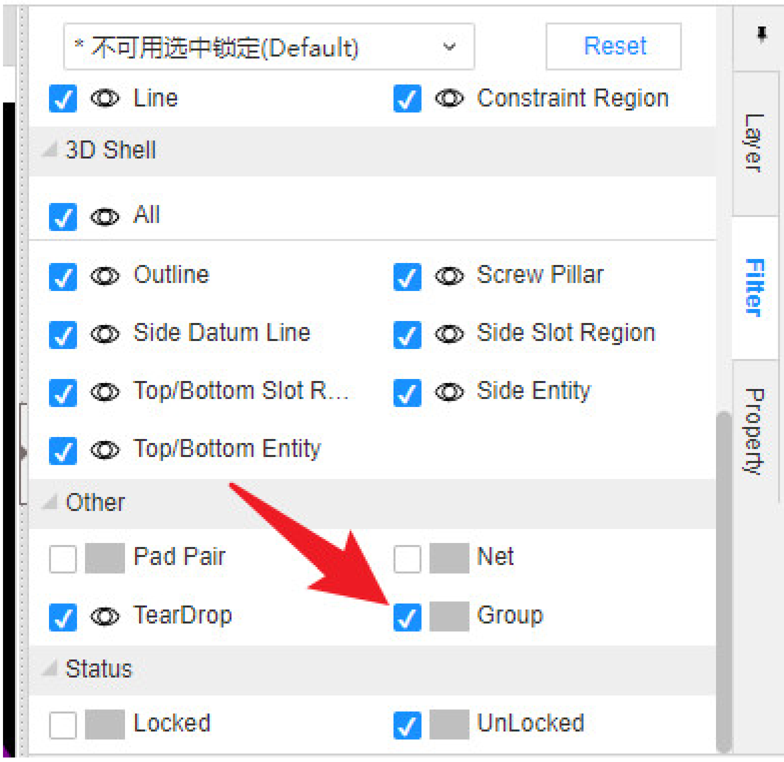
Devices that are not in the group are added to the group.

- Select device - Right mouse button - Add group, select the group module to be added.

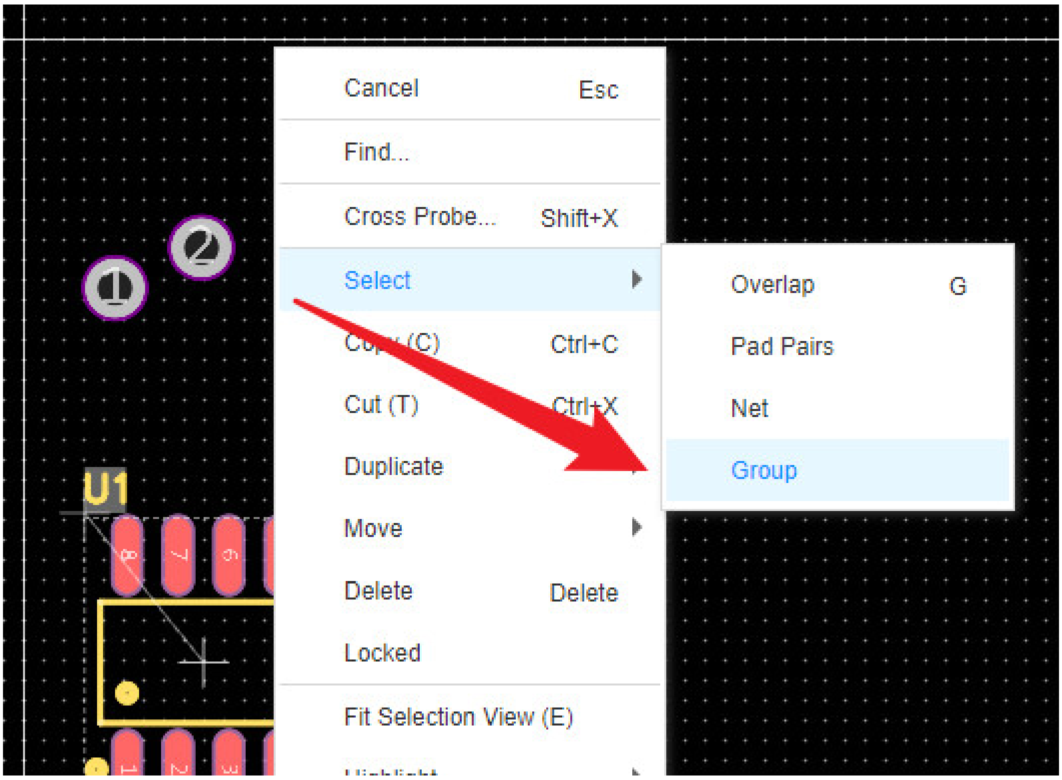
How to select a combination

You can select a combination in the following five ways

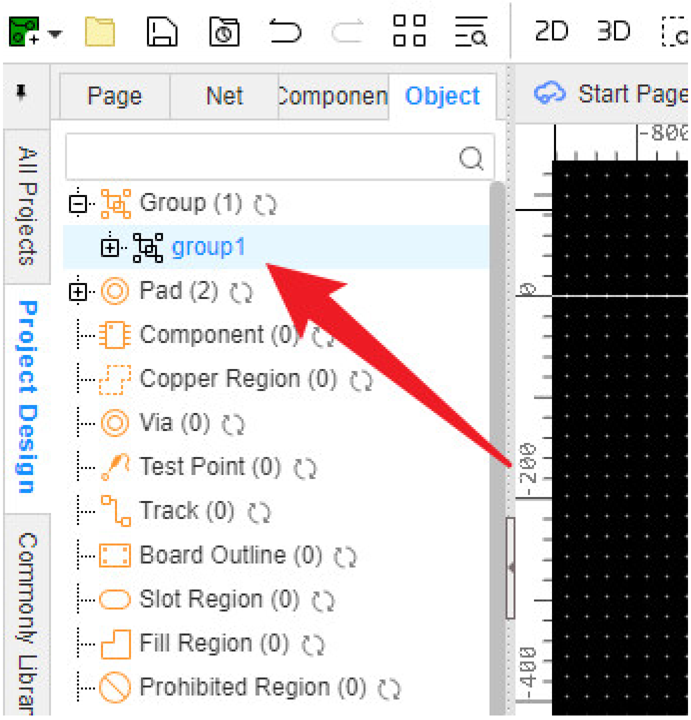
1. Select the combination option in the right filter panel,Then click one of the primitives in the combination on the canvas



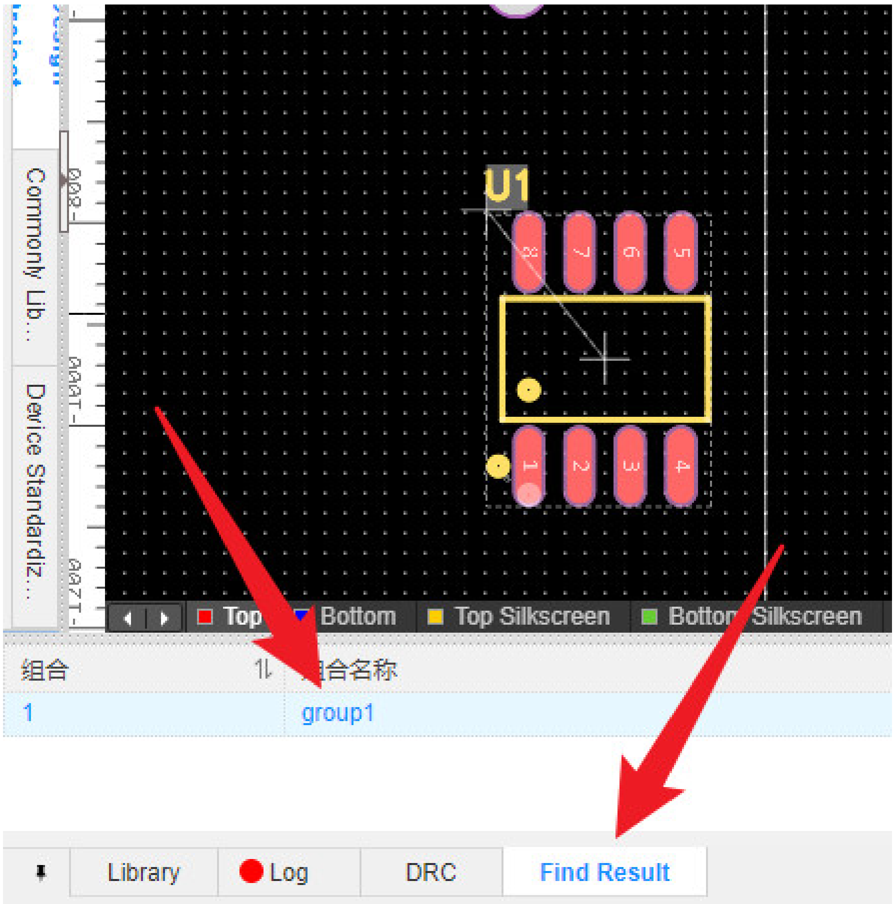
2. Select primitives within a combination,Then right-click - Select - Combine



3. Click on the concrete group name in the left object tree



4.Search combination,Then click Combination in the search results



5. Select the components in the assembly,By clicking on `tab` Key to toggle the selection range to the combination

After combination selection,You can modify some properties that are common to primitives in a combination in the properties panel on the right

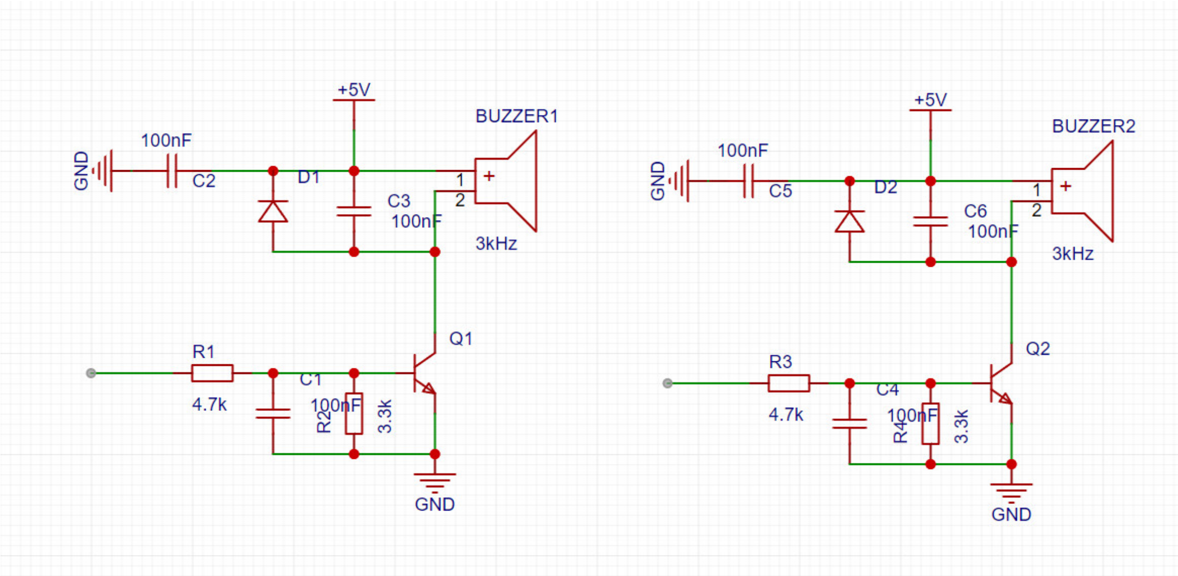
Multiplex layout

EasyEDA Pro supports a very fast way to repeat the layout and routing of similar circuits, no need to use copy and paste and no need to lay out one by one.

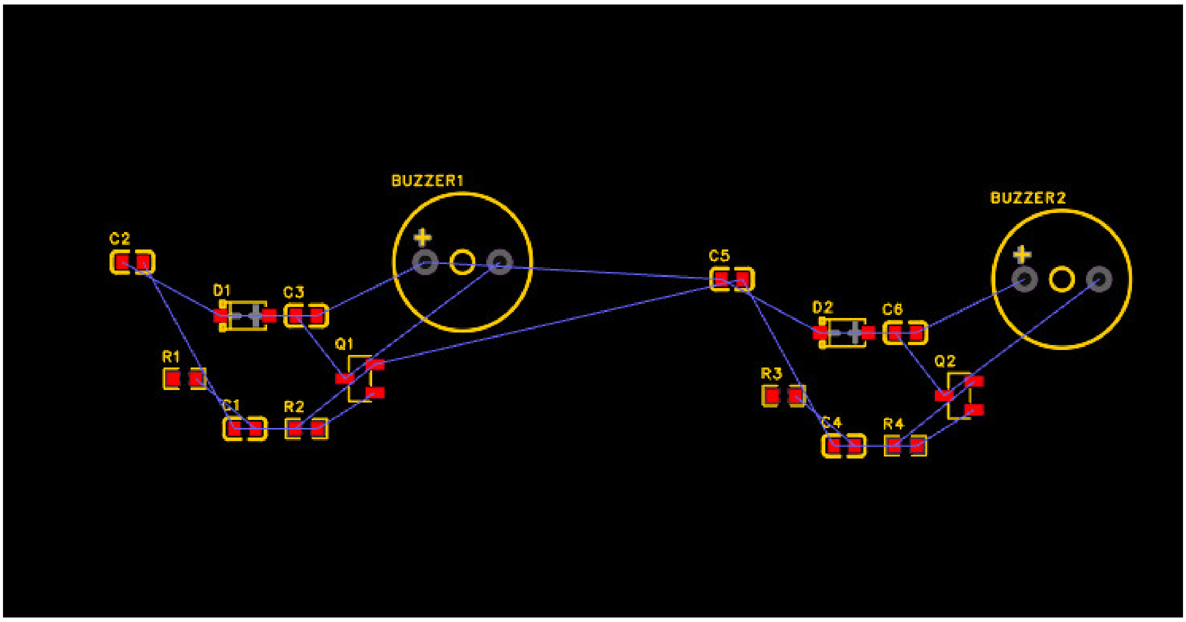
entrance： Top menu - Layout - Reuse layout routing.

Operation steps:

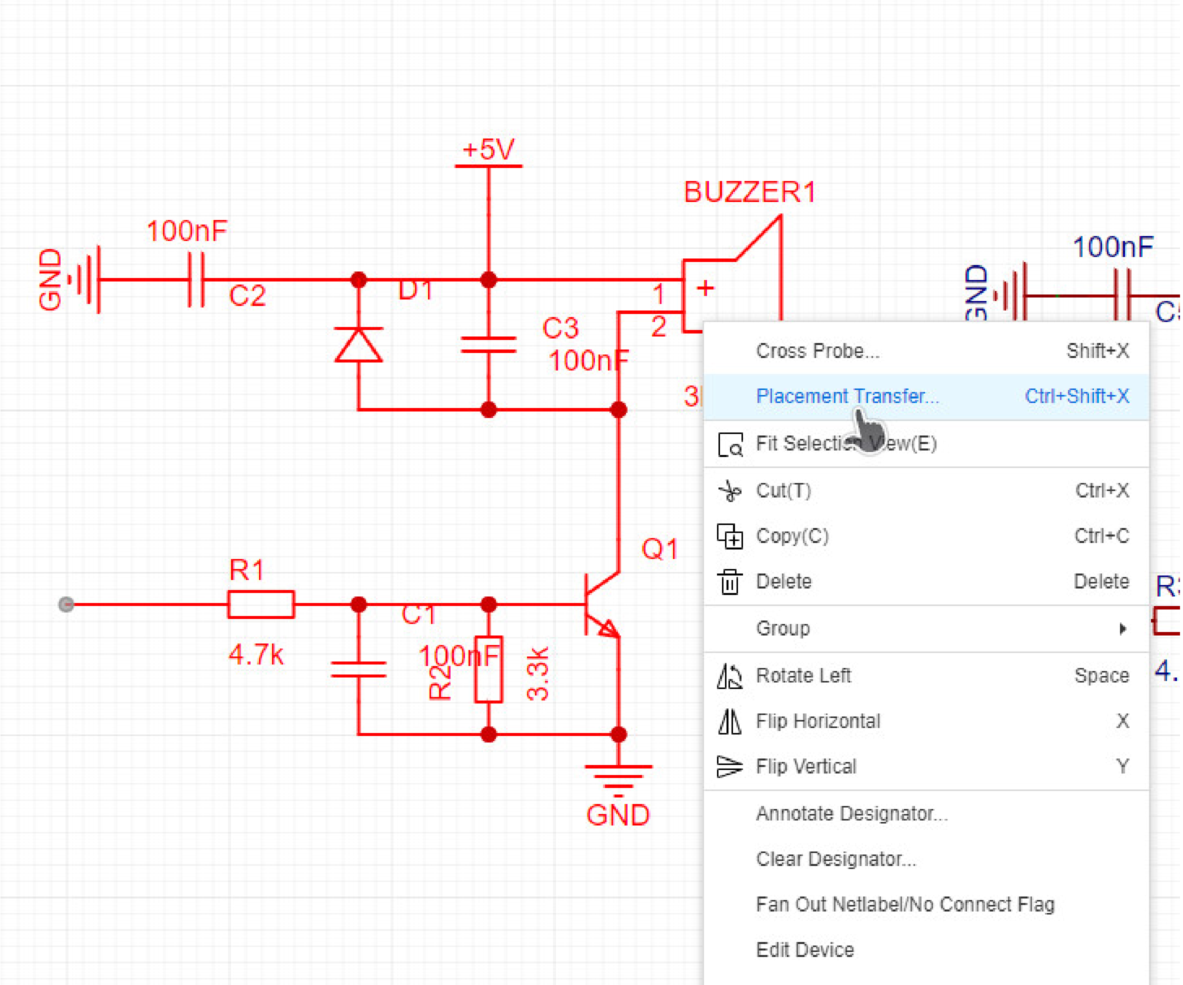
1. First draw a number of groups of circuits in the schematic diagram, such as multi-channel circuit modules.



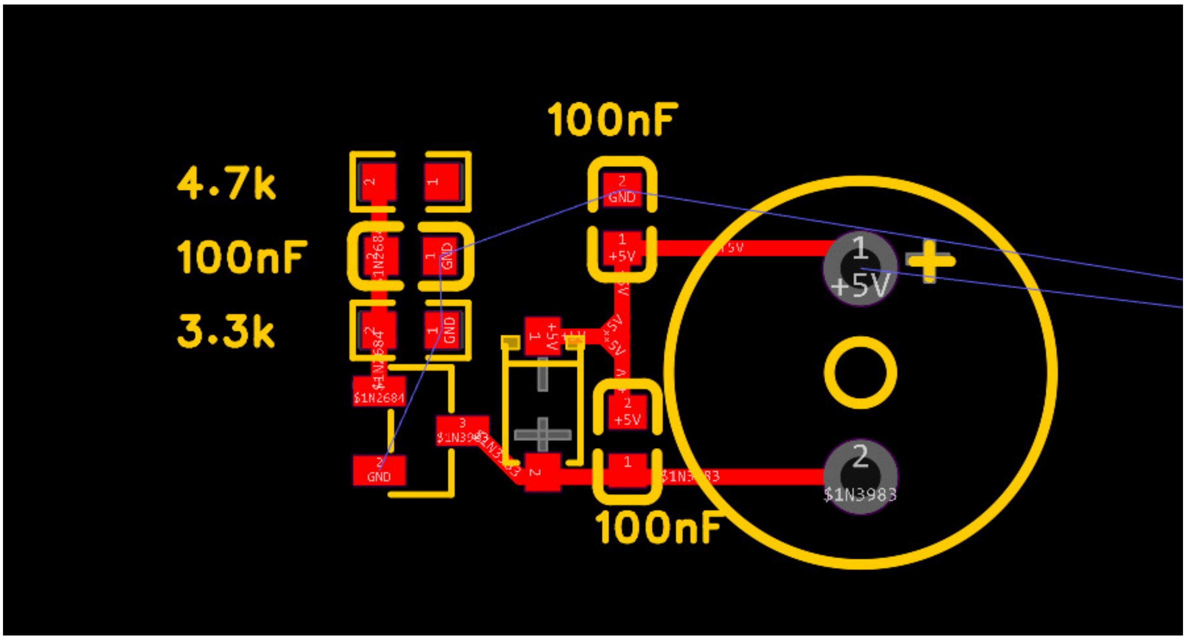
2. After completion, the schematic diagram is transferred to PCB. The default is to group together roughly.



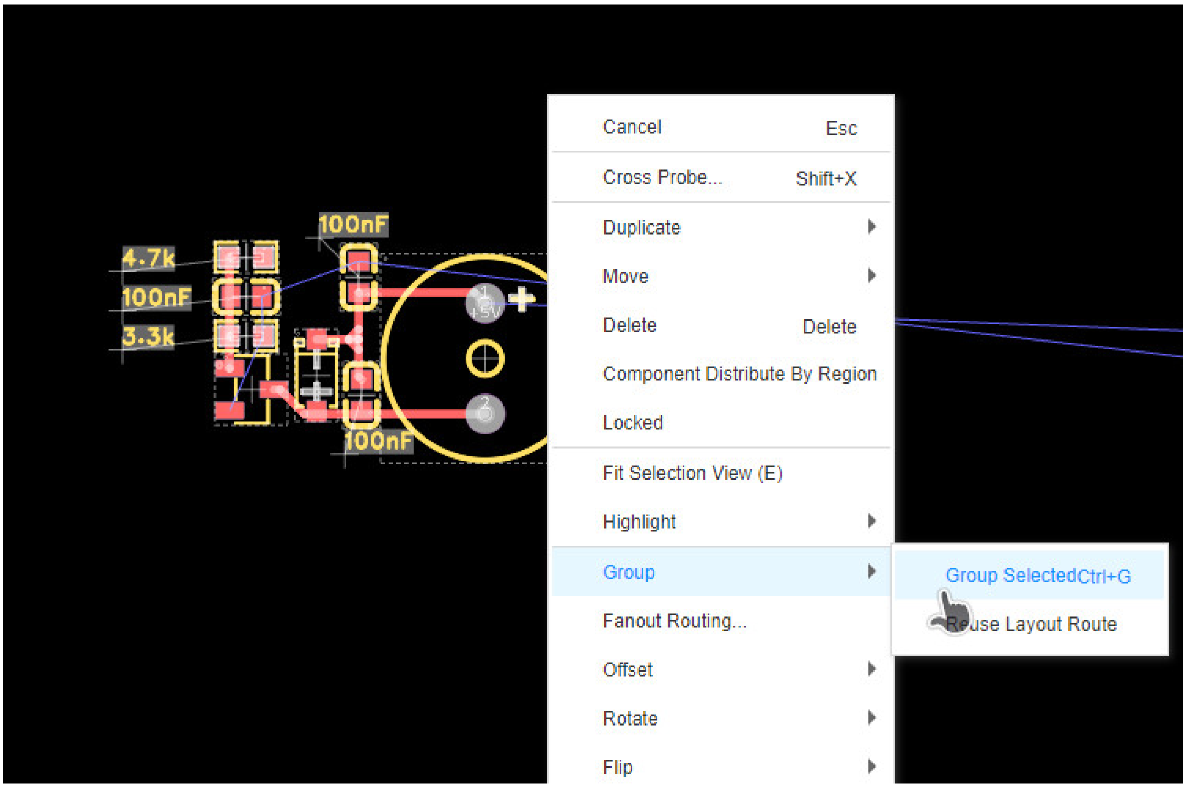
3. Go back to the schematic diagram, select a group of circuits in the box, and use the cross selection function (right-click menu, top design menu, or shortcut key SHIFT+X, etc.) to cross select the component corresponding to the PCB.



4. It will automatically switch to PCB and select the corresponding component. After manually dragging to the side, it will complete the operations such as layout, wiring and adding vias.

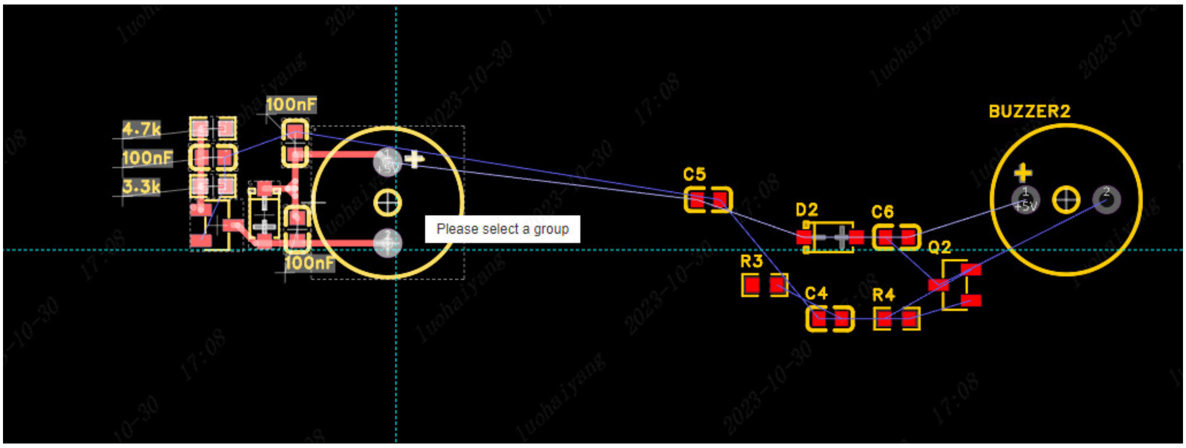


5. the box after the completion of the circuit, right-click the combination for a new combination.

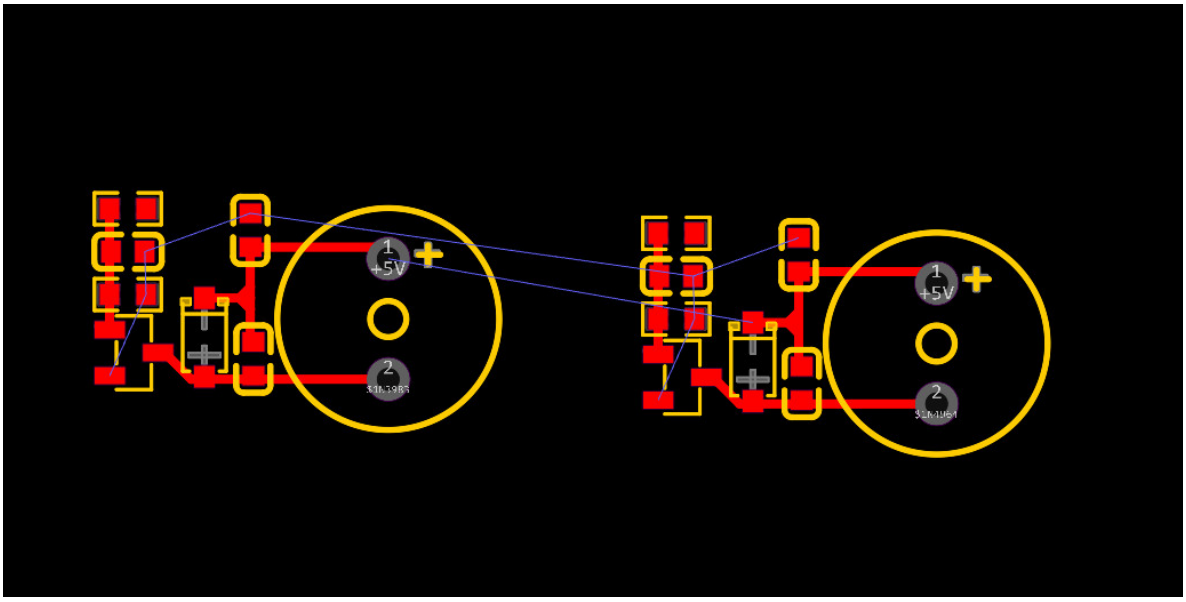


6. back to the schematic diagram, box select the second set of circuits, right click to use the cross check function. Select the component corresponding to PCB, and then right-click menu - Combination - Multiplexing layout routing

7. After the cross cursor appears, select the first combination and click. The components of the second group will be automatically combined into a new combination, and reuse the component position and wiring of the first group of circuits.



8. other groups of circuits repeat according to the above operation, you can quickly carry out the circuit layout and routing multiplexing, reduce the workload of layout and routing.



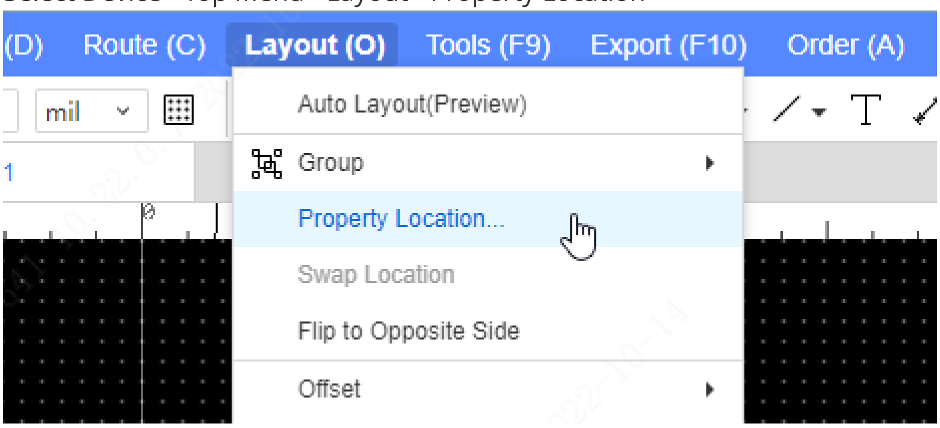
After completion, if no combination is needed, the combination can be dissolved manually.

Property Location

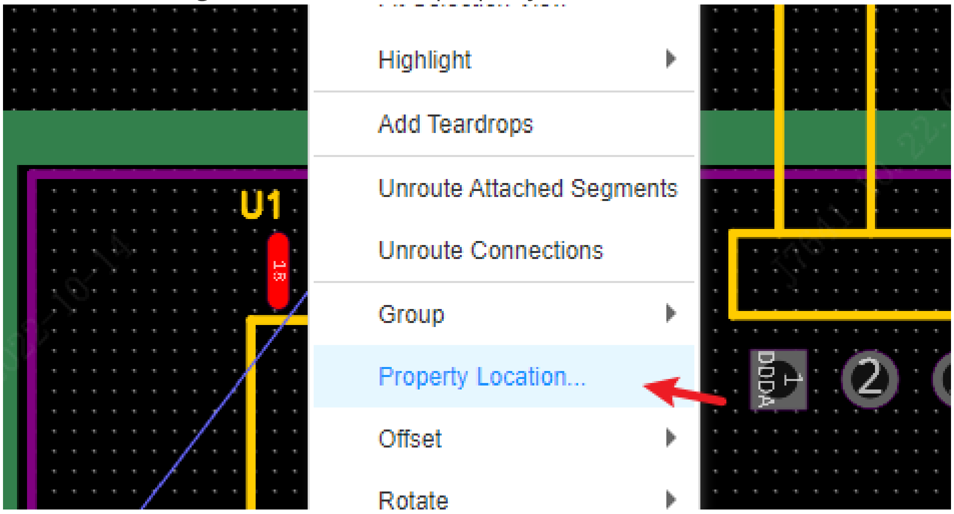
Modify the Place of the device name and tag number

Steps:

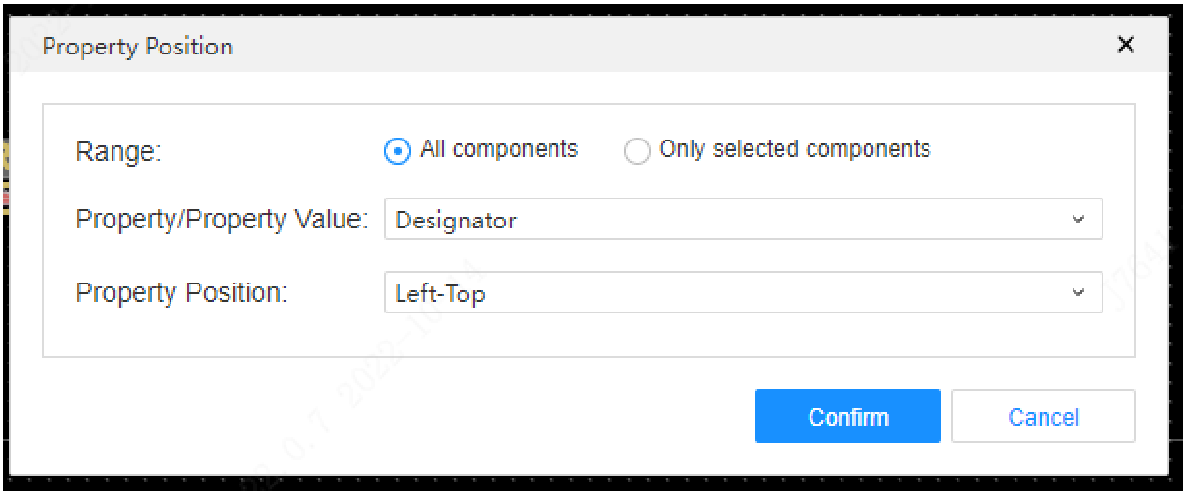
- Select Device - Top Menu - Layout - Property Location

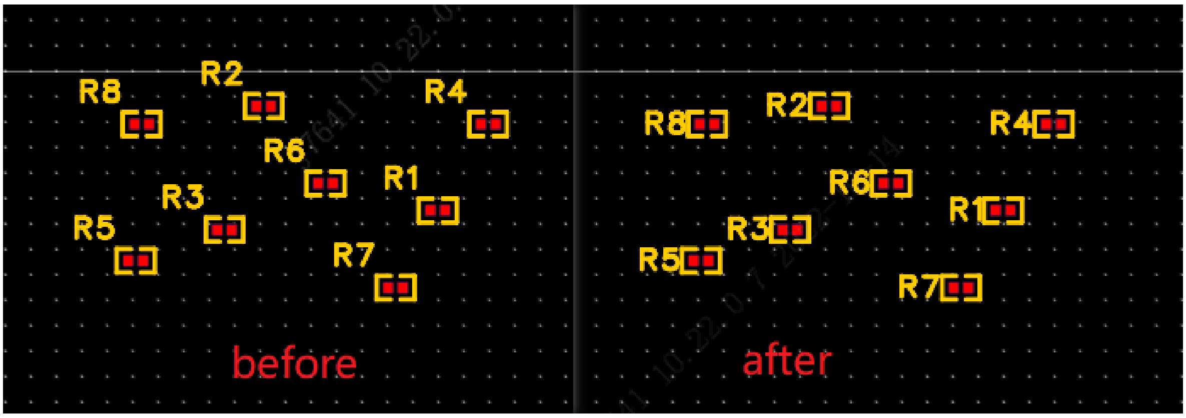


- select device - right mouse button - property location



Select the attribute position of the footprint text in the pop-up window, click OK, and the attribute tag number of the device can be placed in the selected position.



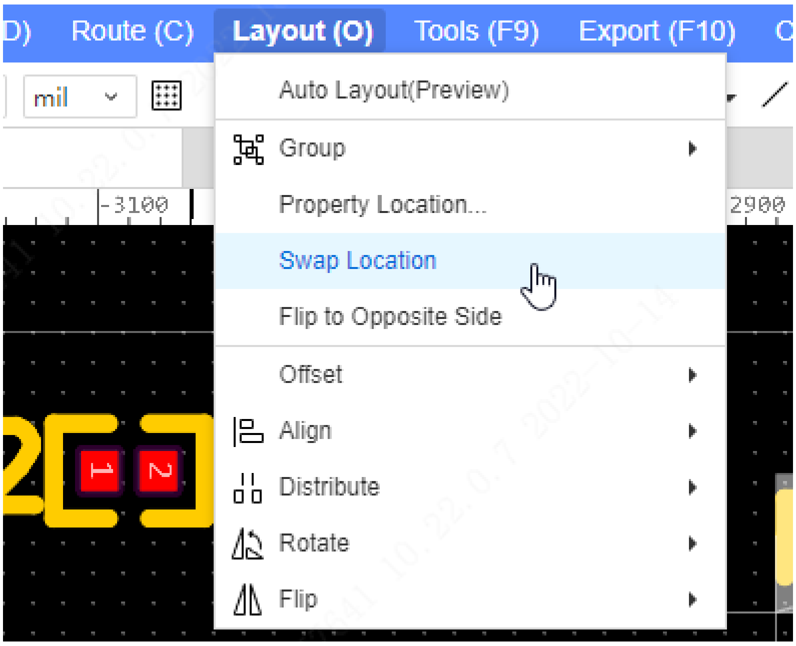


Swap Positions

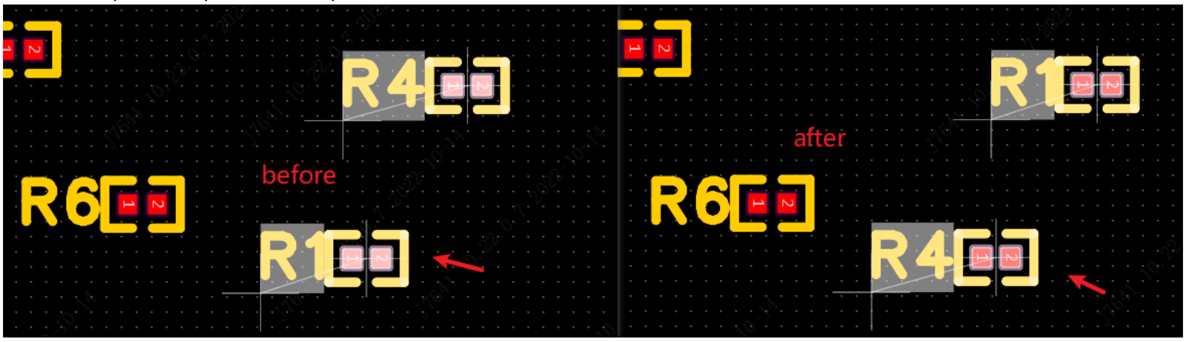
Swap the positions of the two devices, the wires do not follow the device swap

Steps:

- Select two devices to be swapped - Top Menu - Layout - Swap positions
- Select two devices to be swapped - right mouse button - swap positions



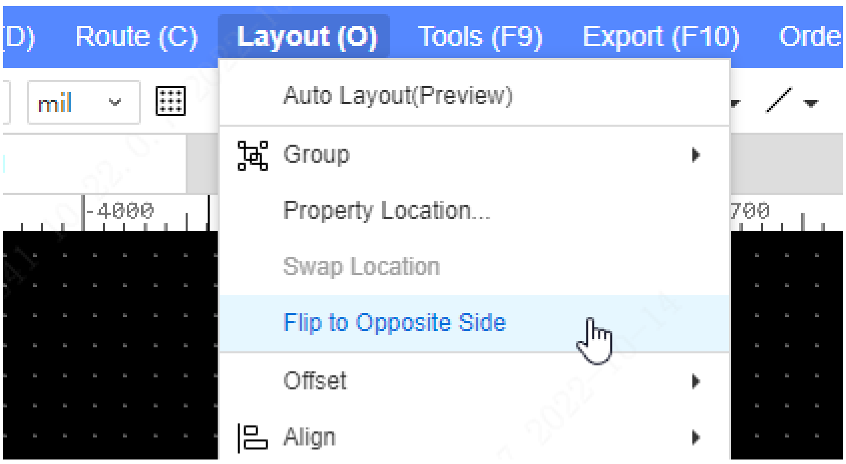
for example swap two components



Flip to Opposite Side

If you need to flip the primitive to the opposite layer, such as flipping the top surface to the bottom surface, you can use the flip function.

Entrance: Layout - Flip to Opposite Side.



After selecting the element of the canvas, click the flip menu, and the element that can be flipped will be automatically flipped to the opposite layer, and will be mirrored automatically. If the element of the top layer + inner layer 1 is flipped, it will be flipped to the inner layer. 2+ ground floor.

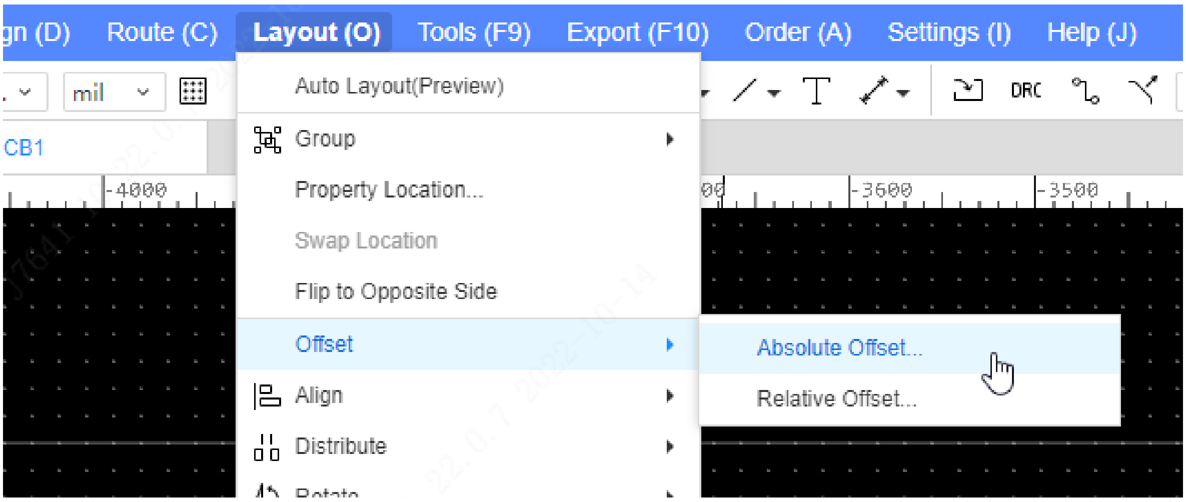
Offset

Offset the device on the X/Y axis.

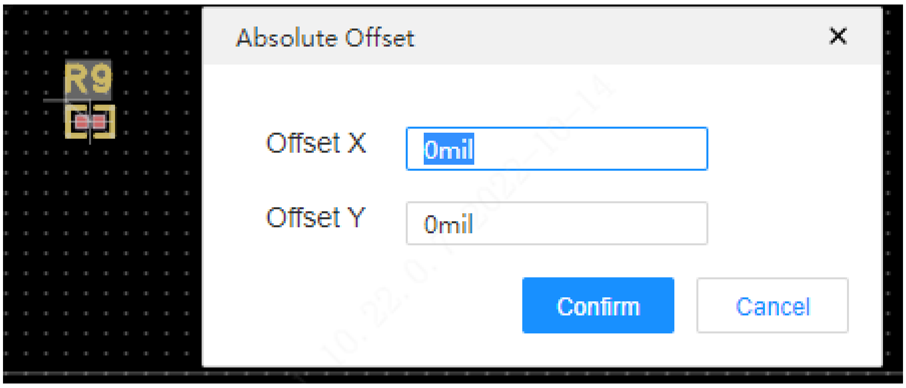
absolute offset

How to operate:

- Select the device to be offset - Top Menu - Layout - Offset - Absolute offset
- Select the device to be offset - Right mouse button - Offset - Absolute offset



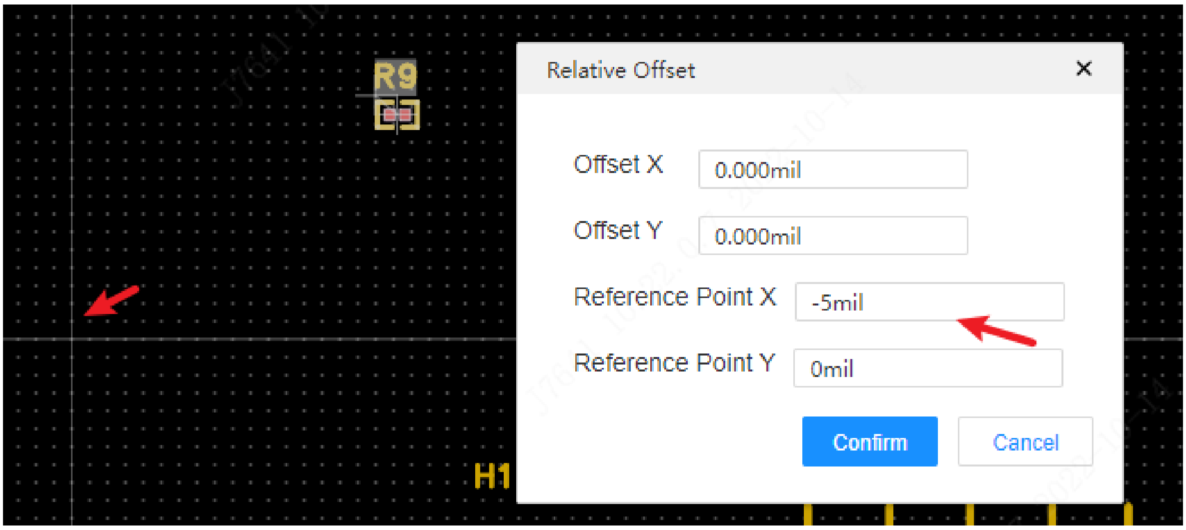
Absolute offset is to offset the coordinates of the device to be offset to zero.



Enter the coordinate value you want to offset, and click OK to offset the input coordinate position.

Relative offset

The difference between relative offset and absolute offset is that relative offset needs to select a reference point for offset operation.

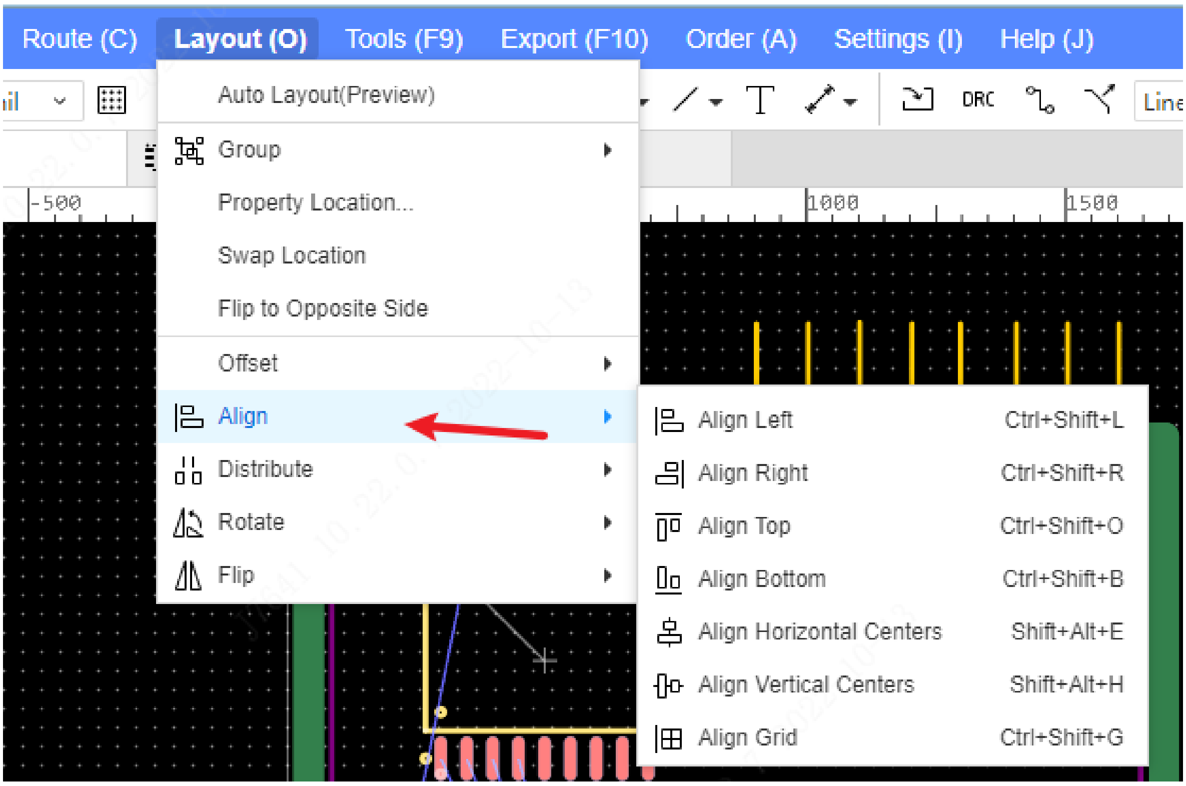


Align

The Align function is available in the schematic diagram, multiplexed block, symbol, footprint, and PCB, and the operation method is the same.

Align operation method:

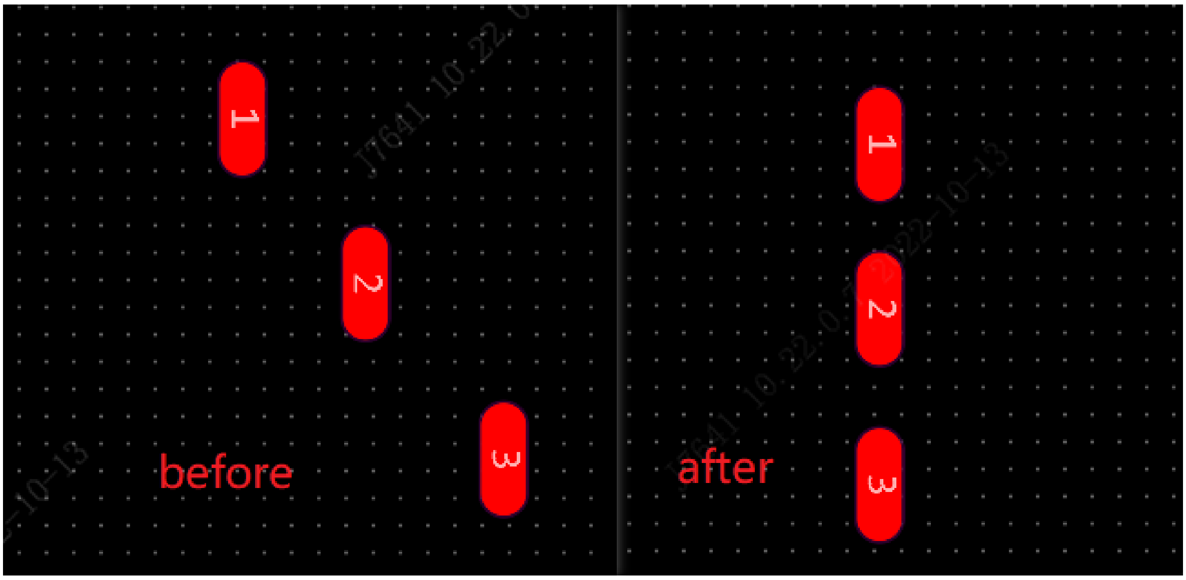
- Select the device to be aligned - Top Menu - Layout - Align
- Shortcut keys `Ctrl`+`Shift`+`↑`



Align Left

To select the devices that need to be aligned, you need to select more than two. There is no reference point for a single device, which cannot be aligned.

- Top Menu - Layout - Align - Align Left
- Shortcut `ctrl`+`shift`+`←`



Align Right

Select the device to be aligned

- Top Menu - Layout - Align - Align Right
- or shortcut keys ctrl + shift + → 2-1608540345551

Align Top

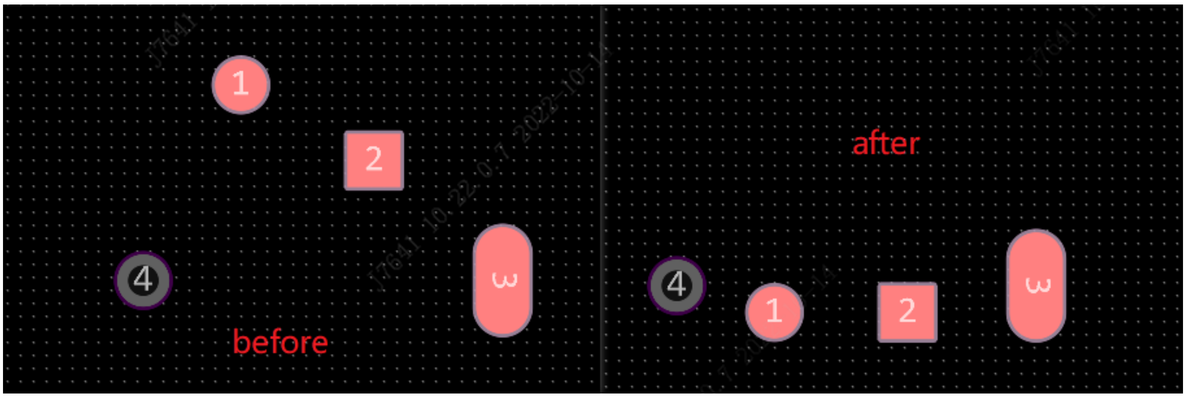
Select the device to be aligned.

- Top Menu - Layout - Align - Align Top
- Shortcut keys ctrl + shift + ↑

Align Bottom

Select the device to be aligned,

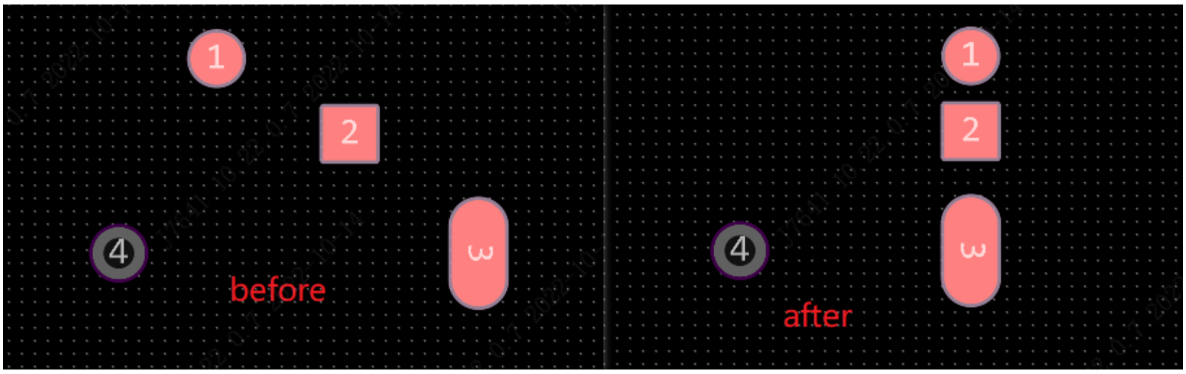
- Top Menu - Layout - Align - Align Bottom
- Shortcut keys ctrl + shift + ↓



Align Horizontal center

Select the device to be aligned

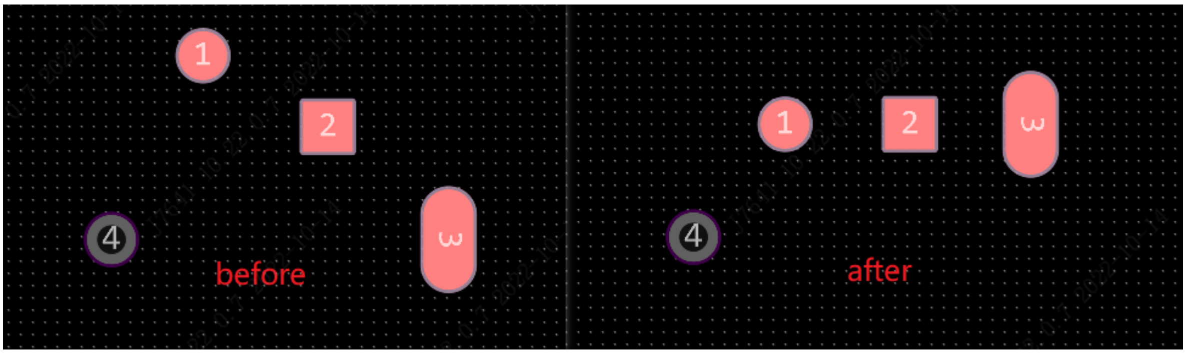
- Top Menu - Layout - Align - Align Horizontal center
- Shortcut keys `ctrl`+`shift`+`M`



Align Vertical Center

Select the device to be aligned,

- Top Menu - Layout - Align - Align Vertical Center
- Shortcut keys `ctrl`+`shift`+`C`



Align Grid

Select the device to be aligned, and align the selected device to the grid of the PCB editor.

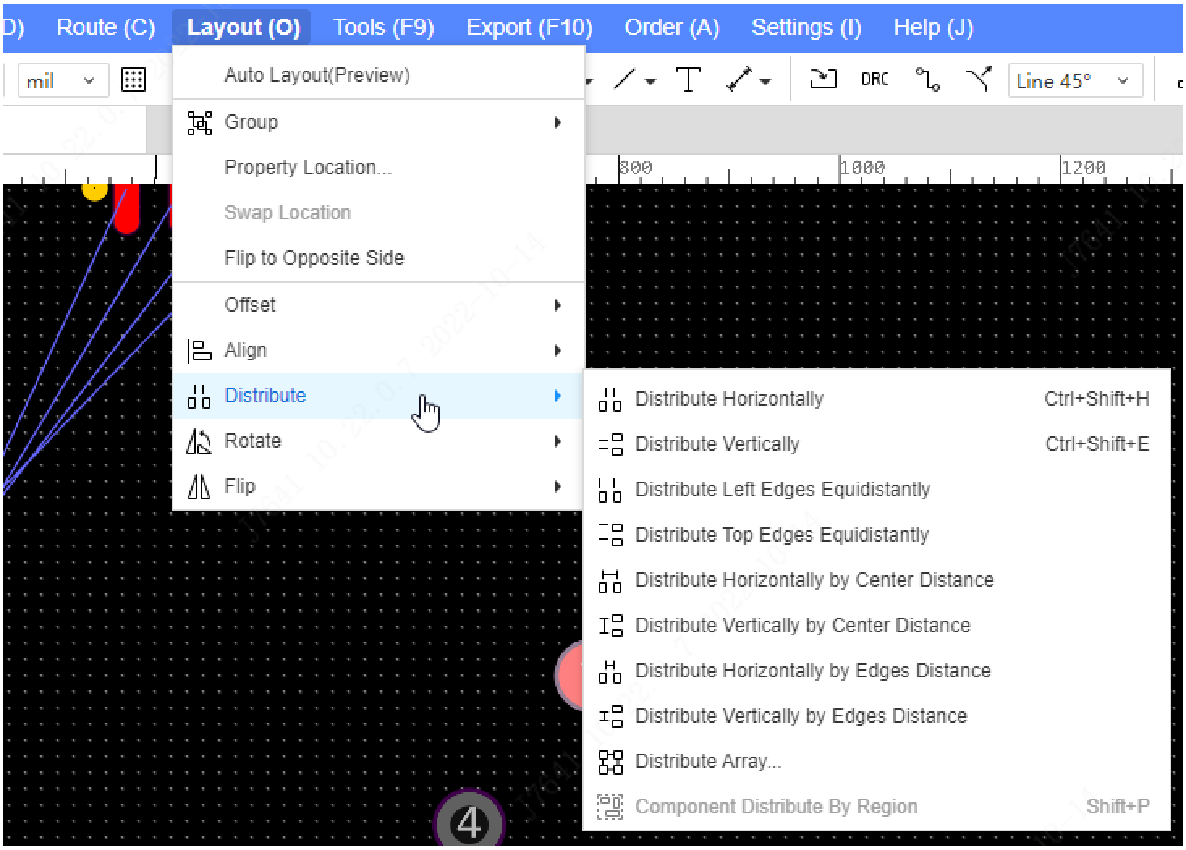
- Top Menu - Layout - Align - Align Grid
- Shortcut keys `ctrl`+`shift`+`G`

Distribution

EasyEDA Pro supports distributed row alignment.

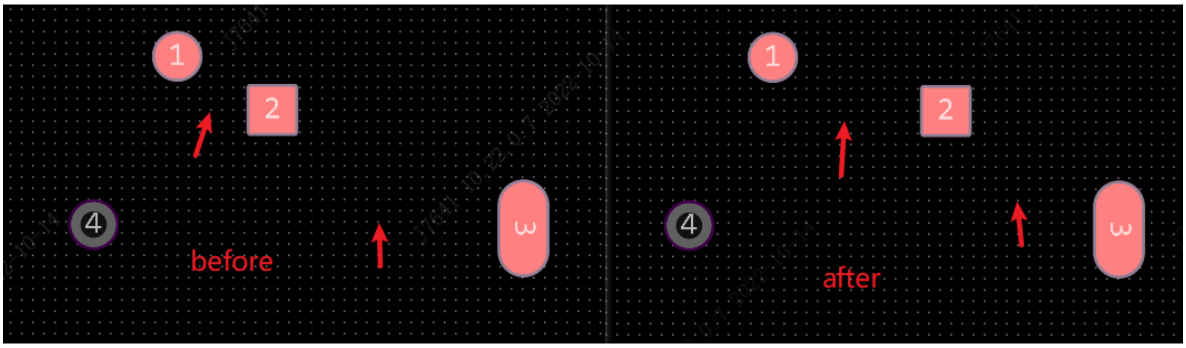
How to operate:

- Top Menu - Layout - Distribute or Shortcut Bar at Top



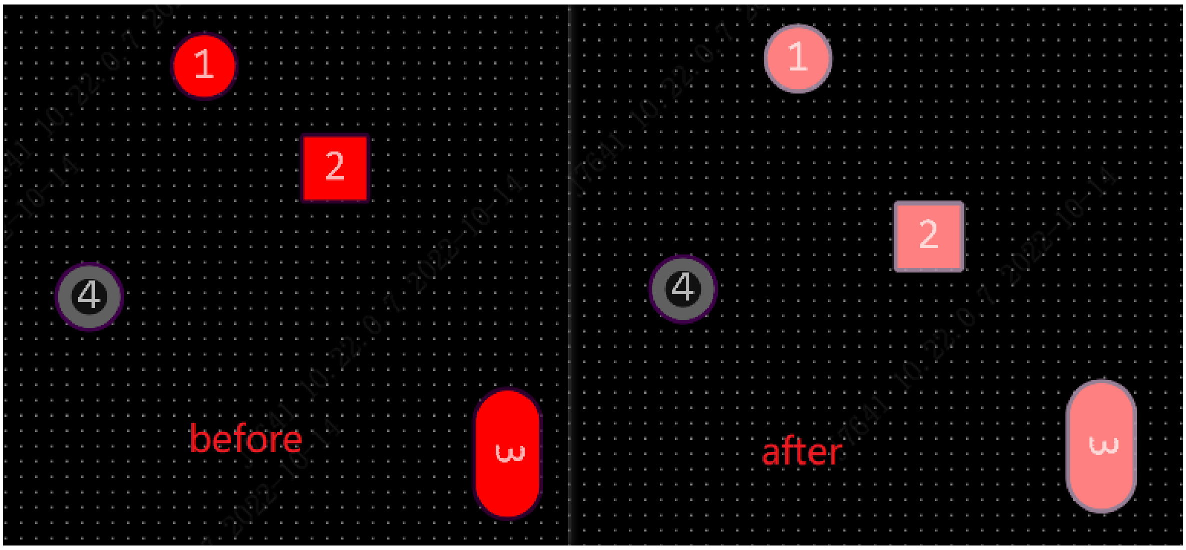
Distribute Horizontally

Select the devices that need to be distributed, click the Top Menu - Layout - Distribution - Horizontal Equidistant Distribution, and EDA will distribute the devices or other elements on the horizontal plane equally.



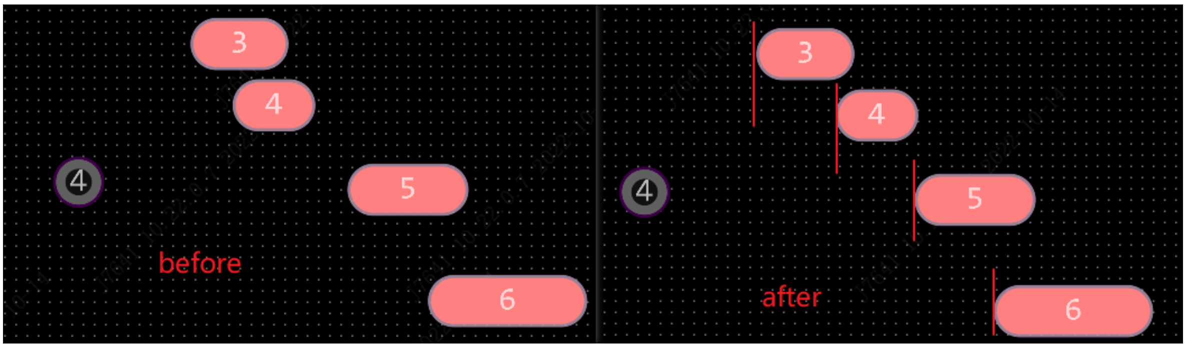
Distribute Vertically

Select the devices that need to be distributed, click the Top Menu - Layout - Distribution - Vertical equidistant distribution, EDA will distribute the vertical devices or other elements vertically and equidistantly.

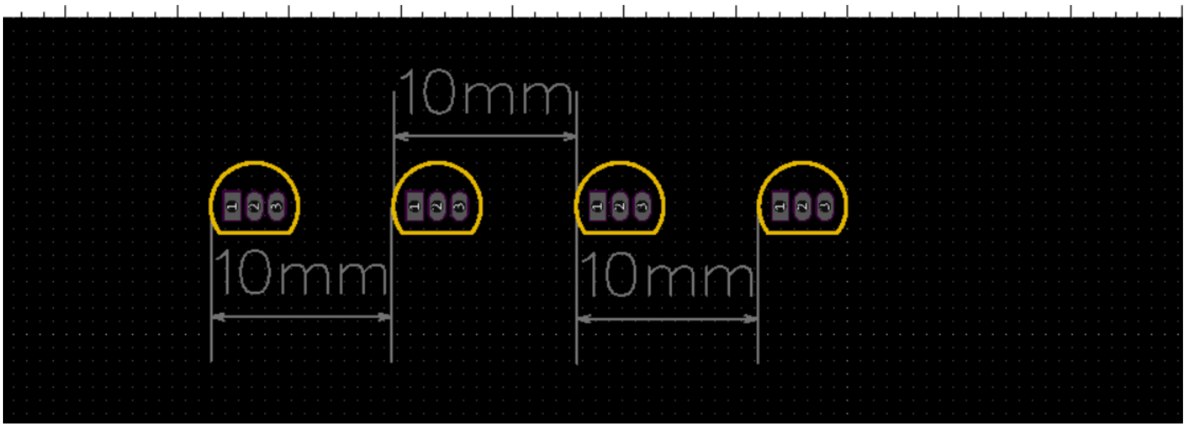


Distribute Left Edges Equidistantly

The silk screen margin of the selected device on the left is the reference point distribution row distance.

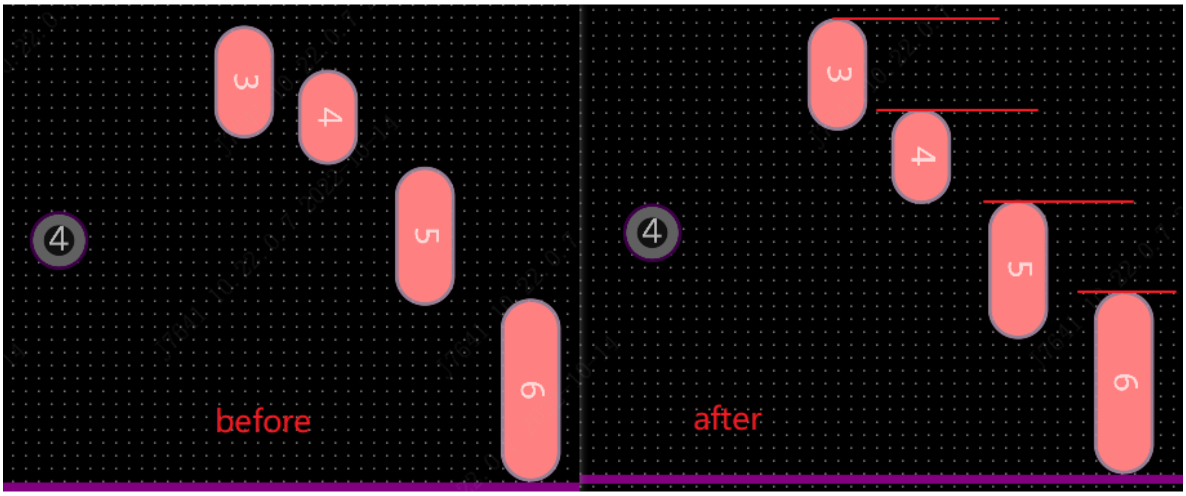


It can be clearly seen that the screen printing spacing on the left side of the first device is the same as the screen printing spacing on the left side of the other device.



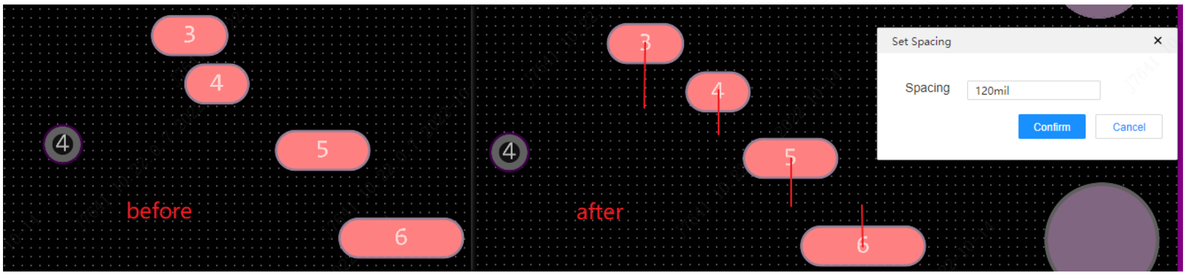
Distribute Top Edges Equidistantly

The screen printing margin of the selected upper device is the reference point distribution row distance.



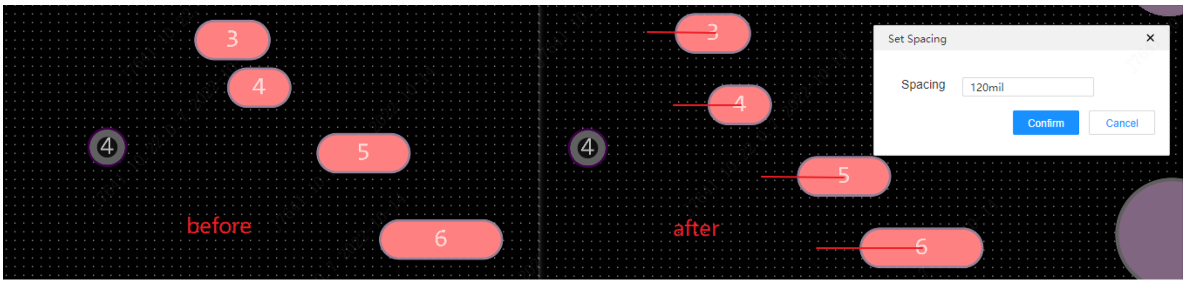
Distribute Horizontally by Center Distance

The selected devices are arranged according to the horizontal spacing of the data entered by the user.



Distribute Vertically by Center Distance

The selected devices are arranged with vertical spacing according to the spacing data entered by the user.



Distribute Horizontally by Edges Distance

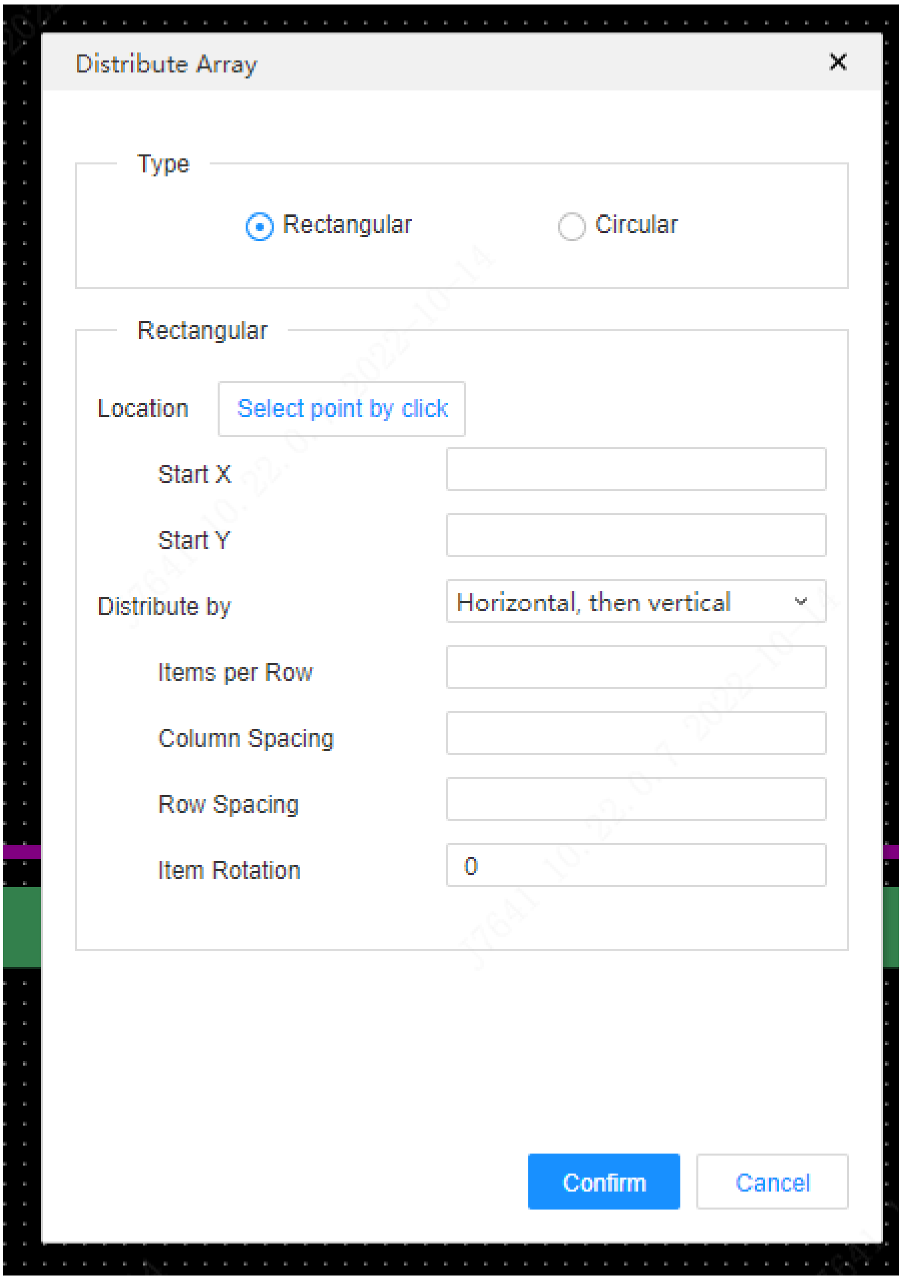
it likes Distribute Horizontally by Center Distance, just by object edge

Distribute Vertically by Edges Distance

it likes Distribute Vertically by Center Distance, just by object edge

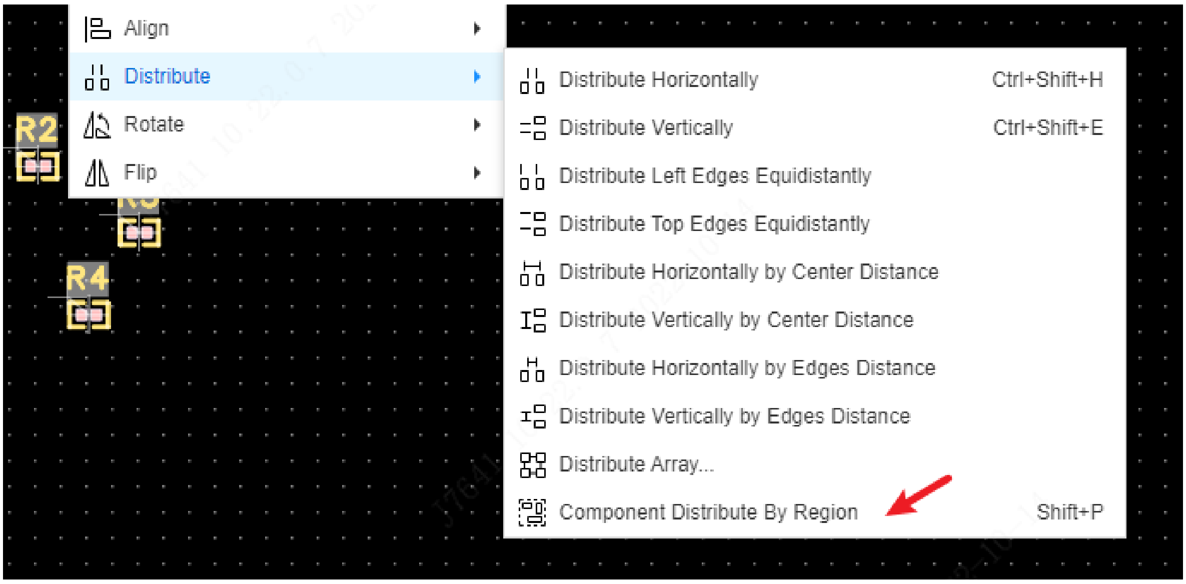
Distribute Array

The function of array distribution is the same as that of array object, so I won't introduce it here. The detailed operation steps have been explained in the array object tutorial.

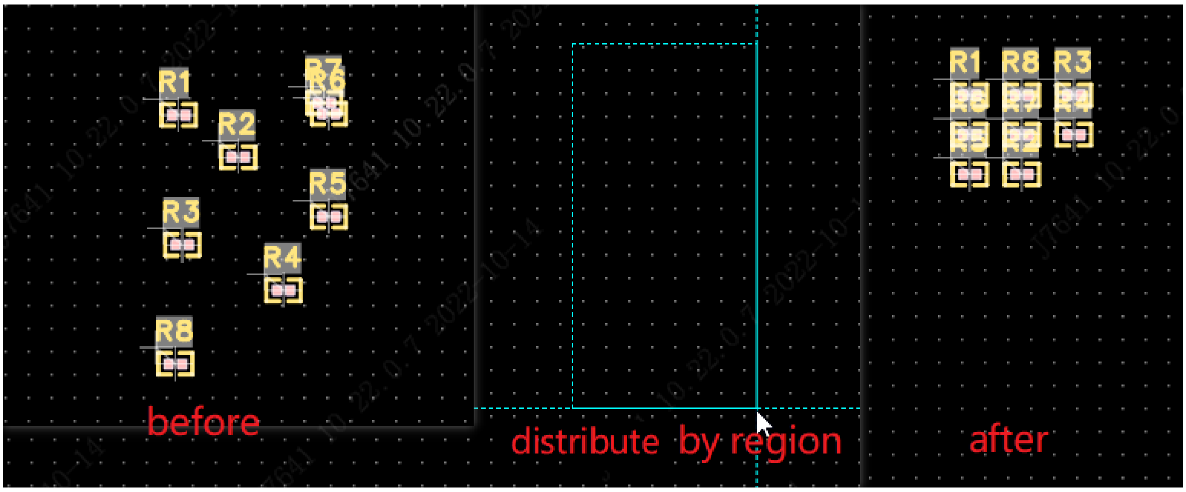


Component Distribute By Region

when you select multiple components, you can use this feature to aligan to a region you frame region select, it is very quickly to re-arrange the component location.



select components

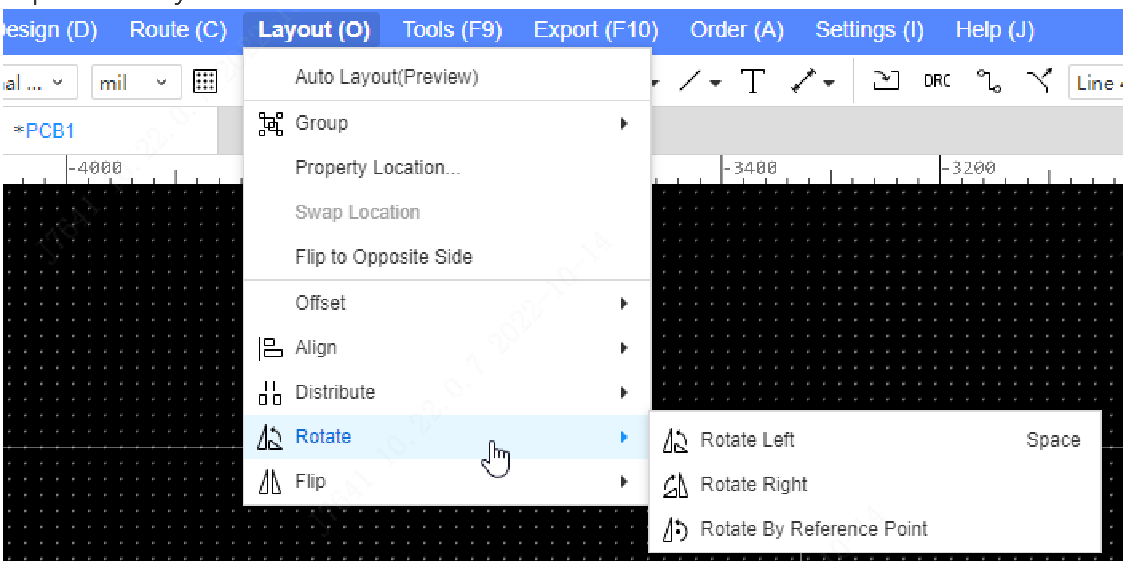


Rotate

Rotate the device

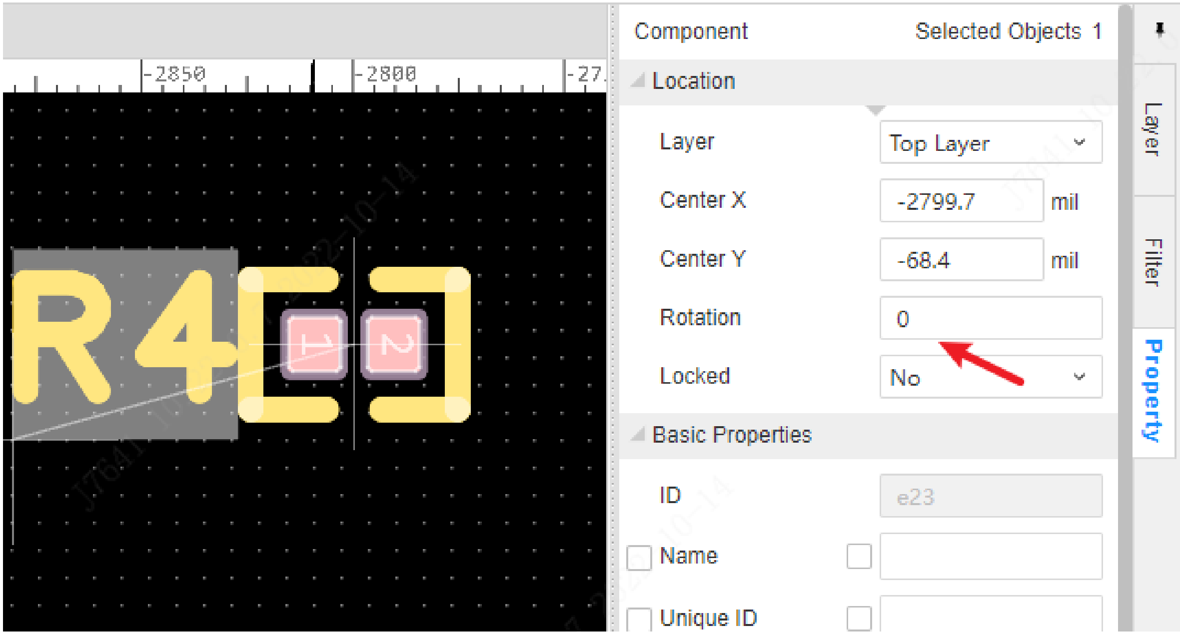
Operation method:

- Top Menu - Layout - Rotation

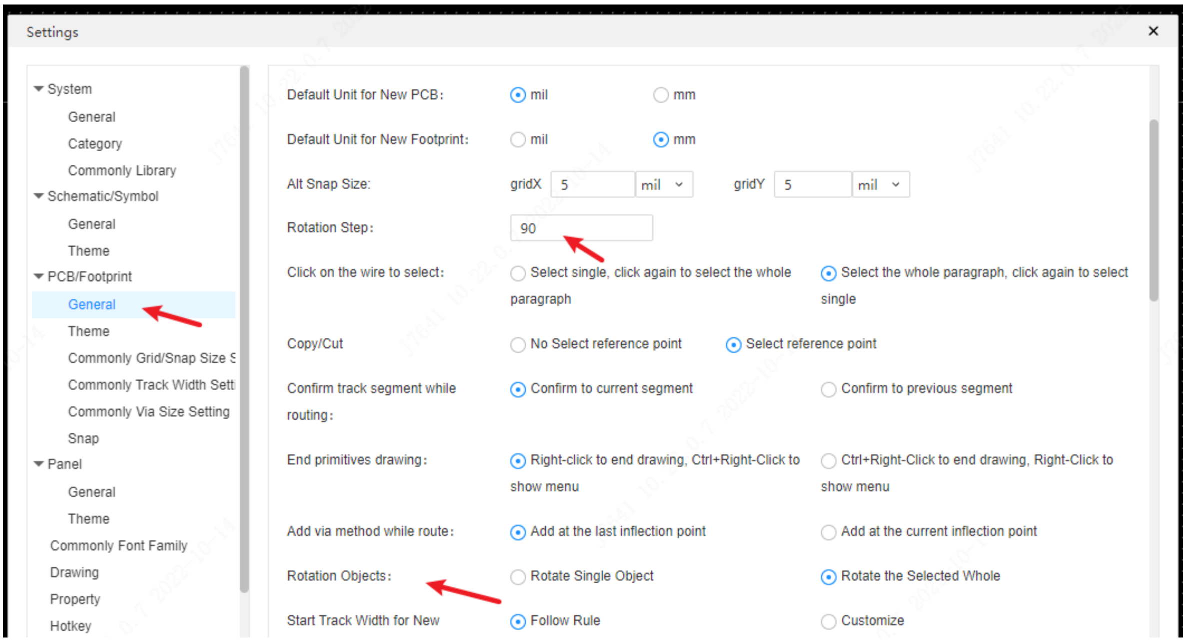


- **Rotate Left**: Rotate the selected device to the left.
- **Rotate Right**: Rotate the selected device to the right.
- **Reference point rotation**: The reference point rotation is to select a point as the reference point, and the rotation will be based on the selected reference point, and will not leave the area of the selected reference point.

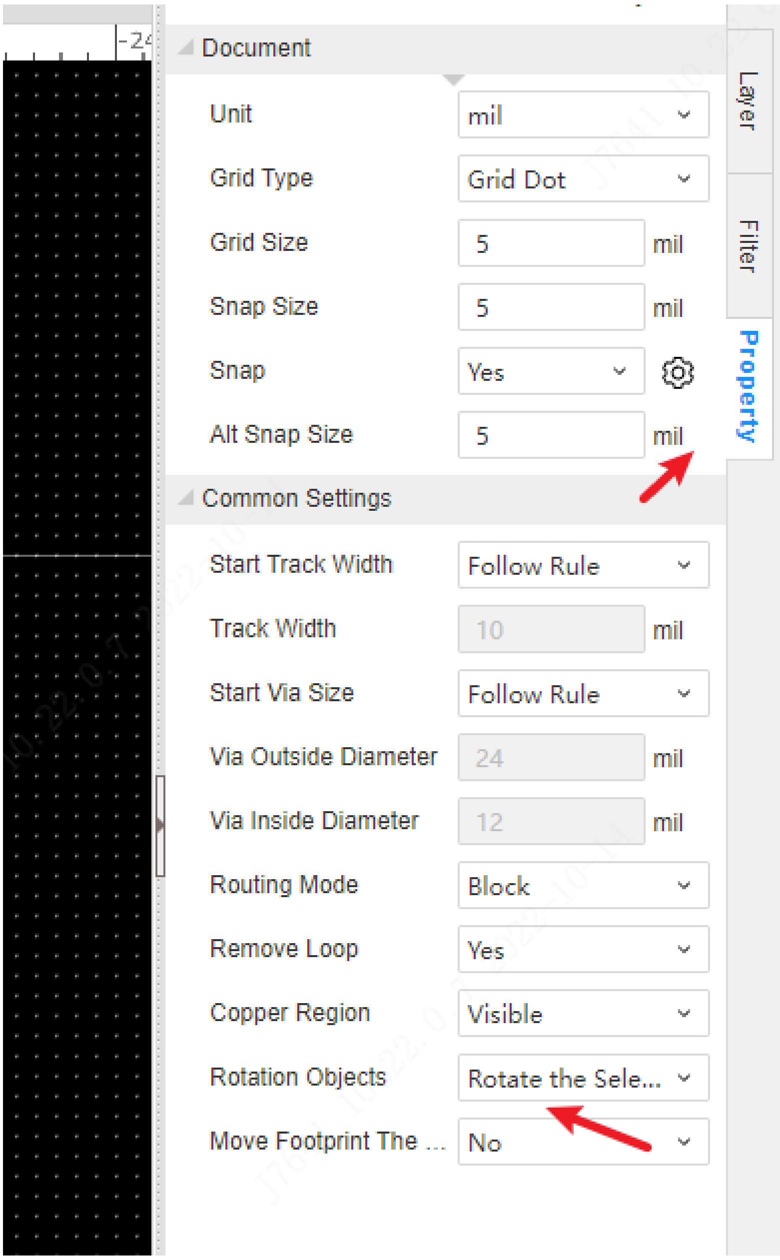
Or you can select a object, and change its rotation at property panel, or press hotkey `Space` to rotate.



Every time you rotate, the device is rotated by 90 degrees by default. If you want to modify the shortcut key to rotate the device angle, you need to change it in the system settings.



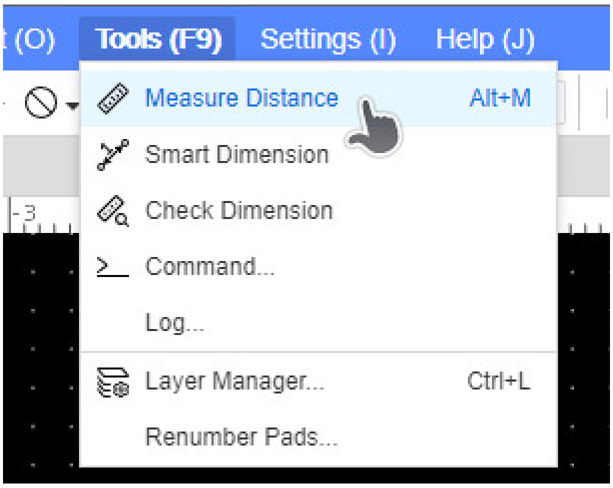
Rotation Objects support to rotate each one or rotate the whole selected objects at the same time, you can change the setting at right propety panel



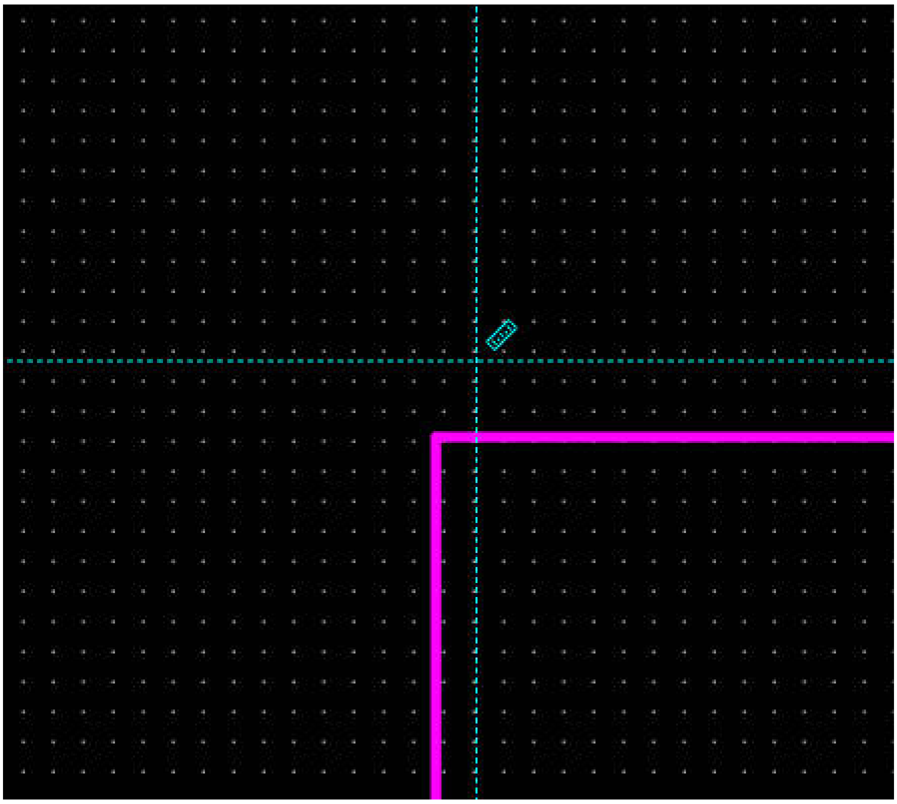
Measure Distance

Steps:

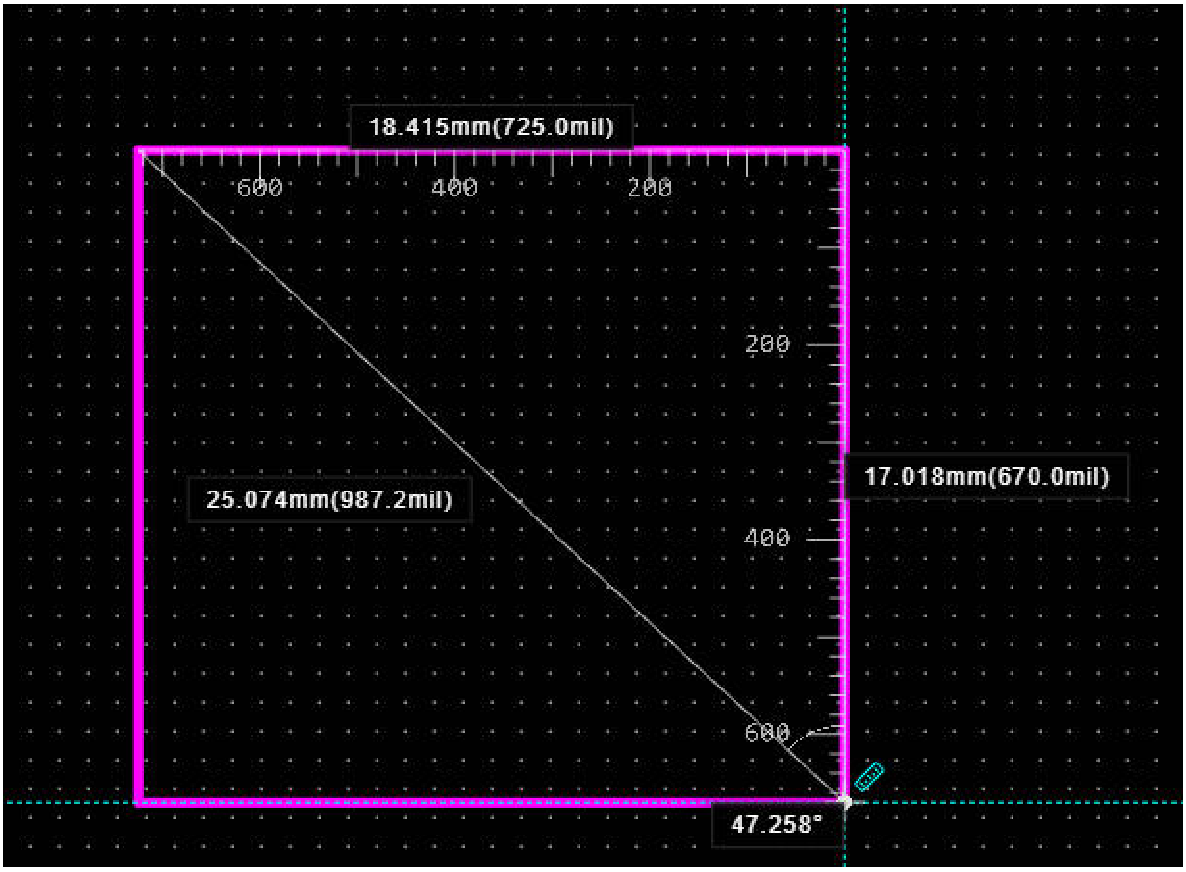
- Top Menu - Tools - Measure Distance
- Shortcut `Ctrl` + `M`
- Measure distance icon in the top toolbar (if not, check whether Measure distance is checked in Settings - Top toolbar -PCB)



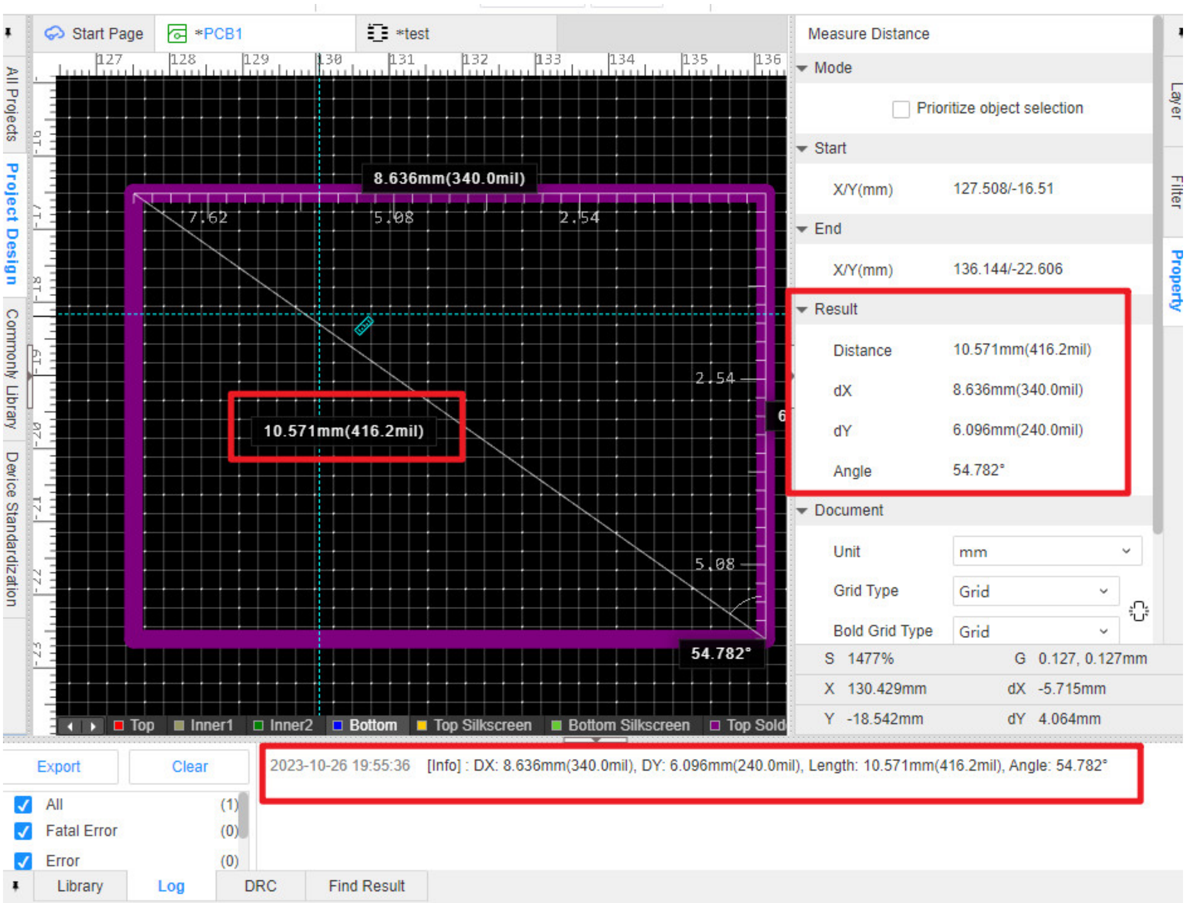
After clicking, the mouse becomes the cross cursor, and the starting point of measurement can be set by clicking.



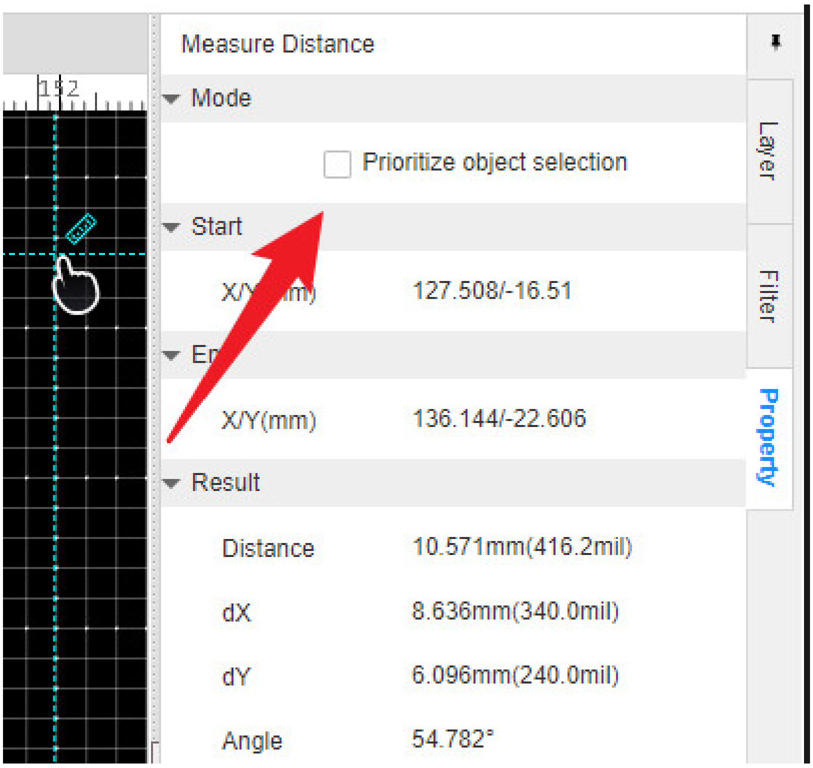
To set the end point of the measurement, move the mouse to another point you want to measure and click again



The measurement information will be displayed on the canvas, in the right properties panel, and in the log. These include straight-line distances, distances on the X-axis, distances on the Y-axis, and angles. Measurements on the canvas and in the right panel will be cleared automatically at the start of the next measurement or when you exit the measurement, while measurements in the log will remain unless you manually clear them



At any time during the measurement process, you can select the preferred object in the right panel. If you use this feature often, you can add a shortcut to it in System Settings - Shortcuts to quickly switch between them



If this is checked, you will be able to select one of the primitives in the process of selecting the starting point or the ending point. And after the measurement is completed, you can also switch different layers or positions in the right attribute, and the corresponding data will be refreshed immediately when switching

Old version of adsorption operation demonstration video:https://www.bilibili.com/video/BV1ca411G7an/?spm_id_from=333.999.0.0

Command

EasyEDA Pro can use system commands to draw quickly. Currently, only PCB and footprint commands are supported, and schematic diagrams and symbols are temporarily not supported.

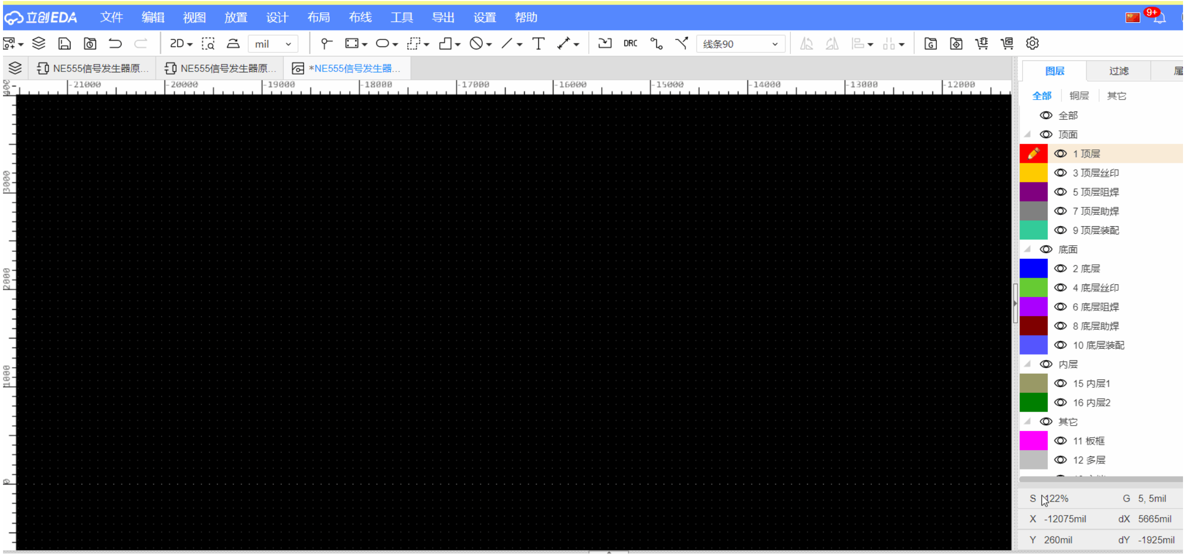
Instructions:

- Click on the top menu - Tools - Commands in the PCB interface or footprint interface
- Shortcut `C`

After clicking, a command input box will pop up in the lower left corner of the interface.

X-direction relative coordinate command IX

Click on the wire, shortcut `C` to call up the command input box, and enter IX 300 in the named input box (note that ix must be separated by a space to enter coordinates)



Y direction relative coordinate command iy

The operation mode of the relative coordinate command in the Y direction is the same as that of the relative coordinate command in the X direction.

Change layer command

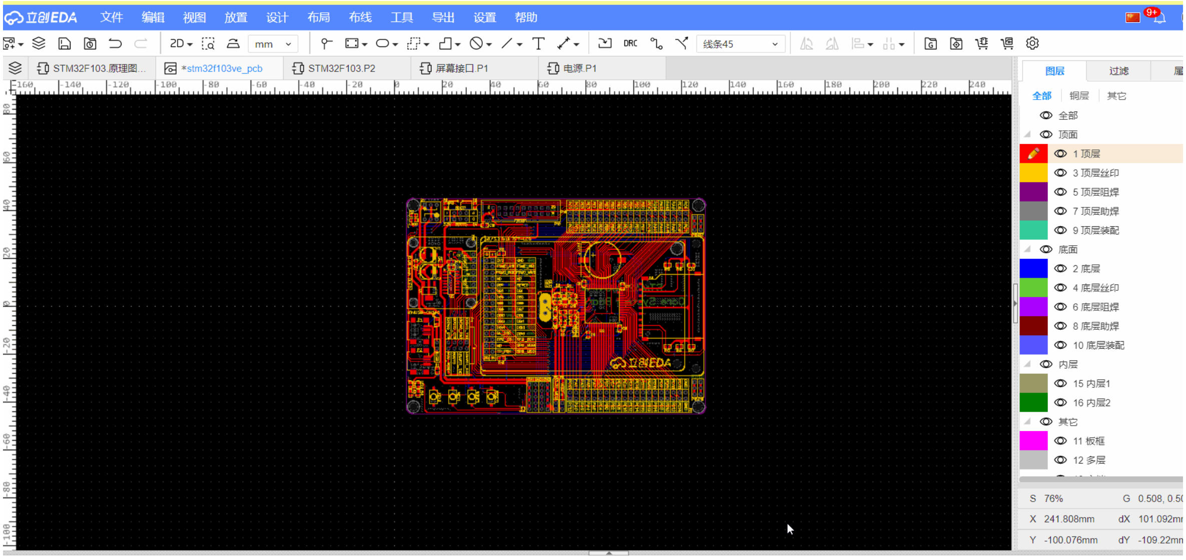
The shortcut key `C` brings up the command input box, enter "L 2" in the input box to switch to the bottom layer.

- Enter "L 1" to switch to the top level;
- Enter "L 2" to switch to the top level;
- Enter "L 3" to switch to the top silk screen layer;

The number of the corresponding layer can be seen here in the property panel layer on the right.

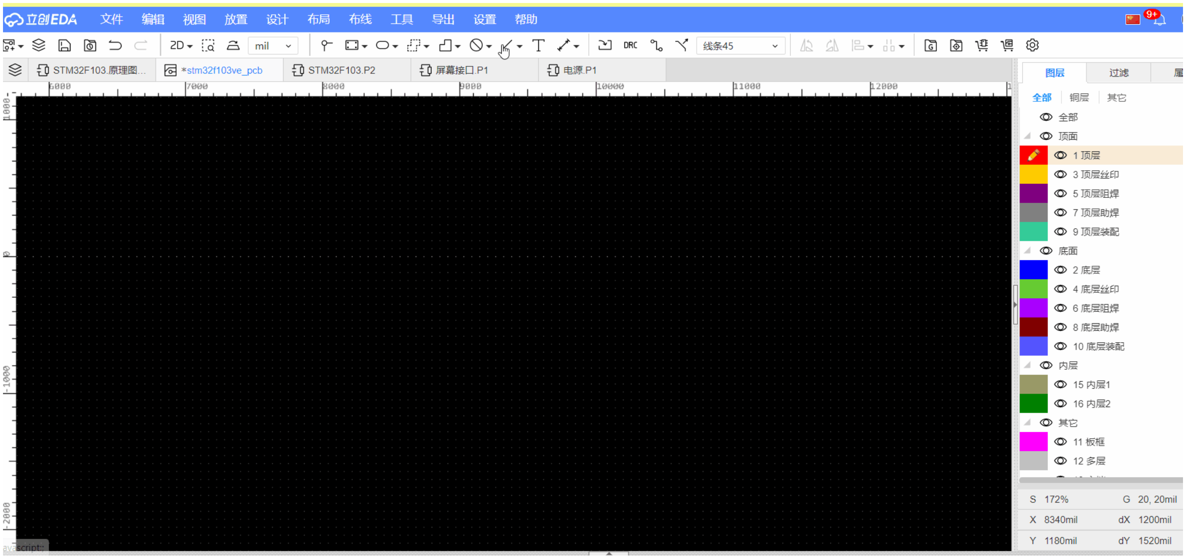
find component command

The shortcut key `C` brings up the command input box, input "S plus the bit number of the device" to find the device in the editor and highlight it.



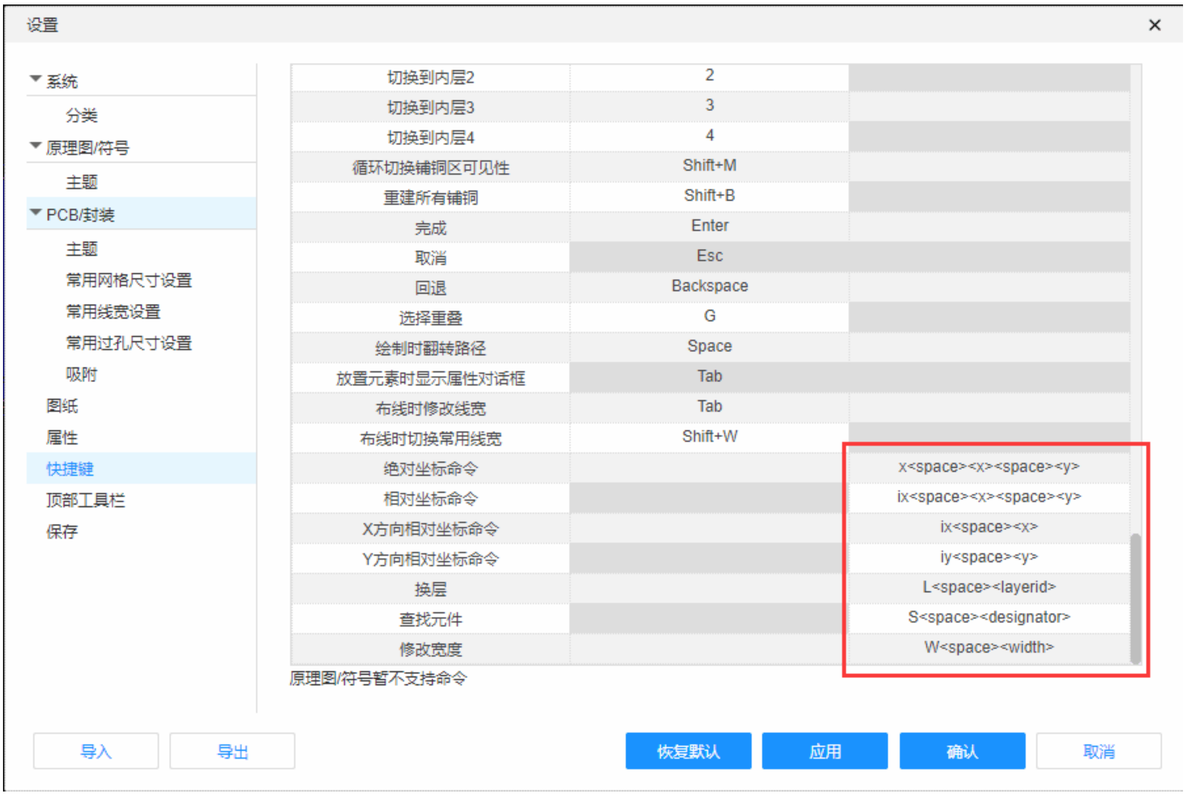
Modify line width

Click on the wire, and press the shortcut key `C` to call up the command input box, and input W to change the line width.



The search of the command is in the settings, the shortcut key setting query, or you can double-click to modify the command.

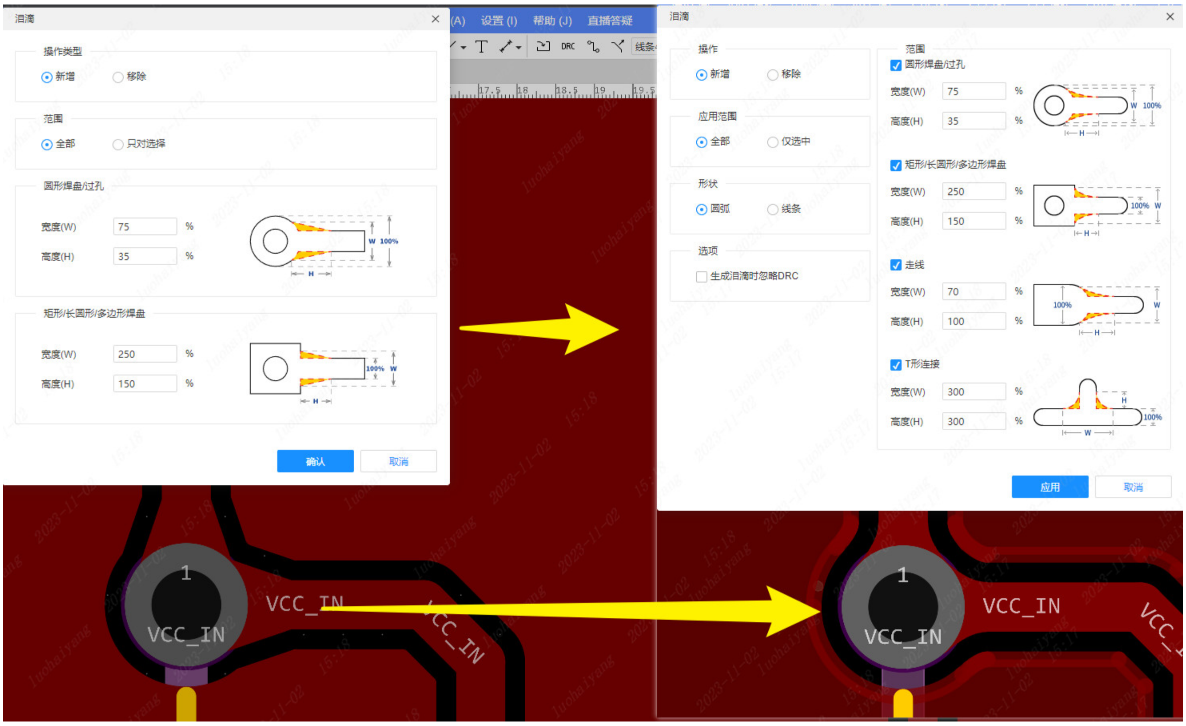
There are not many commands currently supported, and more commands will be introduced in the future.



Teardrop

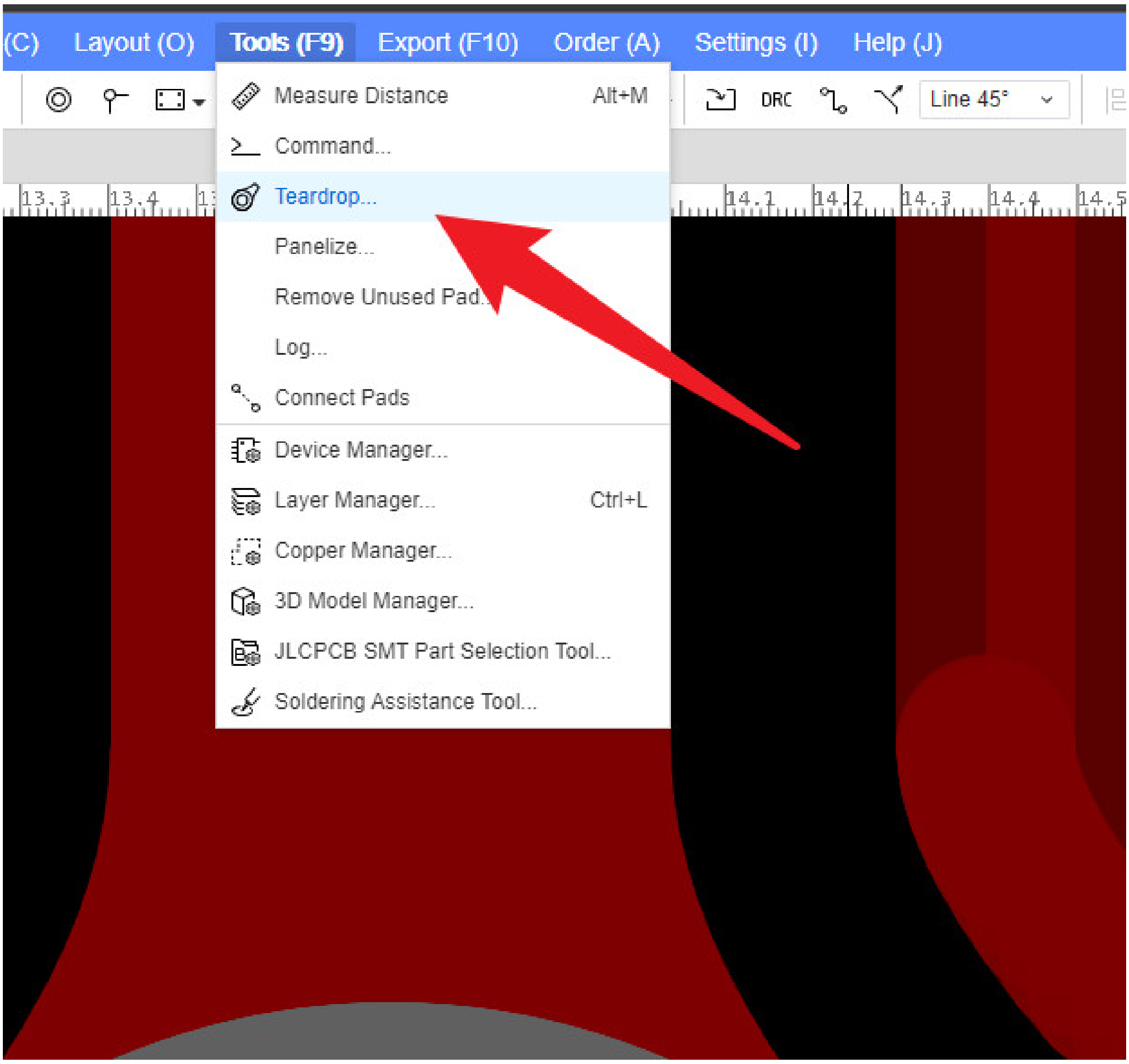
When drawing the PCB circuit board, we can add teardrops to avoid the circuit board being hit by a huge external force when the wire is disconnected from the pad or vias. In addition, the addition of teardrops can also make the PCB more beautiful, as well as to protect the pad from falling off during multiple soldering. In the production process, the use of tear drops can also avoid uneven etching caused by cracks and through hole misalignment. In addition, teardrops can also smooth the impedance in signal transmission, reduce the sharp jump of impedance, and avoid the reflection problem caused by the sudden small line width in high-frequency signal transmission.

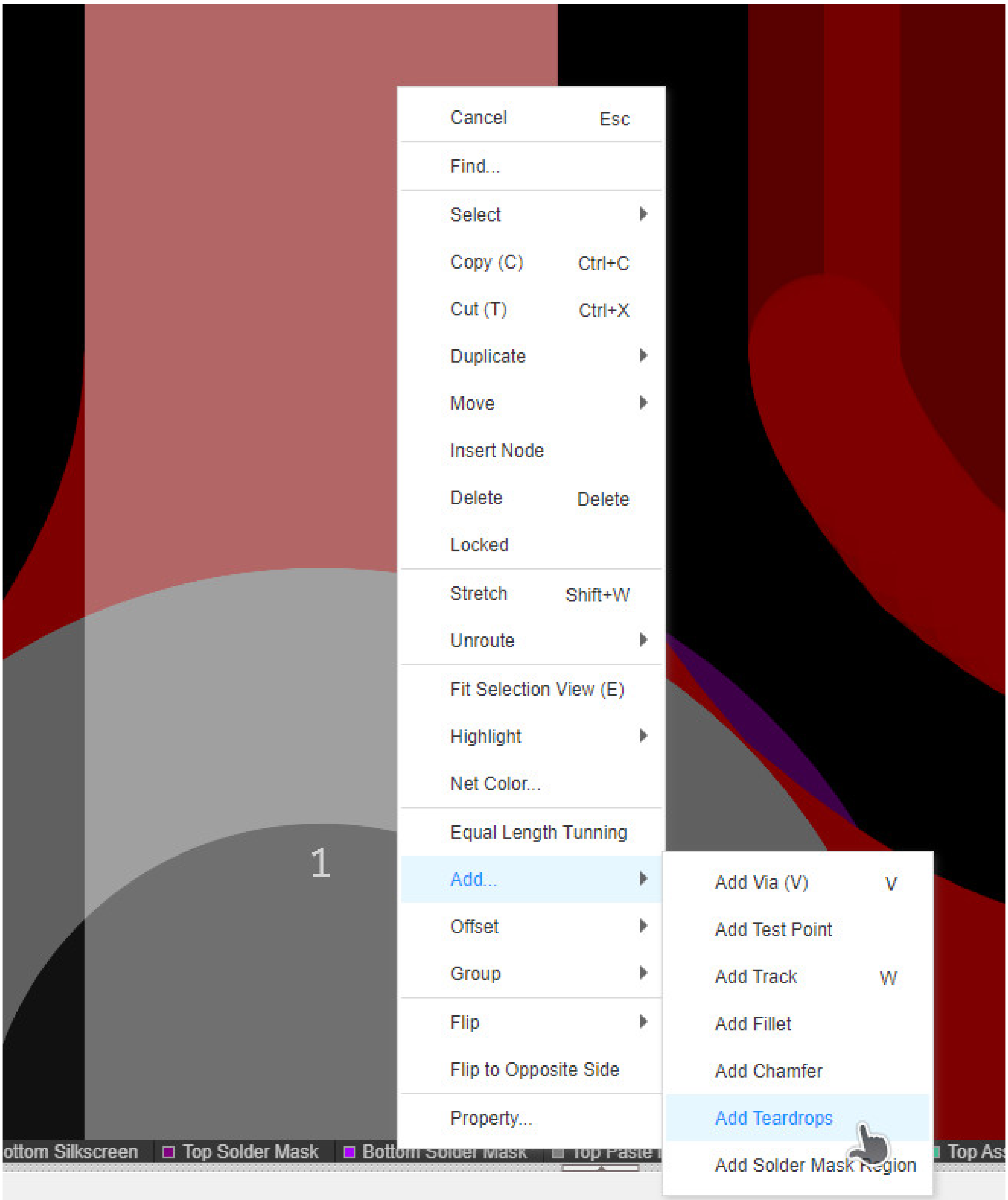
In version 2.1, our teardrop addition tool received a major update with support for trace and T-connected teardrop addition; The edge of the teardrop has also become an arc, which is more beautiful



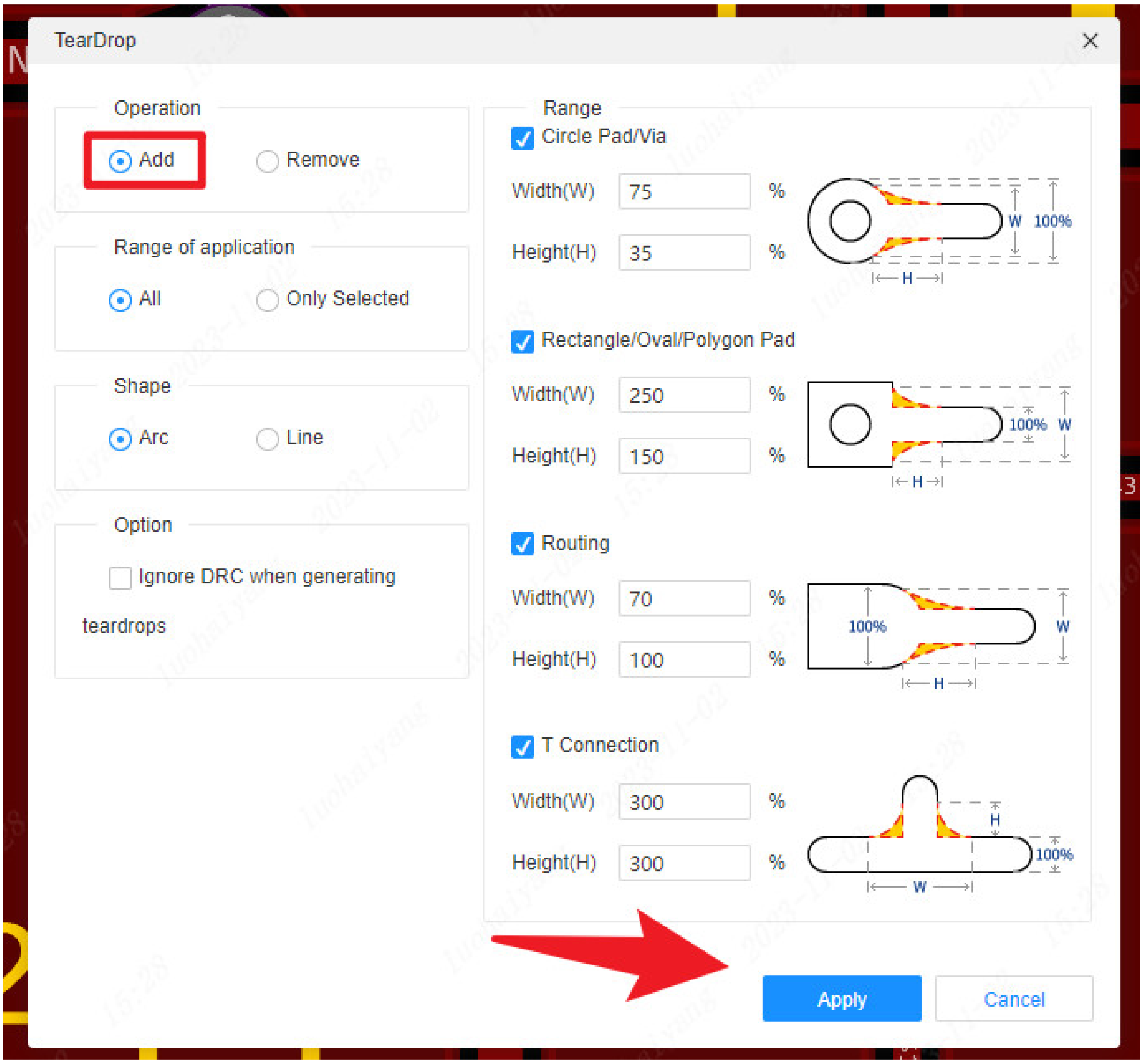
Add tear drops:

- Top Menu - Tools - Teardrop
- Selected wire - Right mouse button - Teardrop

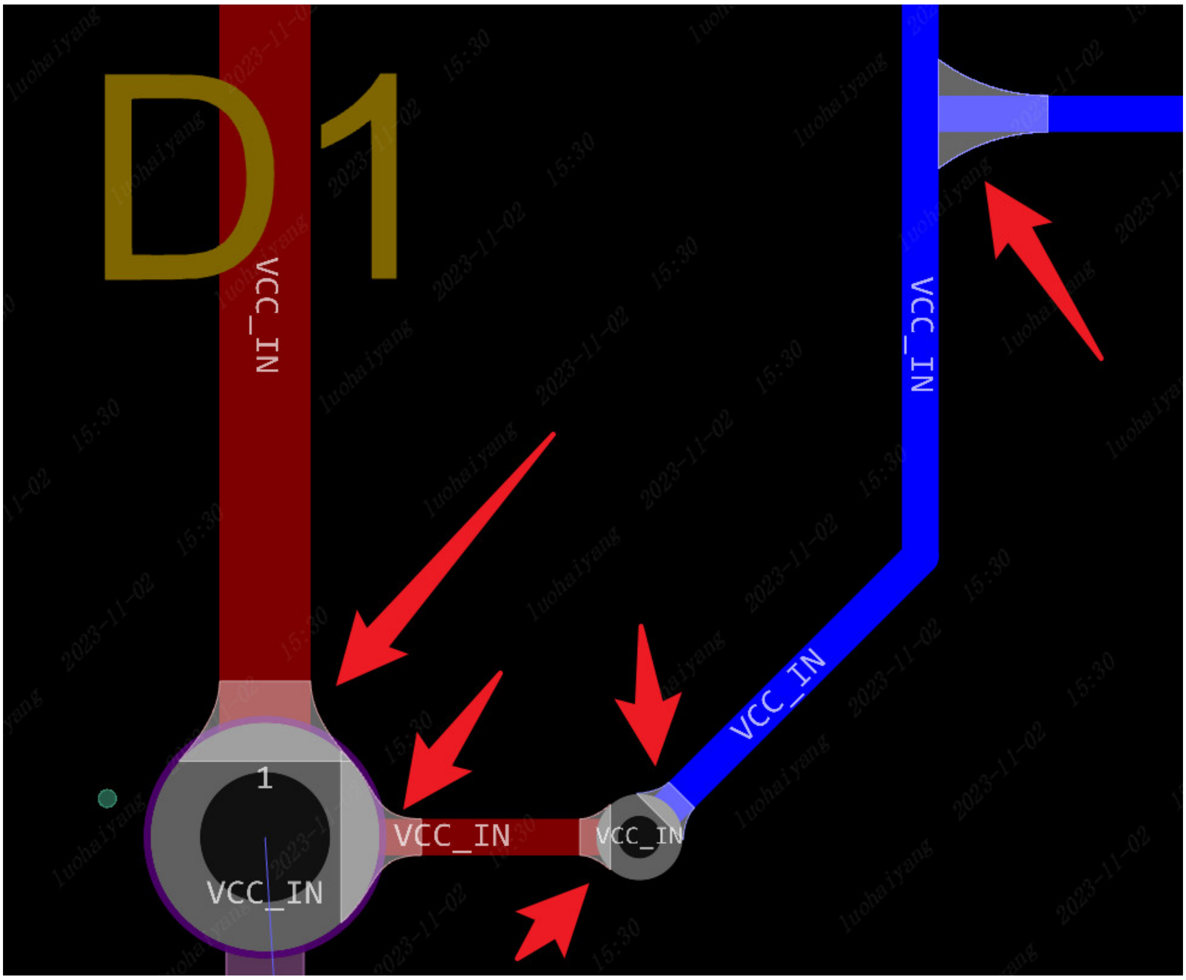

(C)
Layout (O)
Tools (F9)
Export (F10)
Order (A)
Settings (I)
Help (J)
Measure Distance Alt+M
Command...
Teardrop...
Panelize...
Remove Unused Pad...
Log...
Connect Pads
Device Manager...
Layer Manager... Ctrl+L
Copper Manager...
3D Model Manager...
JLCPCB SMT Part Selection Tool...
Soldering Assistance Tool...
DRC
Line 45°



Click OK，Can produce tear drops

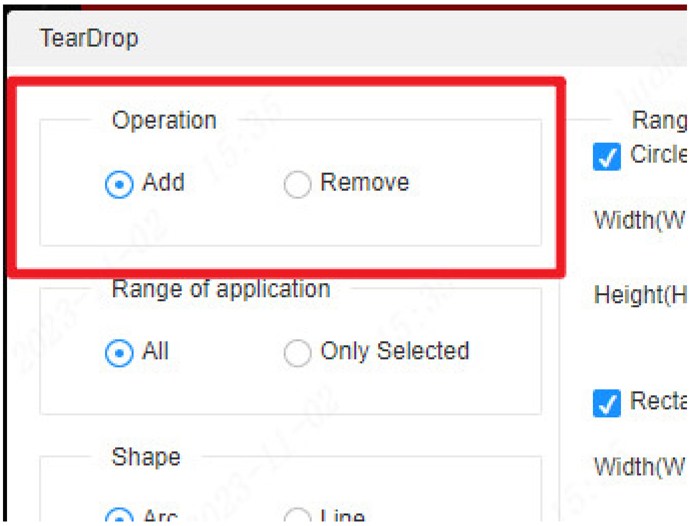


Add teardrop after successful rendering

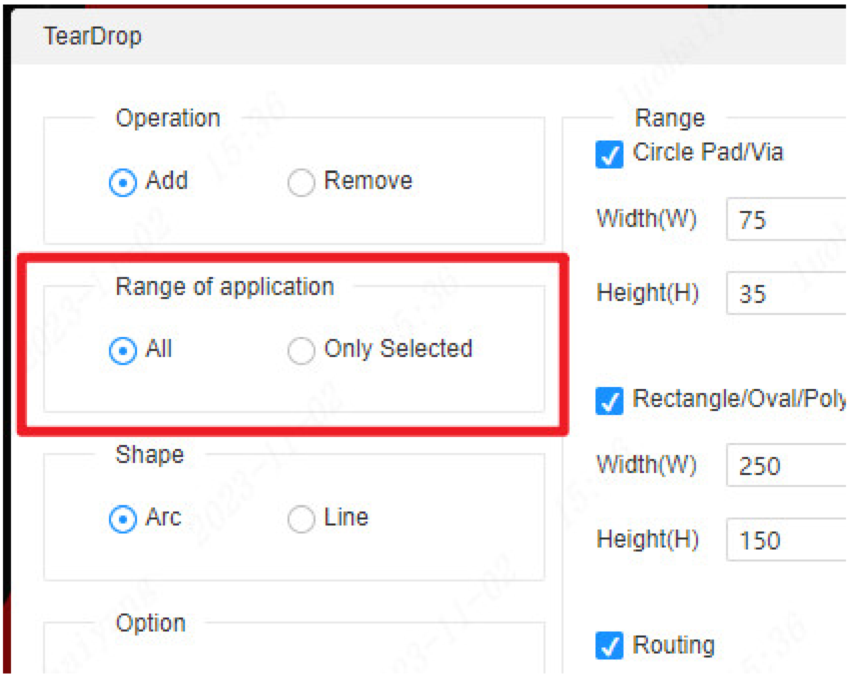


Operation

- Add： Selecting this option will remove the existing teardrops (if any) and add them again
- Remove： Selecting this option removes tears from the canvas



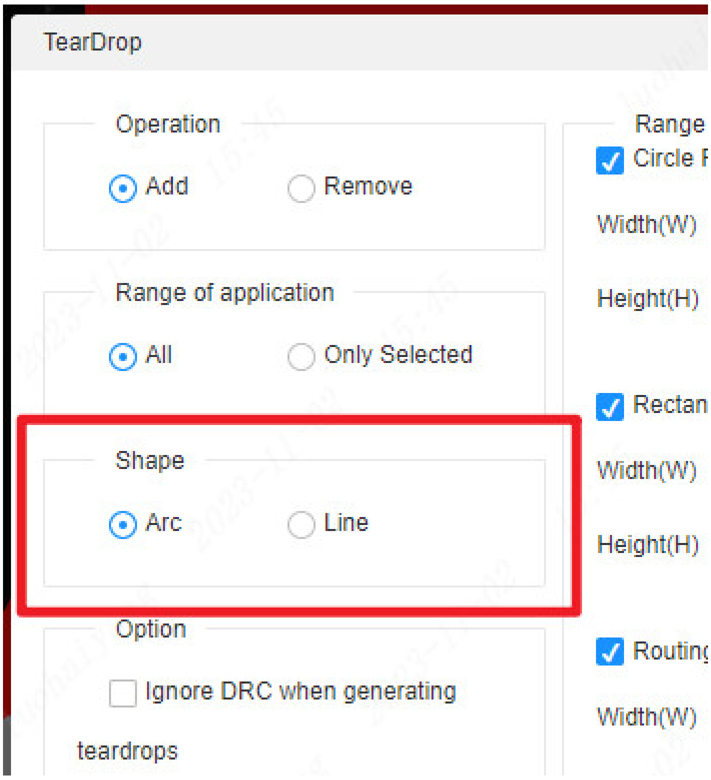
Range of application

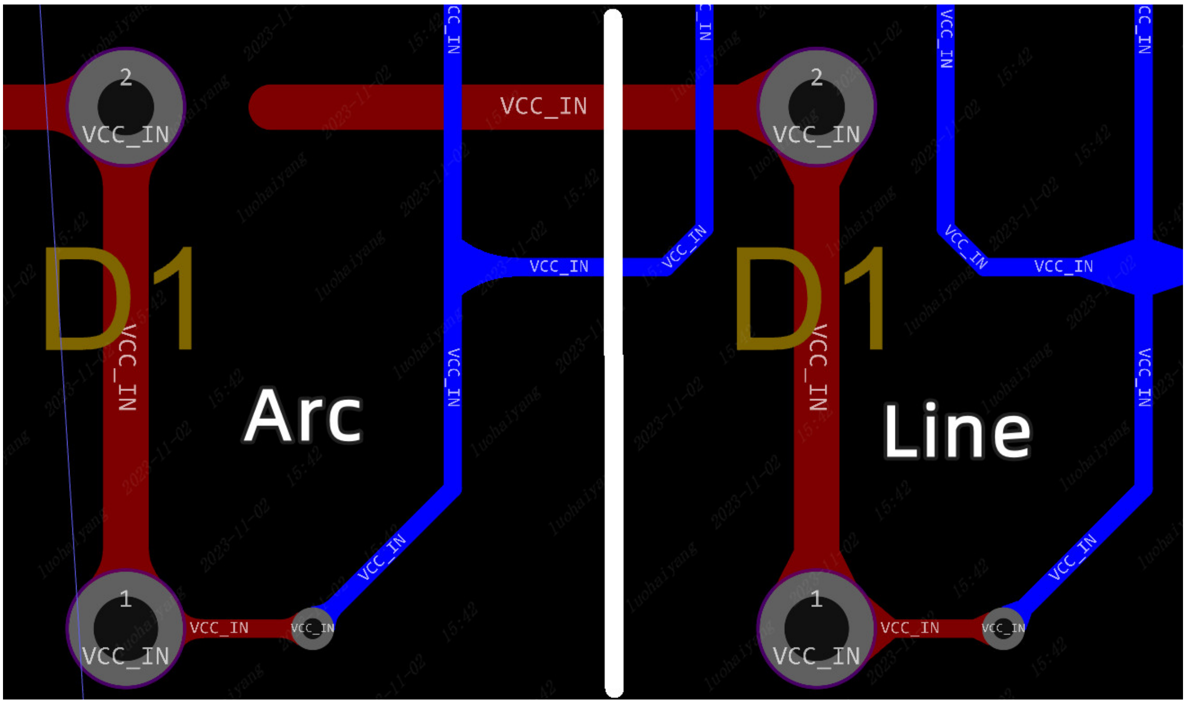


- all： The scope of adding or removing is the entire PCB
- Only Selected： The new or removed range is the selected range, which can be pre-selected before the window is opened, or re-selected after the window is opened

Shape

Supports arc or line



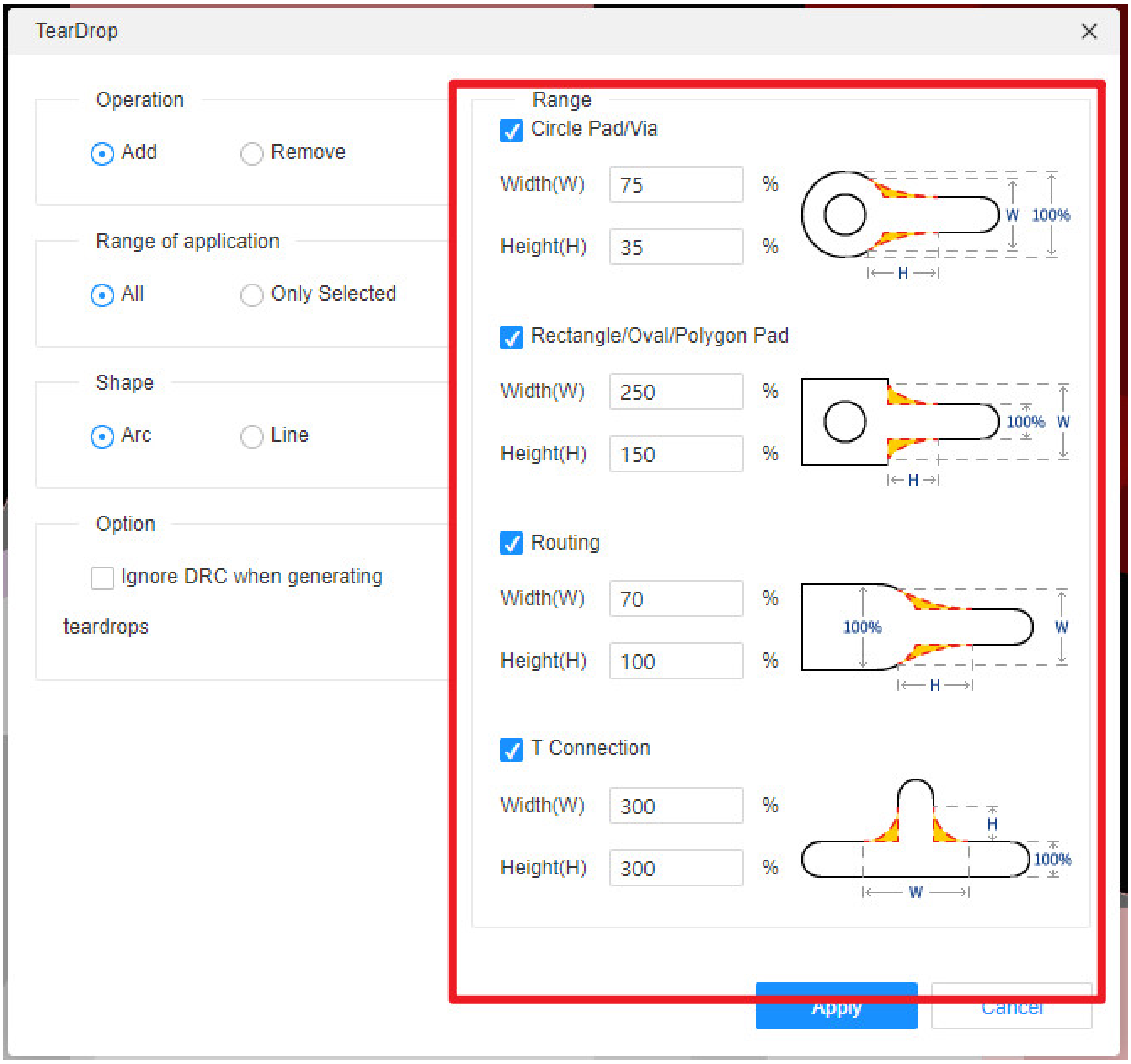


Option

Ignore DRC when generating： When this option is checked, teardrops with DRC may be generated when teardrops are generated.

Range

Here you can set the width or height of the tear drop added in different positions



Panelize

EasyEDA provides a powerful patchwork function, it can be very convenient for the patchwork operation. The assembling function provided by the editor only supports the assembling of the current board, and does not support the assembling of multiple different boards. To assemble multiple different boards, you need to use the manual assembling function.

Panelize

After the PCB is drawn, make sure that the border of the board has been drawn and closed, you can use the board assembling function.

Entry: Top menu - Tools - Panelize

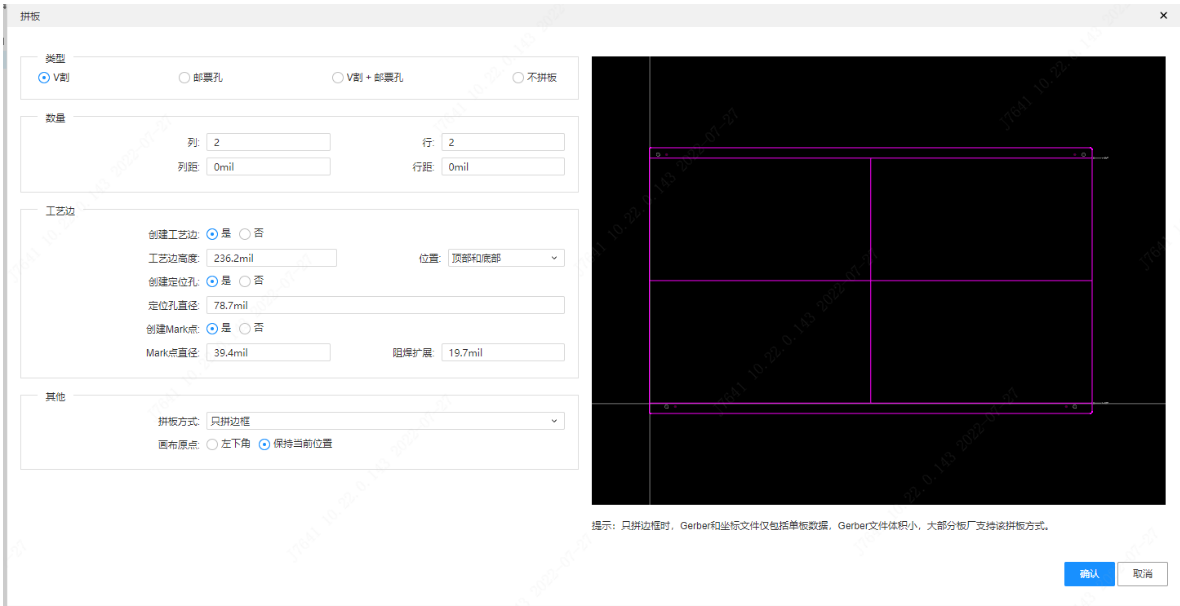
No Panelize

The parameters of the jigsaw are saved in the file. If the jigsaw is not needed, select not to jigsaw

Type of board

V-CUT Panelize :

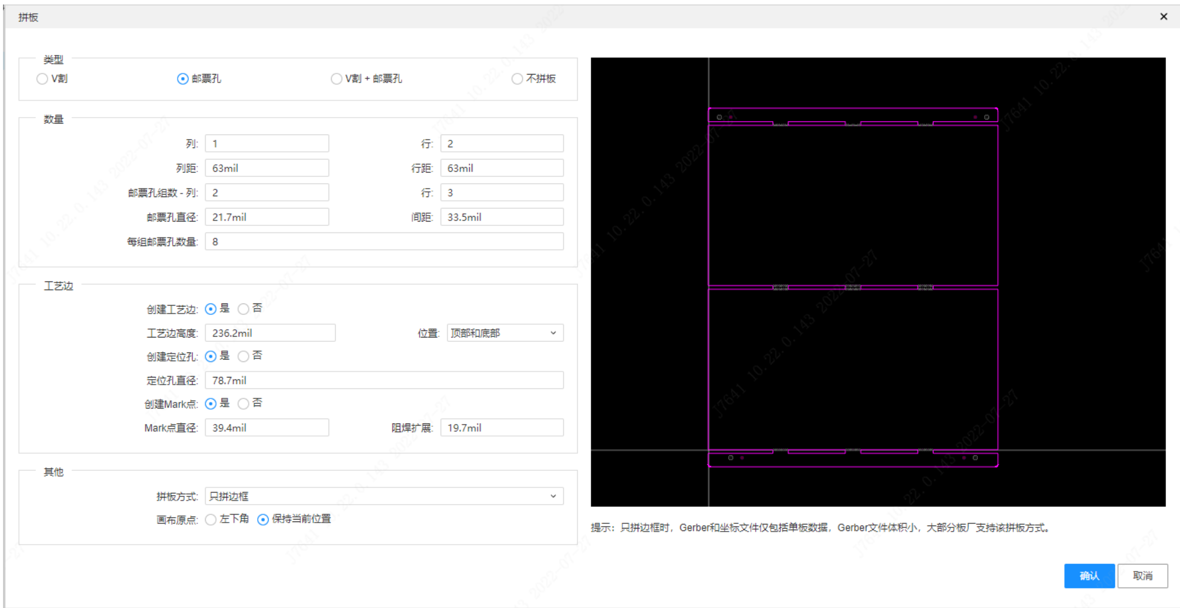
According to the design of the board, the V-cut board is adopted. When assembling the board, the array will be generated according to the border/board frame, and then the panel preview will be generated according to the other parameters set.



In production, it will cut V according to the border line, and cut a fine groove to the V-shape of the board. After completion, it is necessary to manually break off the separation board.

Stamp Hole Panelize :

Stamp hole plate is through multiple groups of through the hole, the need to join the board together, after the production is finished, the link is broken manually.



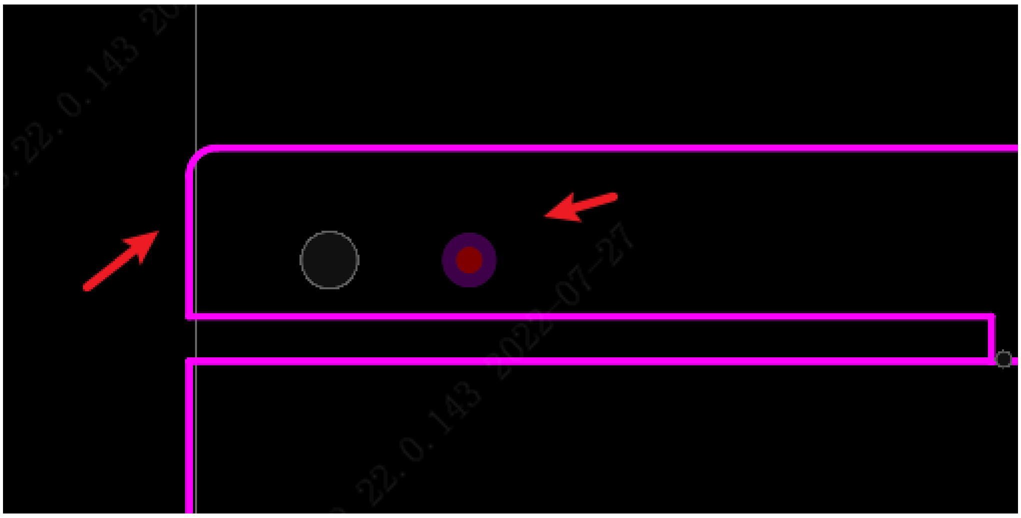
The number and size of the stamp holes can be set according to the need

V Cut + Stamp Hole Panelize：

V cut and stamp hole combination plate, more suitable for irregular shape board, and devices need to highlight the board frame scene.

Technology Edge

Support setting process edge height and positioning hole and anchor point (MARK point) size.

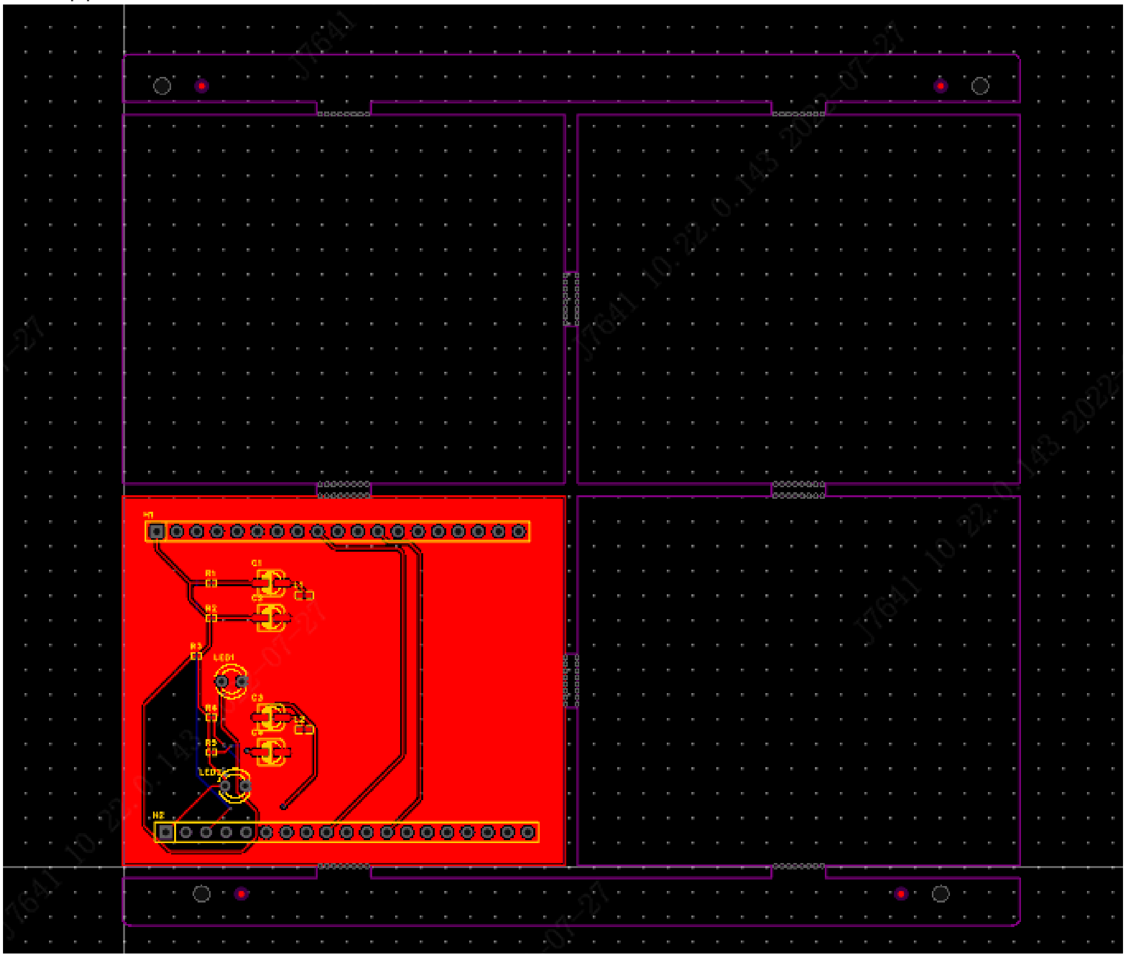


Process edge is usually used for SMT placement when the board enters the placement track, and also facilitates SMT placement positioning. You do not need this option if you are working on the JLCPCB patch.

Other Options

The Panelize mode supports two types:

- Only spell borders. This method is recommended. When only bounding frame is assembled, 2D and 3D preview, Gerber and coordinate files only include board data. Gerber files are small, and most board manufacturers support this assembling method. Calicron supports this approach.



- Concatenate all objects. When splicing all pixels, Gerber and coordinate files contain all board data. Gerber files are large, and some board manufacturers need this method.

Manual Panelize

When automatic assembling cannot meet the needs, such as the need to assemble different boards, Yin and Yang assembling (such as Tai chi graphics), manual assembling can only be used.

It should be noted that if the PCB of manual assembling is too large, it will lead to PCB stalling and difficult alignment, so it is recommended to use the automatic assembling function.

Steps:

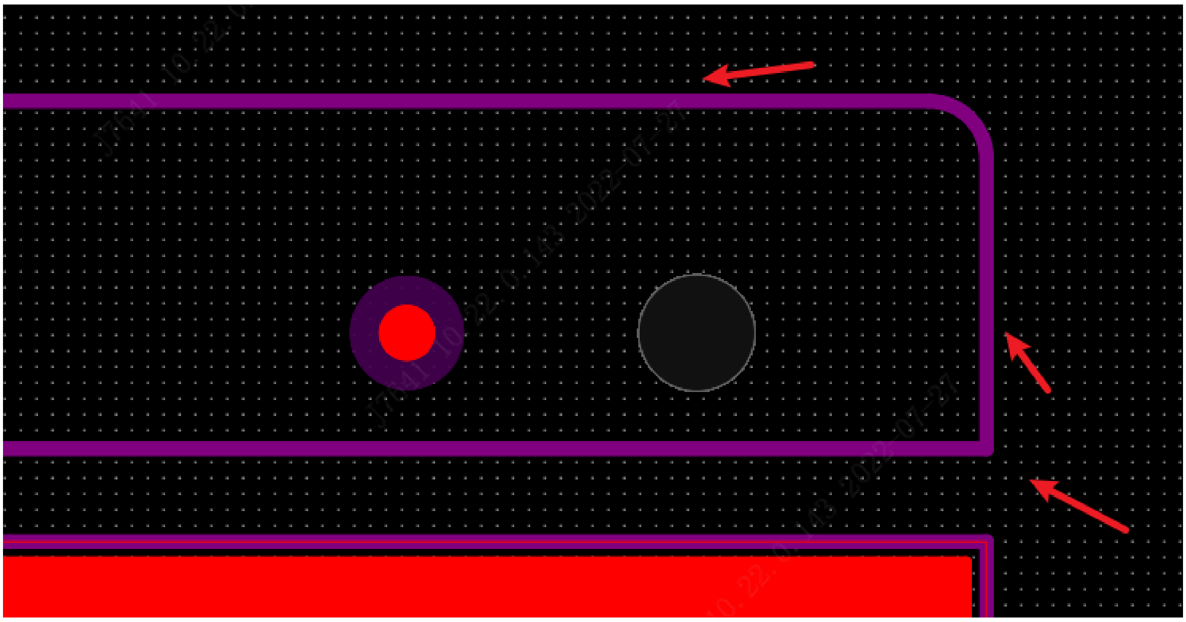
1. Open the PCB that needs to be assembled, or other different PCB

2, box select or CTRL+A select all PCB contents, CTRL+C copy

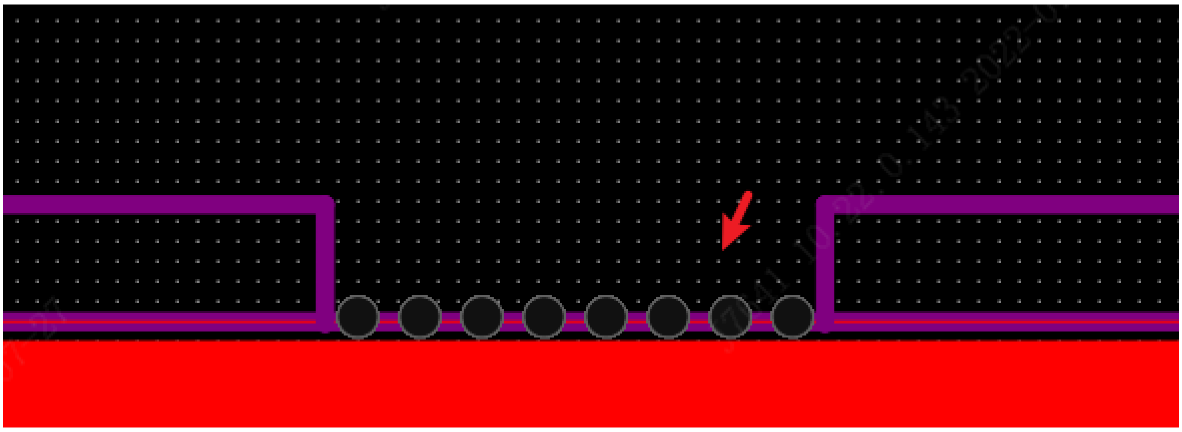
3. Switch to the PCB that needs to be assembled, CTRL+SHIFT+V paste. The shortcut paste will automatically hide the flyline layer and the hold bit will not increase.

4, you can paste when the point, or placed after the batch box to choose the point, you can also put the board combination after moving the point.

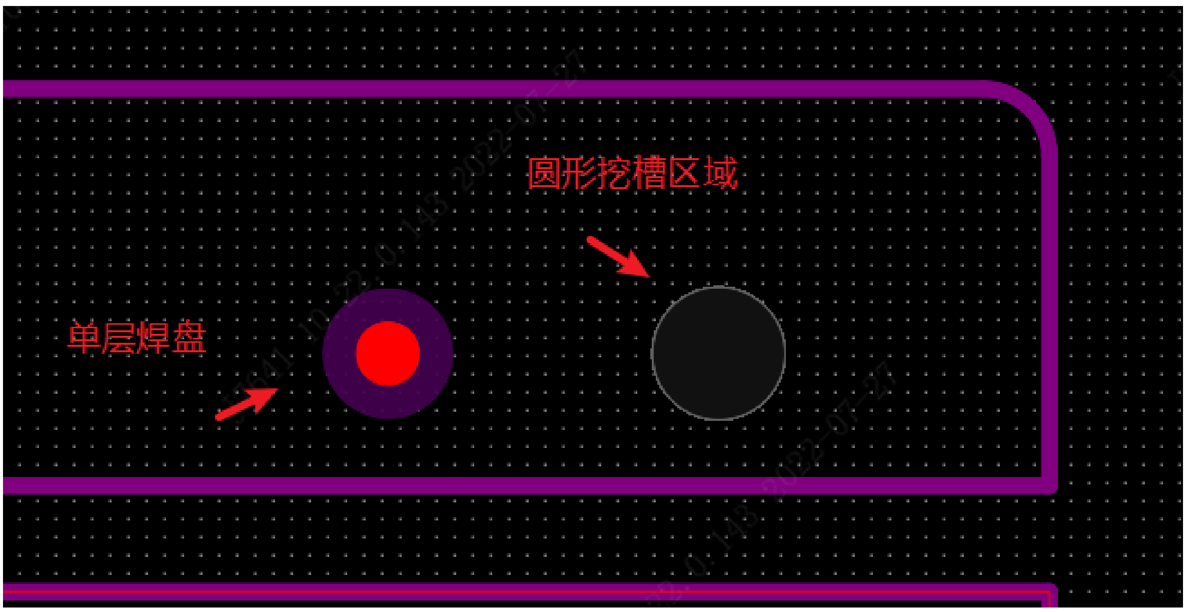
5, if you need the process edge and V secant, then draw the border line in the border layer (plate and frame layer).



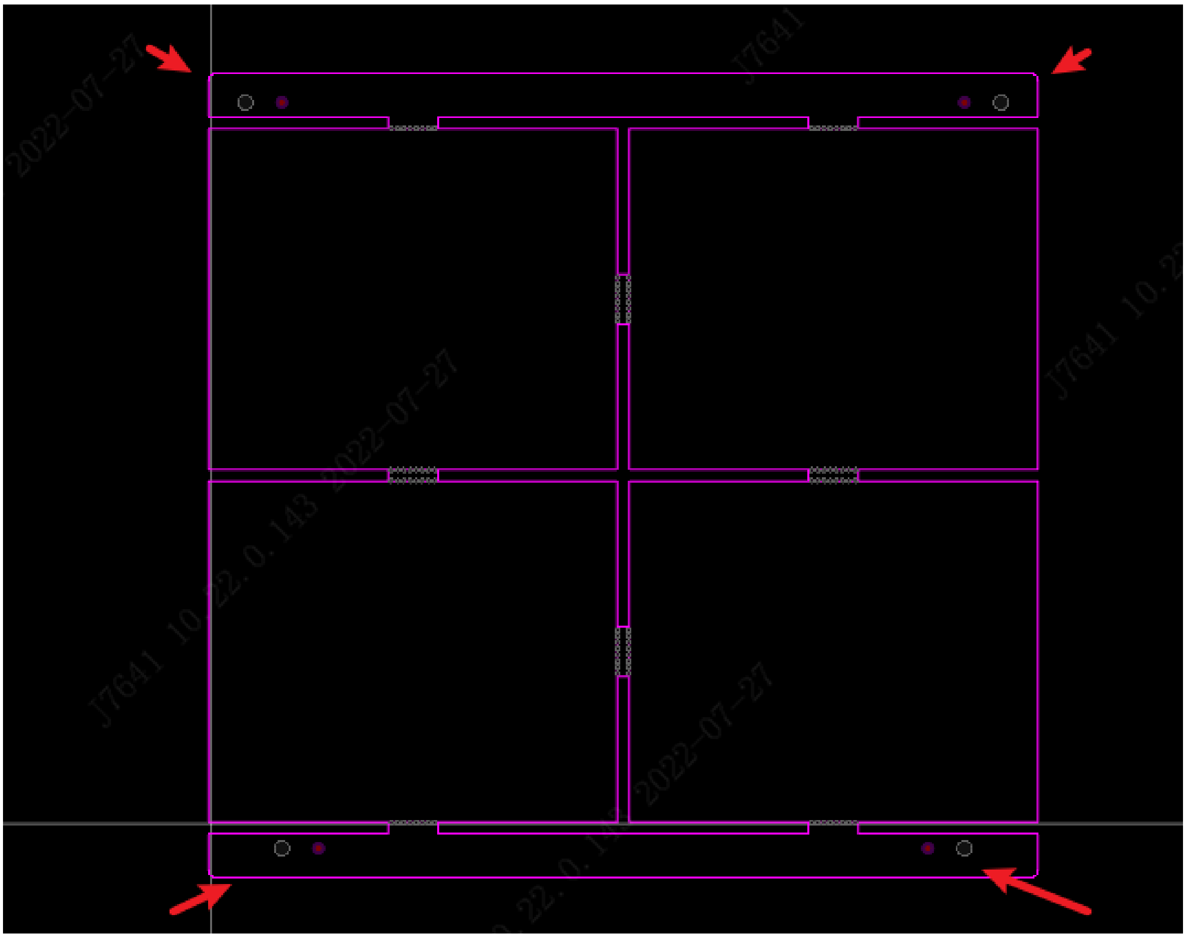
6, if you need a stamp hole, place a circular grooved area, set the size and spacing, on the border line. The diameter of the stamp hole is 0.55mm, and the spacing is 0.85mm



7. If positioning holes and positioning points are needed, circular grooving areas (positioning holes) can be placed, and single-layer solder pads (positioning points) can be added. The positioning point shall be placed on both the top and bottom signal layers. The diameter of the positioning hole is 2mm, the diameter of the positioning spot welding plate is 1mm, and the welding resistance extension is 0.5mm.



The positioning holes and positioning points need to be placed diagonally, and the distance between the upper and lower sides and the left and right boundaries should not be the same, otherwise the SMT cannot be accurately located, as shown in the figure.



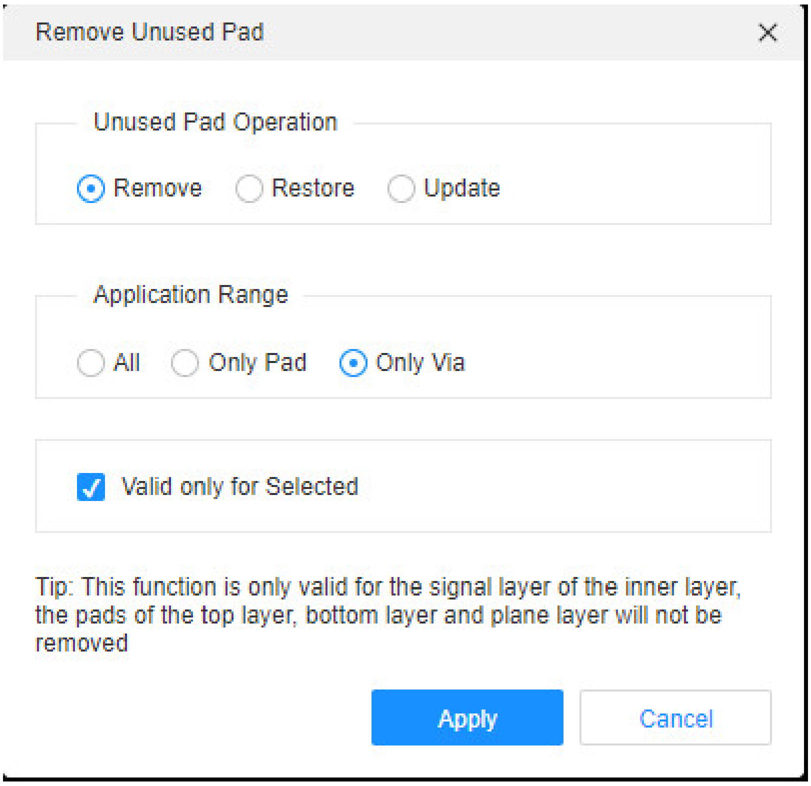
Notice:

1. The board with an internal plane layer cannot be used by manual panels, otherwise the inner electric layer cannot take effect normally.Because the implementation method of the internal plane layer must be available with the full border, manually copying and paste multiple borders in the border's internal plane blocks will be abnormal, and the network cannot inherit. It is necessary to change to the signal layer with copper instead of internal electricity segmentation.
2. The editor's own panel and manual panel cannot be performed at the same time.

Remove Unused Pad Shape

Functional entrance

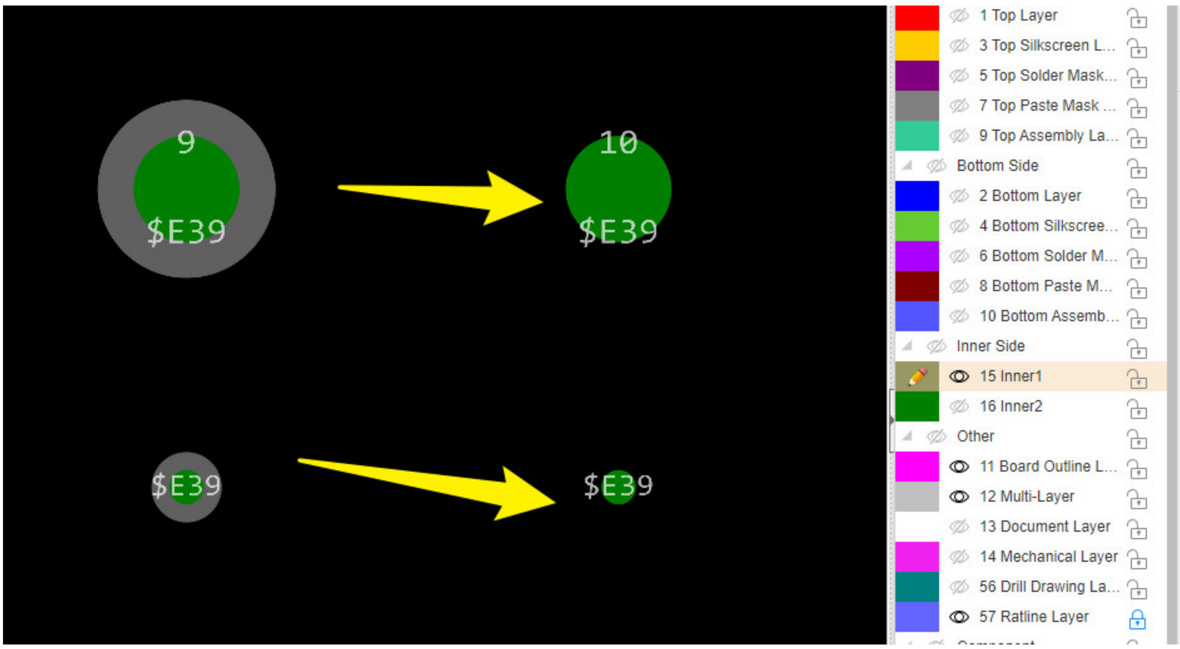
- Top Menu -Tools -Remove Unused Pile
- Shortcut keys (need to be added by yourself)



Unused Pad Operation

You can choose what to do with the unused pad or the inner copper ring that is not used or over -perforated

- Remove： After selecting and applying, the multi -layer pads of the network and the inner copper ring without any connection with the interference will be removed.



- Restore： Will produce the opposite effect to remove
- Update： Depending on whether the pads on the current canvas are connected to the inner copper ring of the vacant inner layer, or the copper ring connected to the dollar recovers

Application Range

-All: include pads and holes
-Solocent pads
-Stopto

Valid only for Selected

The default is to take effect only in the selection, and when the window is opened, you can continue to perform the selection on the canvas. When not checked, the effective range is the entire PCB

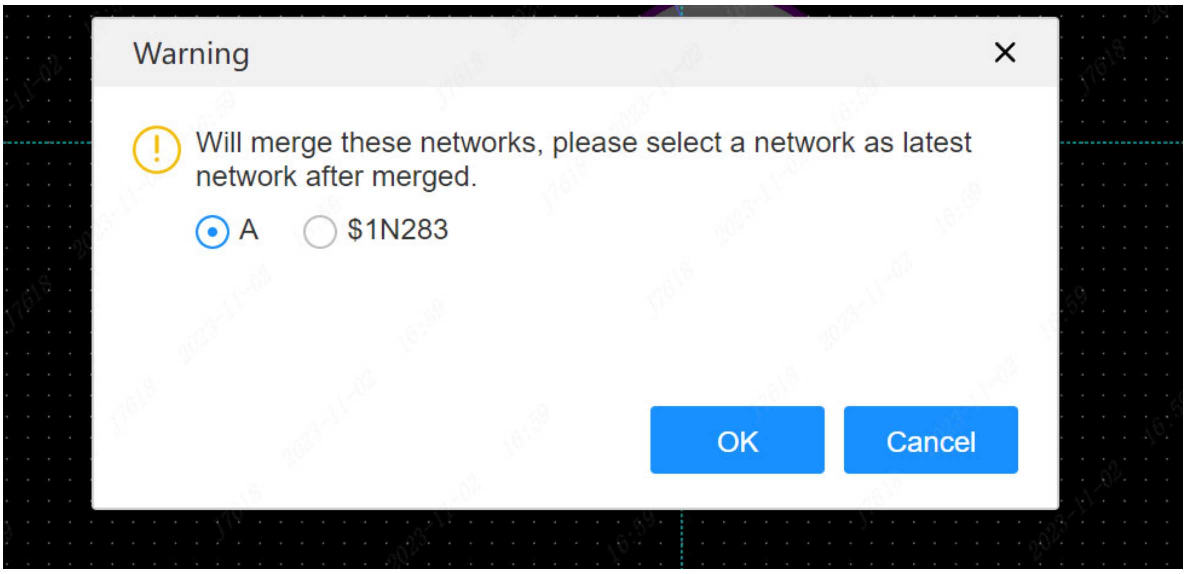
Connect Pads

The pad connection function is mainly used for the design of non -original rational maps, which is convenient to quickly generate the network in PCB.

Steps:Top Menu -Tools -Connect pads

After entering the connection pad mode, click two different pads in turn to connect these two pads

- Connect two pads without networks: generate a new network for them
- Connect a pad with a network and a network without a network: a network of pads without networks as another pad
- Connecting two pads with networks: warning prompts will appear during connection. The merged network name needs to be selected. The merger is merged into this network

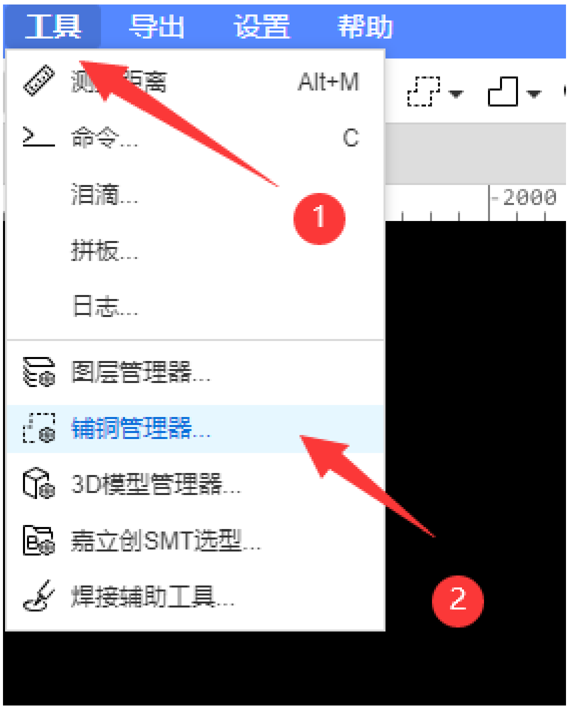


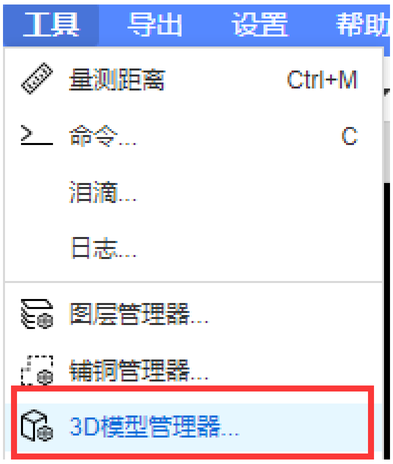
Copper Manager

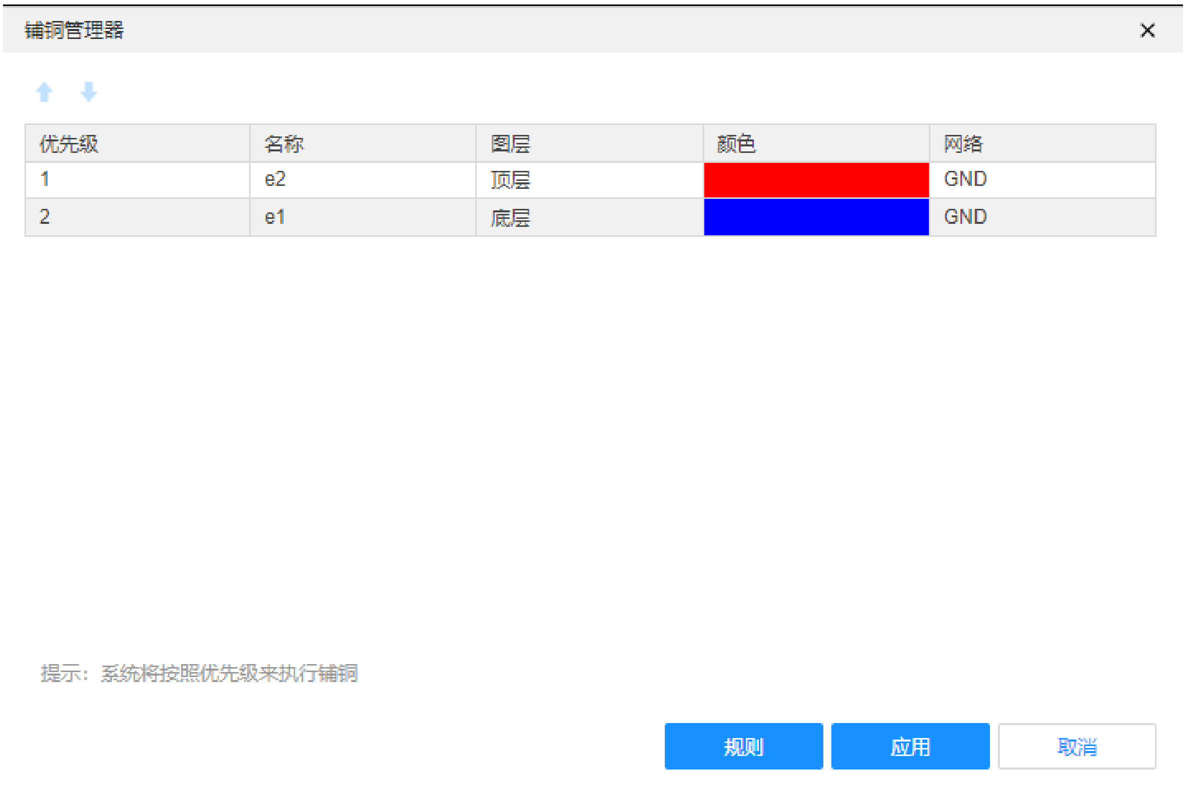
The copper manager can manage copper sheets in batches. Increase the priority of copper plating, modify the network, modify the layer name, and change the order of copper plating.

Operating procedures:

- **In the top menu - Tools - Copper Manager**







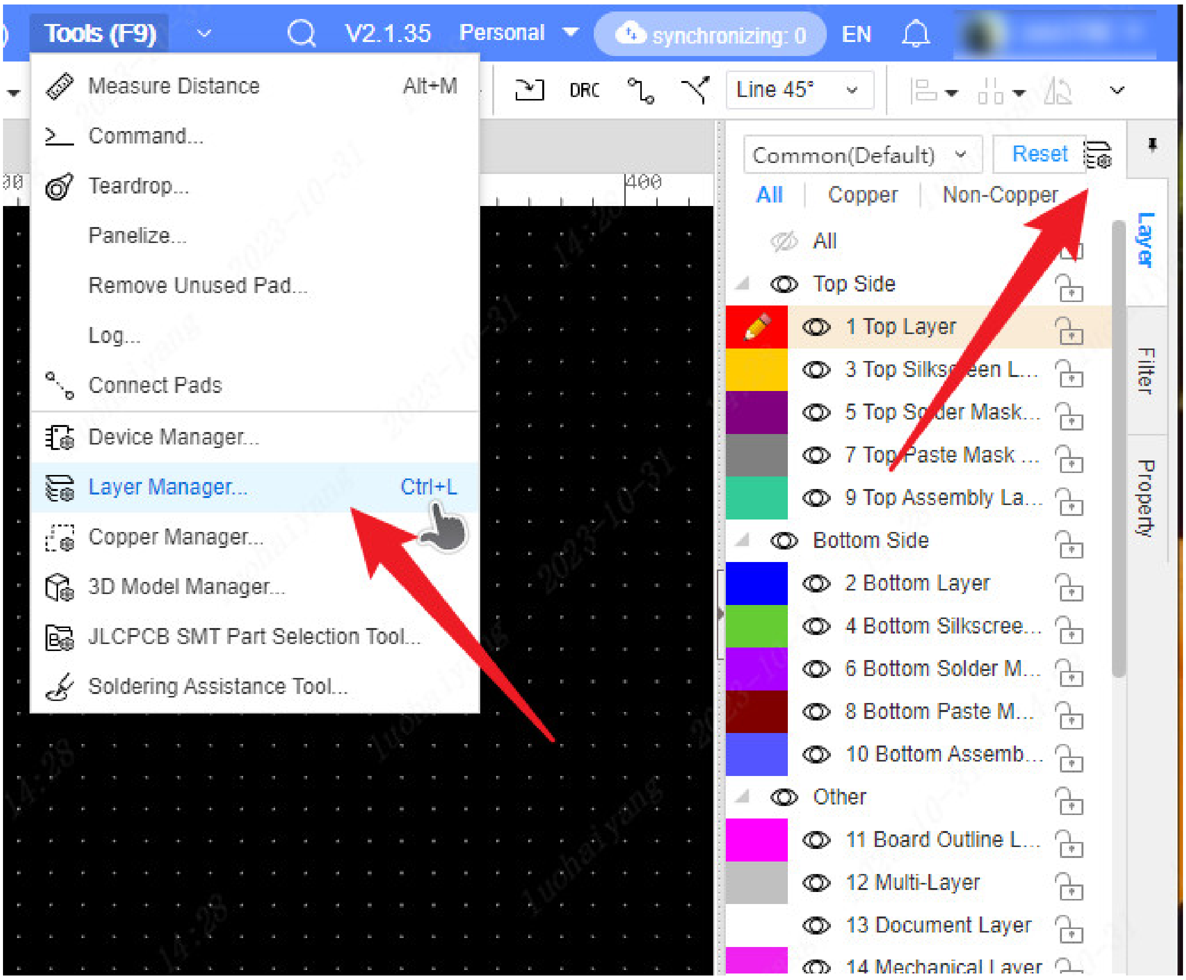
Note: The order of copper is based on the priority of copper . The higher the priority, the first to lay the copper, and the result of the copper is first come, first served.

Layer Manager

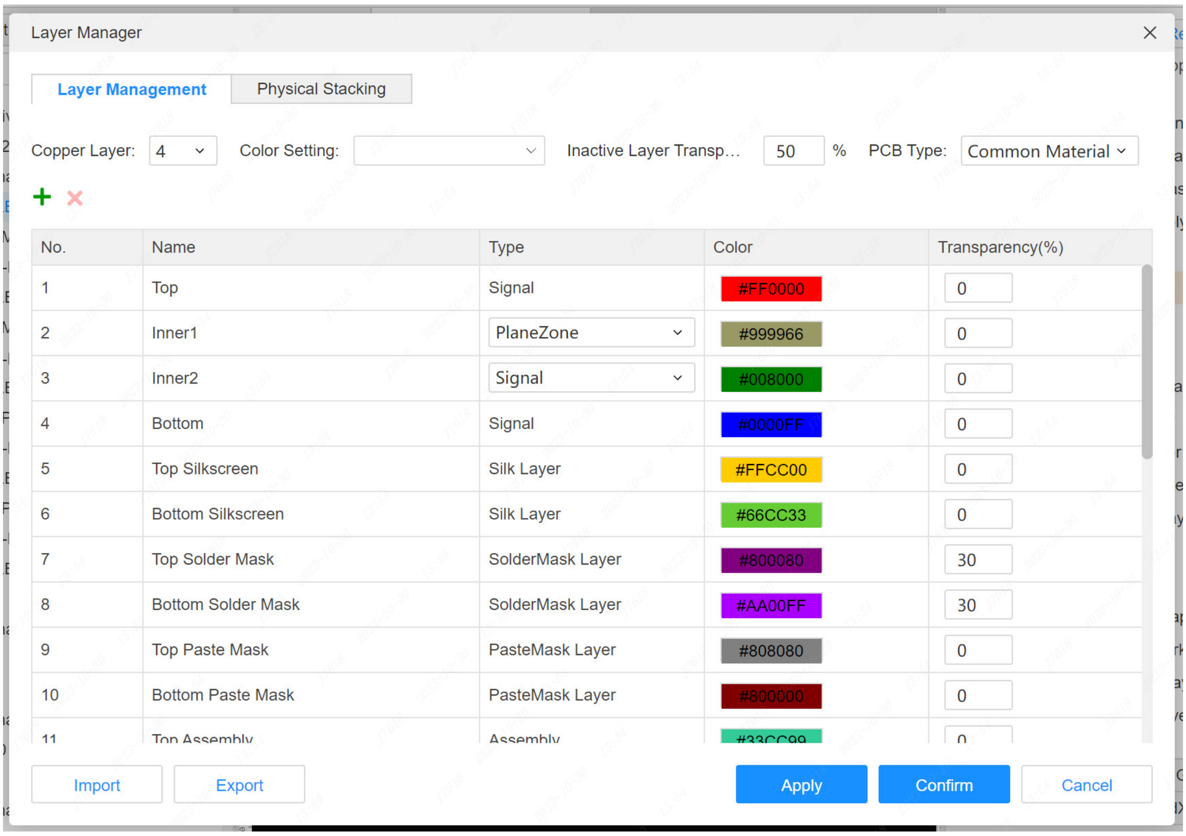
EasyEDA Pro edition provides users with a layer manager, users can quickly configure PCB layer related content in the layer manager, such as the number of PCB layers, layer color, type, transparency and so on

Steps:

- Top Menu - Tools - Layer Manager
- Click the layer manager icon in the upper right of the layer property bar on the right

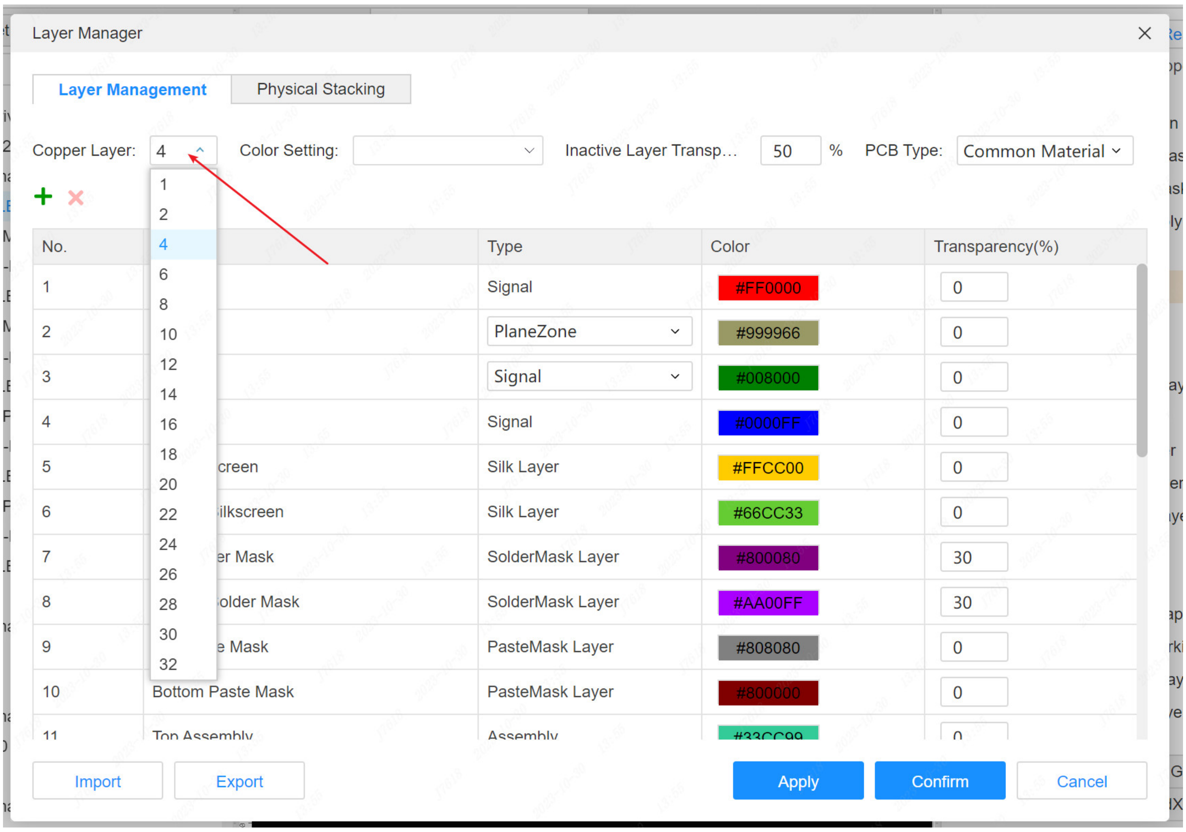


The layer manager window pops up, where you can set the **transparency, name, type, and add layers** of the layer.

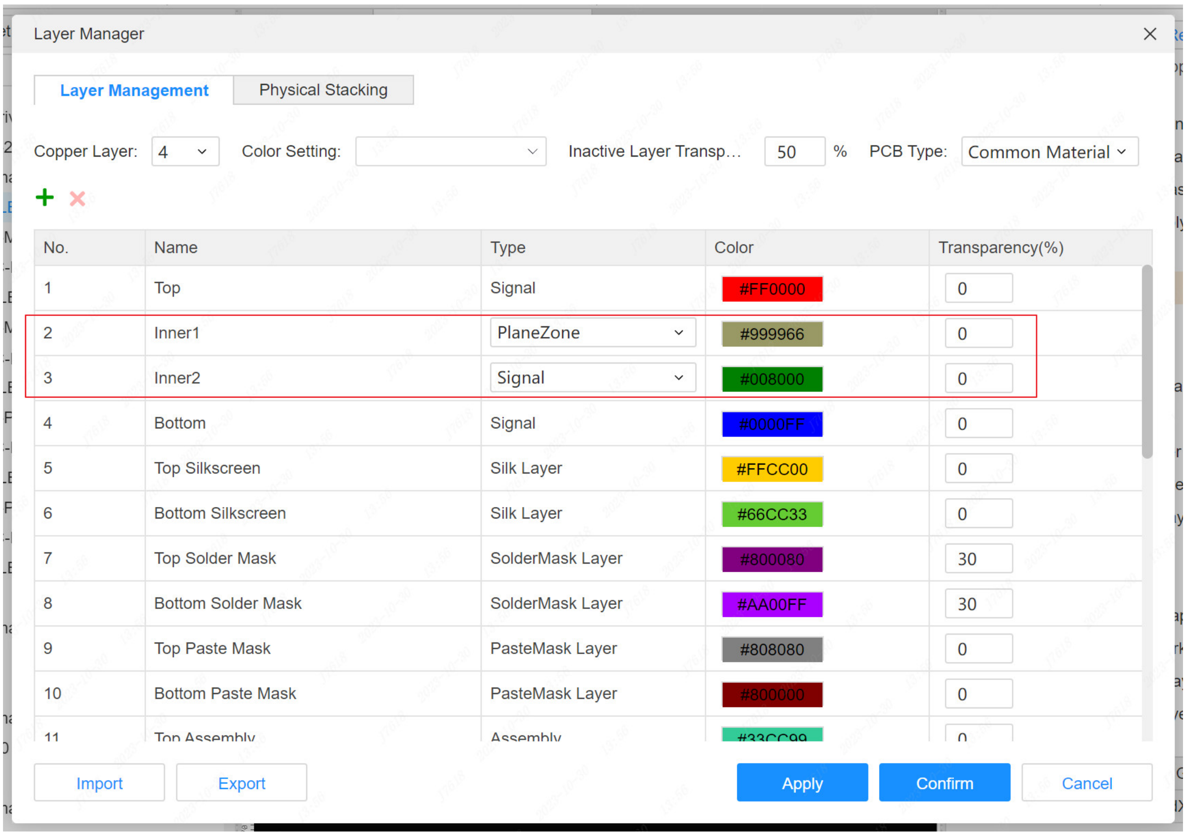


add layer

Select the number of layers to add



After generation, the layer manager page will have two more layers

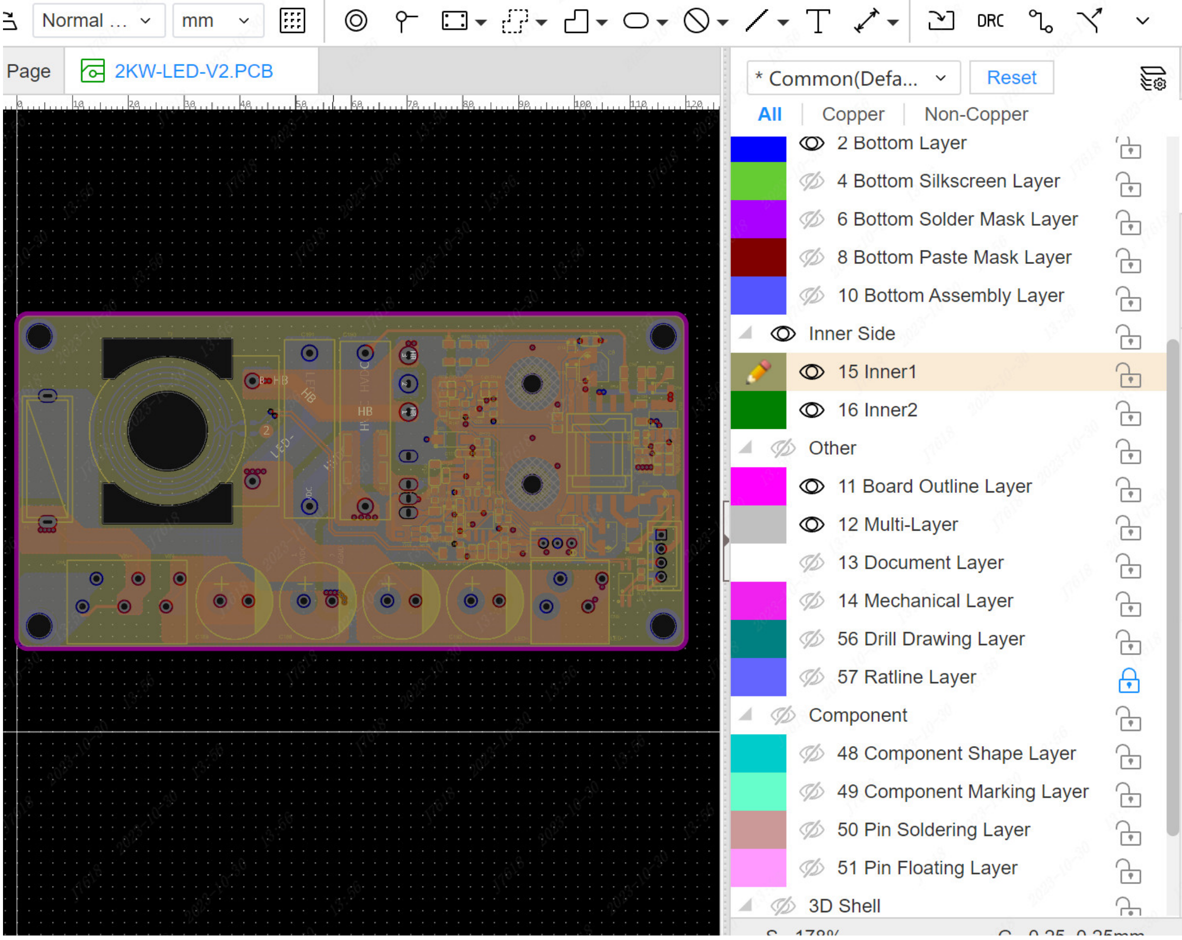


The newly added layer type can be modified, which can be modified to the signal layer and the internal electrical layer. The copper of the printed board is retained where the lines are drawn, and the copper deposits where there are no lines are removed. Such as the top layer and the bottom layer, the signal layer is the positive film.

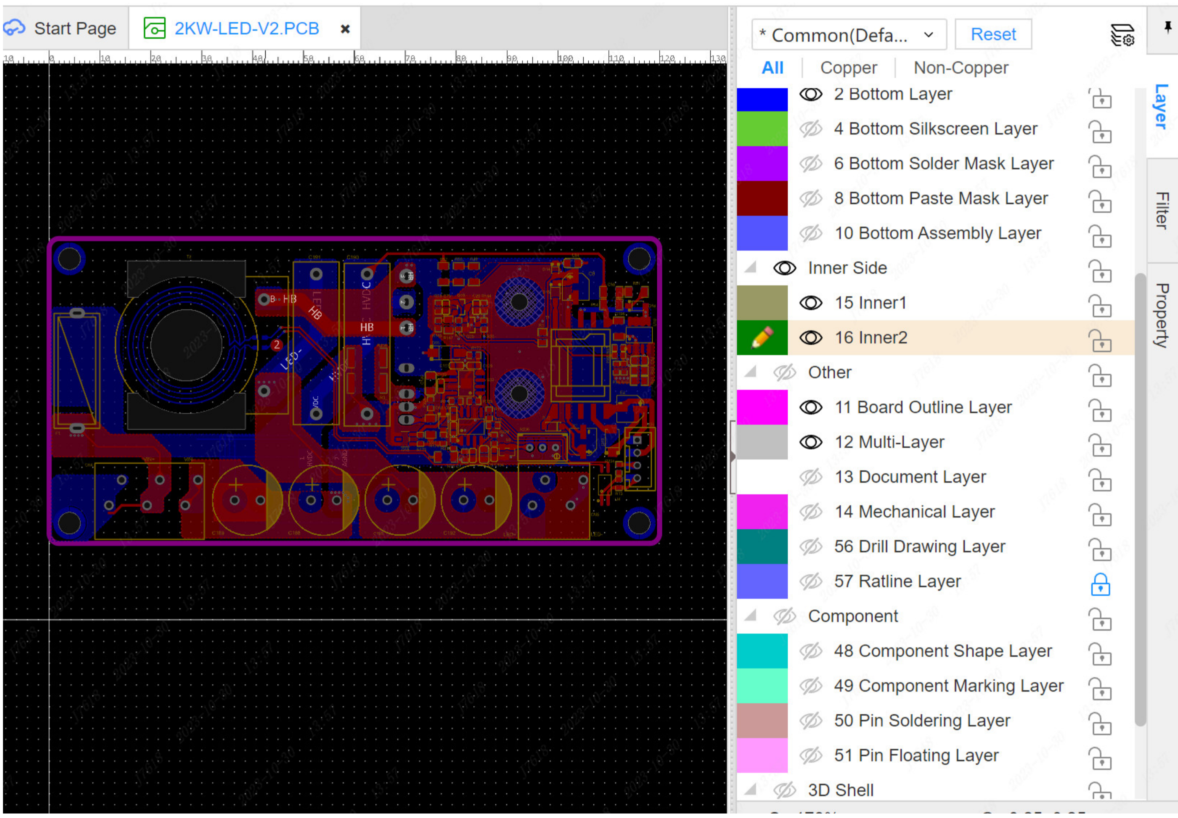
- **Signal layer**: It is also a positive layer. The pcb signal layer is the same copper conductive layer as the top and bottom wiring, but it is a wiring layer sandwiched between the top and bottom layers.
- **Internal electrical layer**: Also known as the plane layer or the negative layer, it is the internal power supply and ground layer (and is connected to each layer through through holes), and the internal electrical layer is divided by "line" primitives. Negative film effect: Where the lines are drawn, the copper coating of the printing board is removed, and the copper coating where there are no lines is retained. Traces or other objects placed on these layers are copper-free areas, ie this working layer is negative. The internal electrical layer of EasyEDA Pro is drawn in negative film mode, but it is positive film output when outputting the Gerber file, please pay attention.
- **Note**: The professional version does not support automatic reconstruction of the internal electrical layer, you need to select the internal electrical measurement to manually rebuild.

In the editor, there is a clear difference between the internal electrical layer and the signal layer

Internal electrical layer

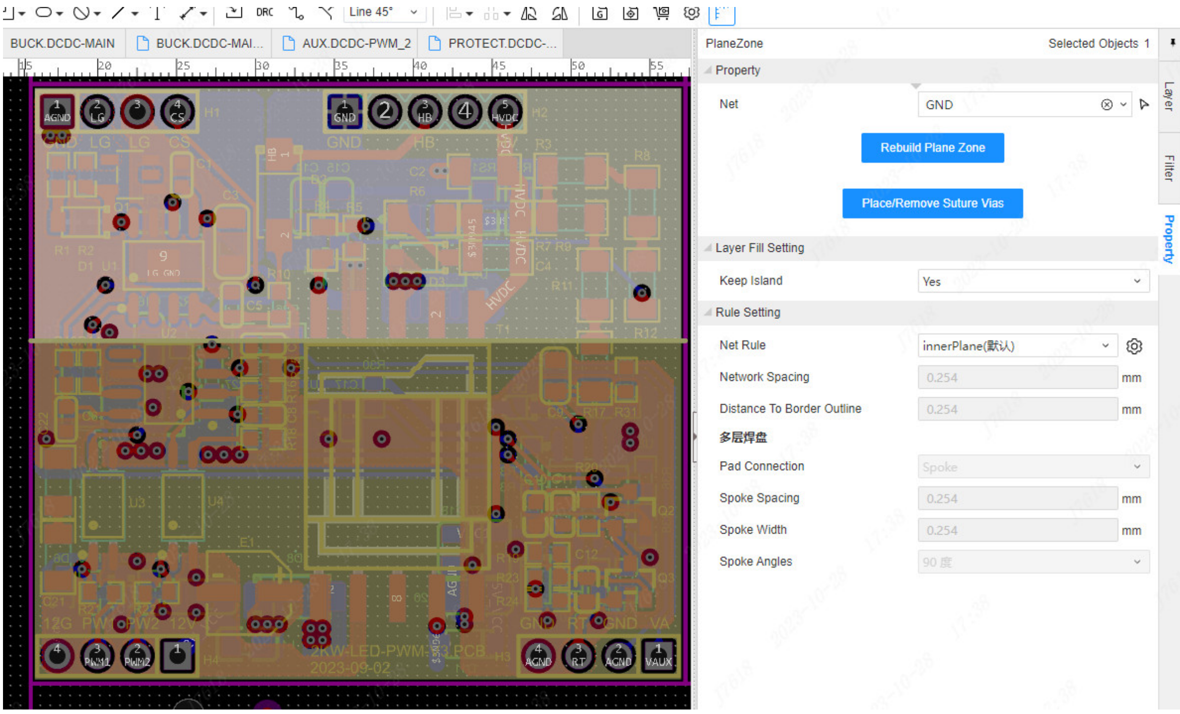


Signal layer



The inner electrical layer is like a piece of copper, which can be networked and divided.

PlaneZone Property



Switch to the internal electrical layer and click the copper skin. On the right panel on the right, you can switch the attributes of the internal electrical layer.

net:After setting up the network of each block, rebuild the internal electrical layer. The copper skin of the block will be connected according to the corresponding network to avoid objects of different networks.

Rebuild Plane Zone:After segmenting the block through the lines, click the entire copper skin. After the function is rebuilt, it will be divided into multiple blocks.

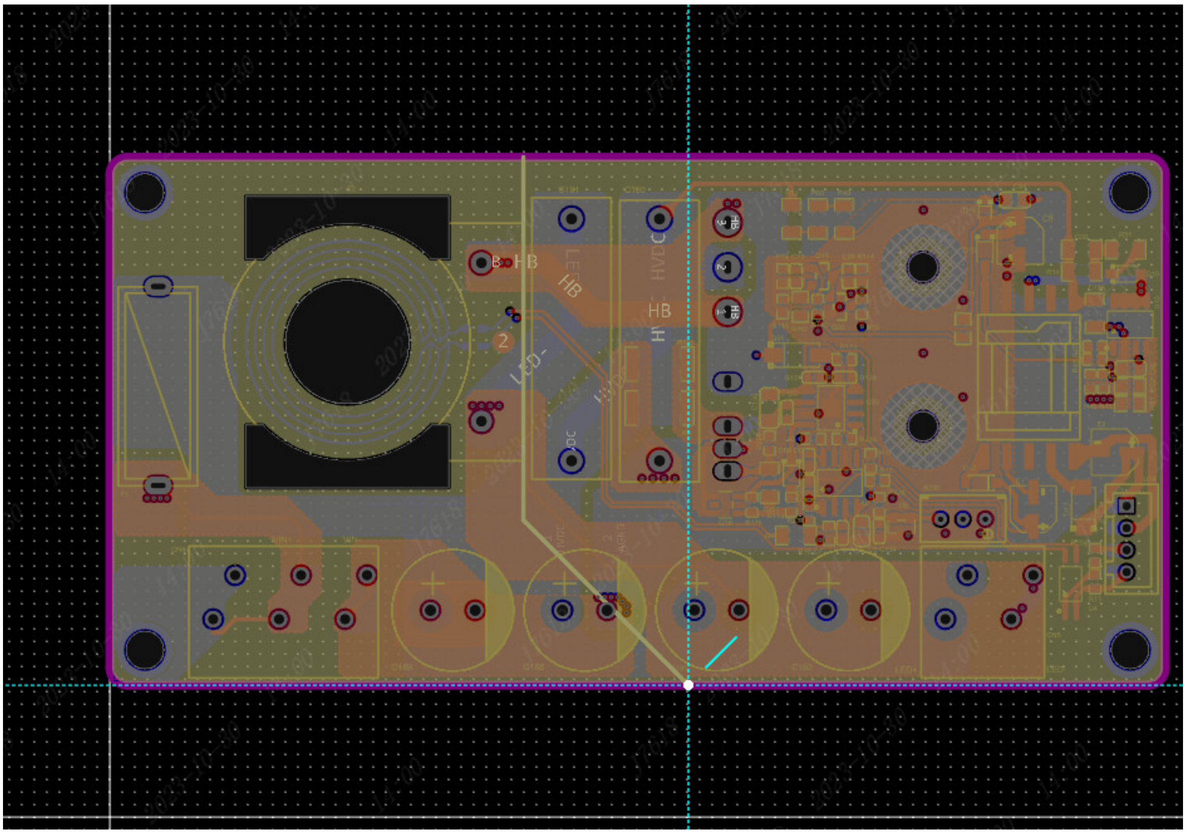
Place/Remove Suture Vias: Place and remove the Suture Vias within the copper leather range of the plane zone.

Keep Island: After it is set, it will be retained without any block of the same network object.

Rule Setting: After switching the net rule, the correspondence of the corresponding network in all internal electrical layers will be changed together.

Split the inner electrical layer

To divide the inner electric layer, you need to use a broken line to divide it, you cannot use a wire, and after the division is completed, you need to manually rebuild the inner electric layer or the shortcut key `Shift`+`B` to rebuild all copper



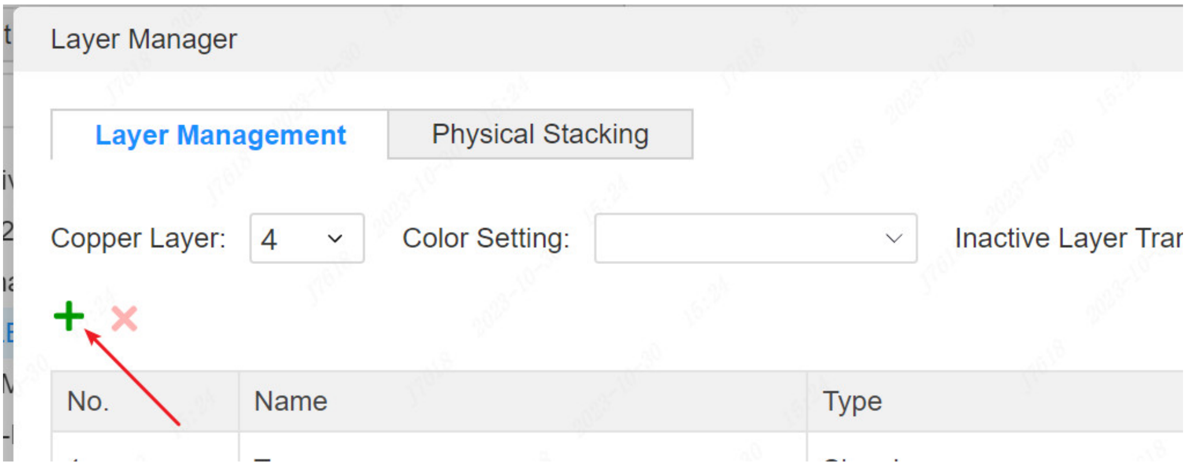
The divided inner electrical layer can set up multiple networks.

Note: The wire drawn by dividing the inner electrical layer must be a complete closed loop.

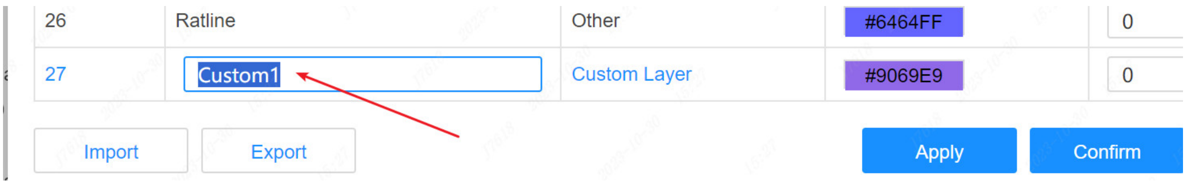
Add custom layer

easyEDA supports the creation of up to 30 custom layers. The custom layer is generally used for additional information recording, function and document layer, mechanical layer, and assembly layer similar.The custom layer does not export Gerber default, does not participate in physical production, you can use custom export when exporting Gerber, you can choose whether to export.

Click the plus number icon above the list to create a custom layer.



The name of the custom layer can be customized



Layer definition

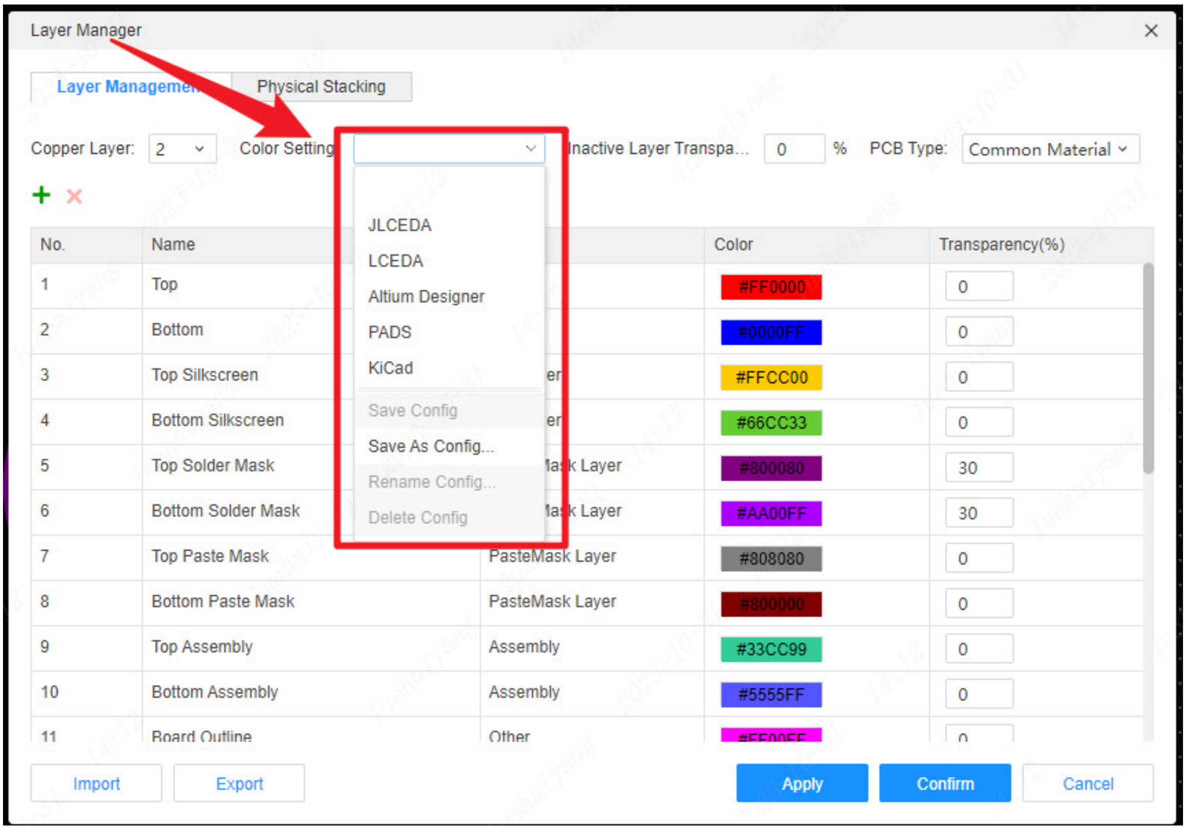
Jialin EDA supports clear layer definition, which can refer to the following layer type and its role.

- **Top/bottom**:The copper foil layer on the top of the PCB board and the bottom surface is used for signal wiring.
- **Inner layer**:Copper foil layer, signal wiring and copper.It can be set to the signal layer and the internal electrical layer.
- **Top silkscreen layer/bottom silkscreen layer**:The white character layer printed on the PCB board.
- **Top paste mask/Bottom paste mask**:This layer is a layer used for steel mesh for pad pads to help welding and determine the area size of the tin paste.If the board does not require a patch, this layer has no effect on production.Also known as the welding layer when the main film process.
- **Top solder mask/Bottom solder mask**:The top layer and bottom layer of the board cover the oil layer, which is generally green oil. The role of green oil is to prevent unnecessary welding.This layer is a negative film drawing method. When you have a wire or the area do not need to cover green oil, it will be drawn at the corresponding location. After the PCB is generated, these areas will not be covered with green oil.To open the window.
- **Board outline**:The shape of the board is defined.Define the actual size of the board, the board factory will produce board according to this shape.In the GKO file generated in Gerber.
- **top assembly layer/bottom assembly layer**:The simplified outline of components is used for product assembly and maintenance.It is used to export document printing, which does not affect the production of PCB boards.
- **Mechanical**:The information recorded in the PCB design in the mechanical layer is only used for information recording.
    - The shape of this layer is made by default during production.
    - Some board factories use the mechanical layer as a border when producing by AD files.In Jialian EDA, the layer does not affect the frame shape of the board.
    - **If the mechanical layer has closed lines.When Jialian is produced, it will use the mechanical layer as the shape of the board. If there is no mechanical layer's frame, the GKO will be used as a border (the historical impact of AD files). Pay attention to the use of the mechanical layer when designing.**
- **Document layer**:Similar to the mechanical layer, it can be used as design -related information records to view.However, this layer is usually used in the editor, which is generated in the Gerber file and does not participate in manufacturing.
- **Ratline**:The display of the PCB network flying line is that this is not a layer in the physical sense. In order to facilitate the use and setting color, it is placed in a layer manager for configuration.
- **Hole**:Similar to the flying line layer, this layer that does not belong to the physical sense is only used for display and color configuration of the pores (non -metallic holes).
- **Multi**:Similar to the ratline layer, the display and color configuration of the metal hole.When the pad layer attribute is multi -layer, it will connect each copper foil layer to include the inner layer.
- **Component Shape**:The shape layer of the component is used in the shape of the component.Convenient and comparison with the size and physical size.

- **Component Marking**:The identifier layer of the component can add the special logo of the component, such as positive and negative poles, polar points, etc.
- **Pin Soldering**:The welding layer of the component's pins is convenient to compare the size of the packaging pad and the size of the physical pins.
- **Pin Floating**:The pins of the component are easy to compare with the size of the packaging pad and the size of the physical pads.
- **3D Shell Outline**:When drawing a 3D shell, the frame of the shell is located.
- **3D Shell Top/3D Shell Bottom**:The top or bottom layer of 3D shell.You can draw digging grooves, entities such as the entity.
- **Drill Drawing**:This is the information that stores the drilling table for manufacturing and production control.
- **Custom**:The custom layer is generally used for additional information recording, role and document layer, mechanical layer, and assembly layer.Do not use it directly for production.

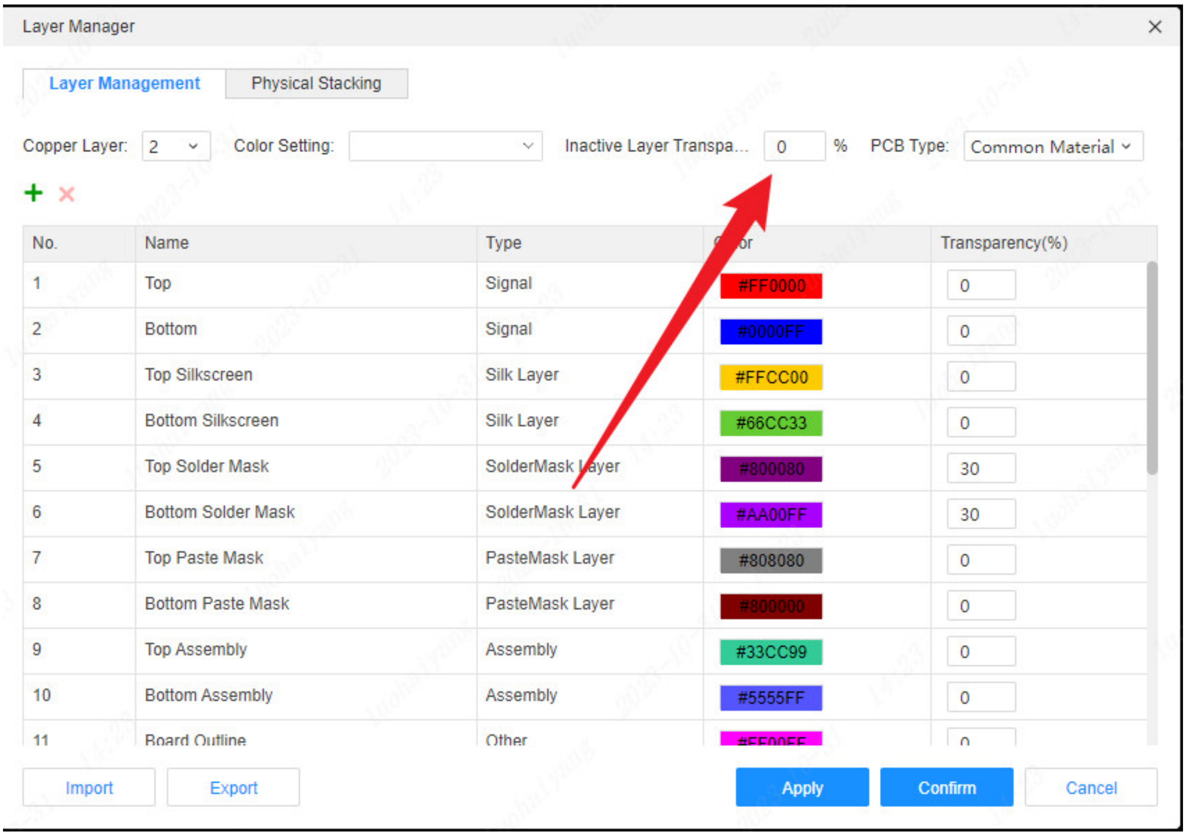
Color Setting

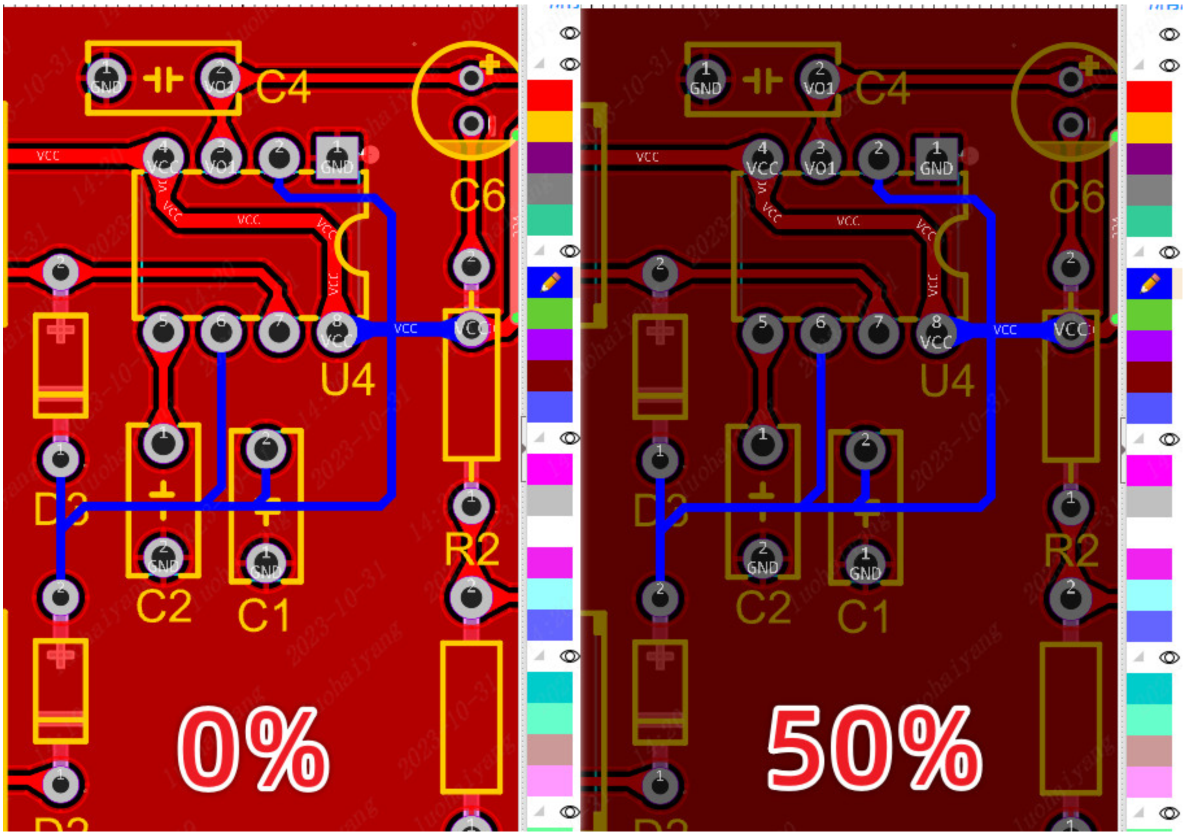
用户可以在此处选择一些预置的不同图层的颜色配置，以适应使用习惯，或者自定义后进行保存。



Inactive Layer Transparency

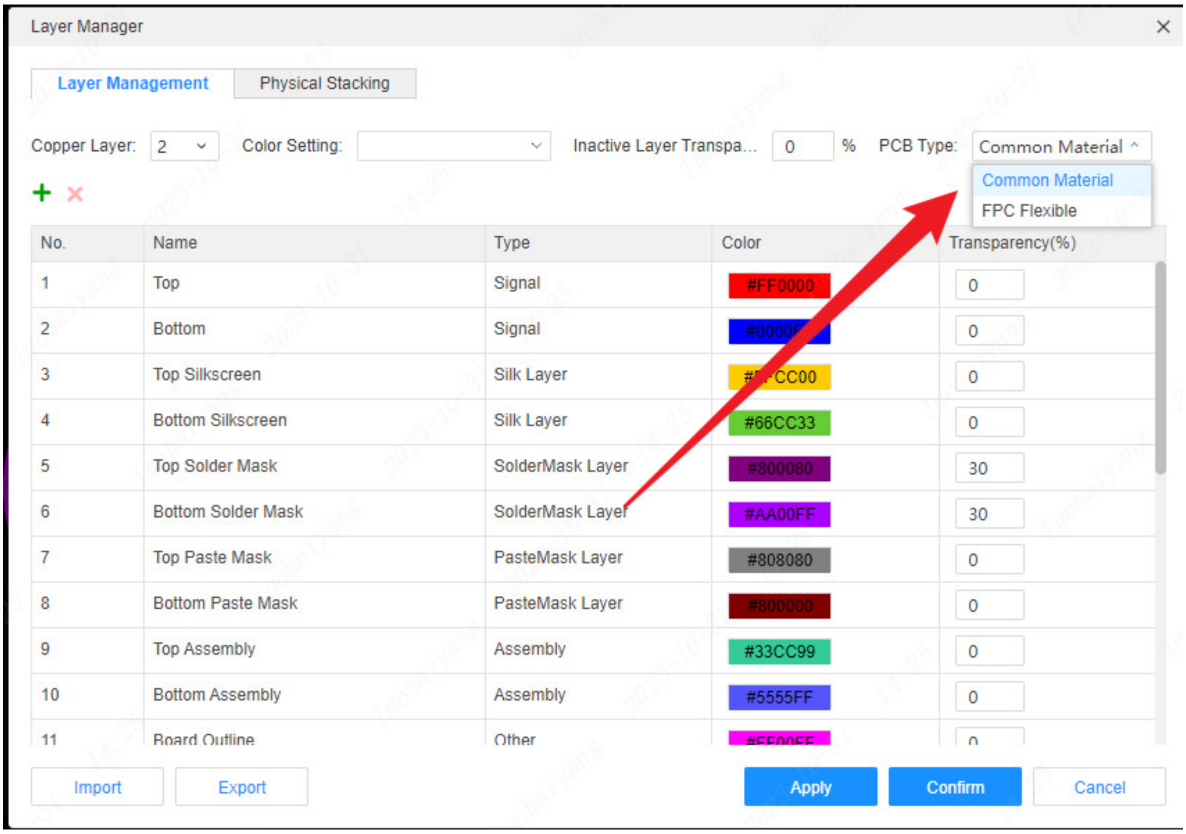
Here you can set the transparency of the contents of the inactive layer.Can improve the display effect





PCB Type

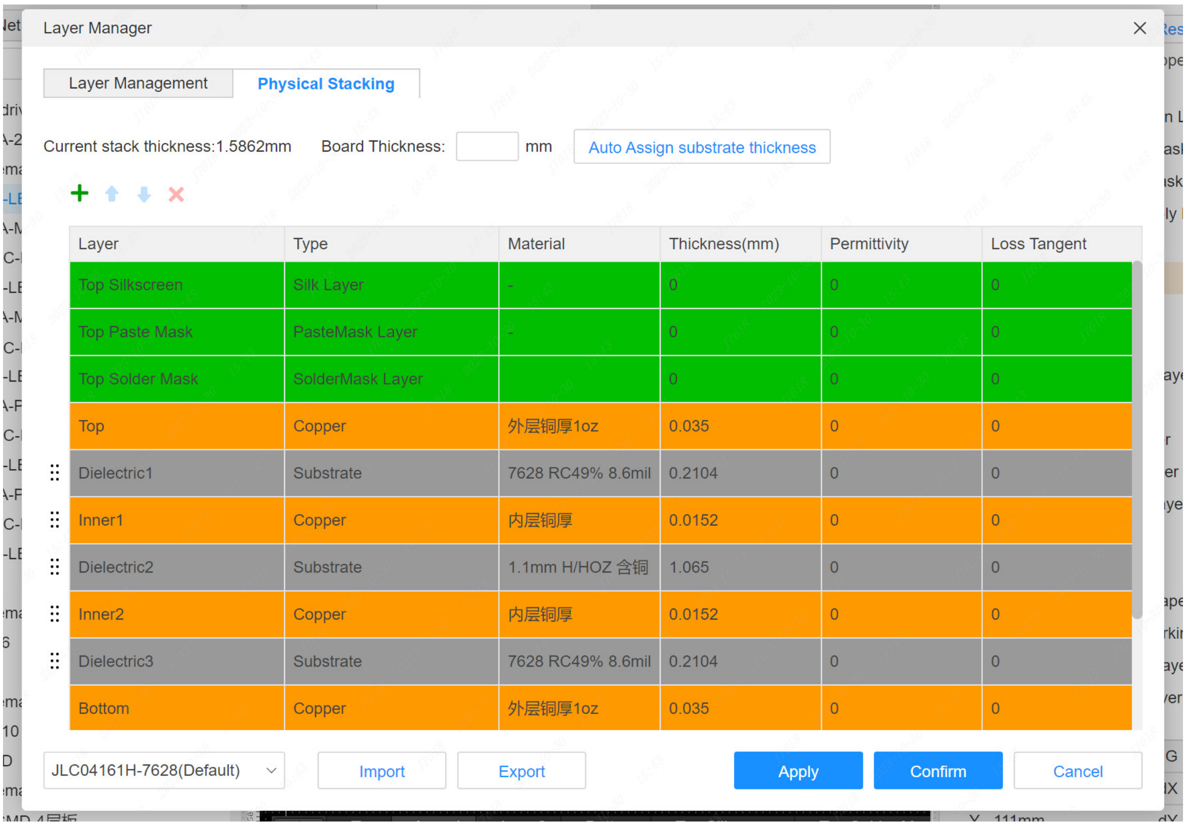
The type of PCB can be set here for ordinary boards or FPC soft boards. After setting to FPC soft board, the exclusive reinforcement layer of the FPC will be opened



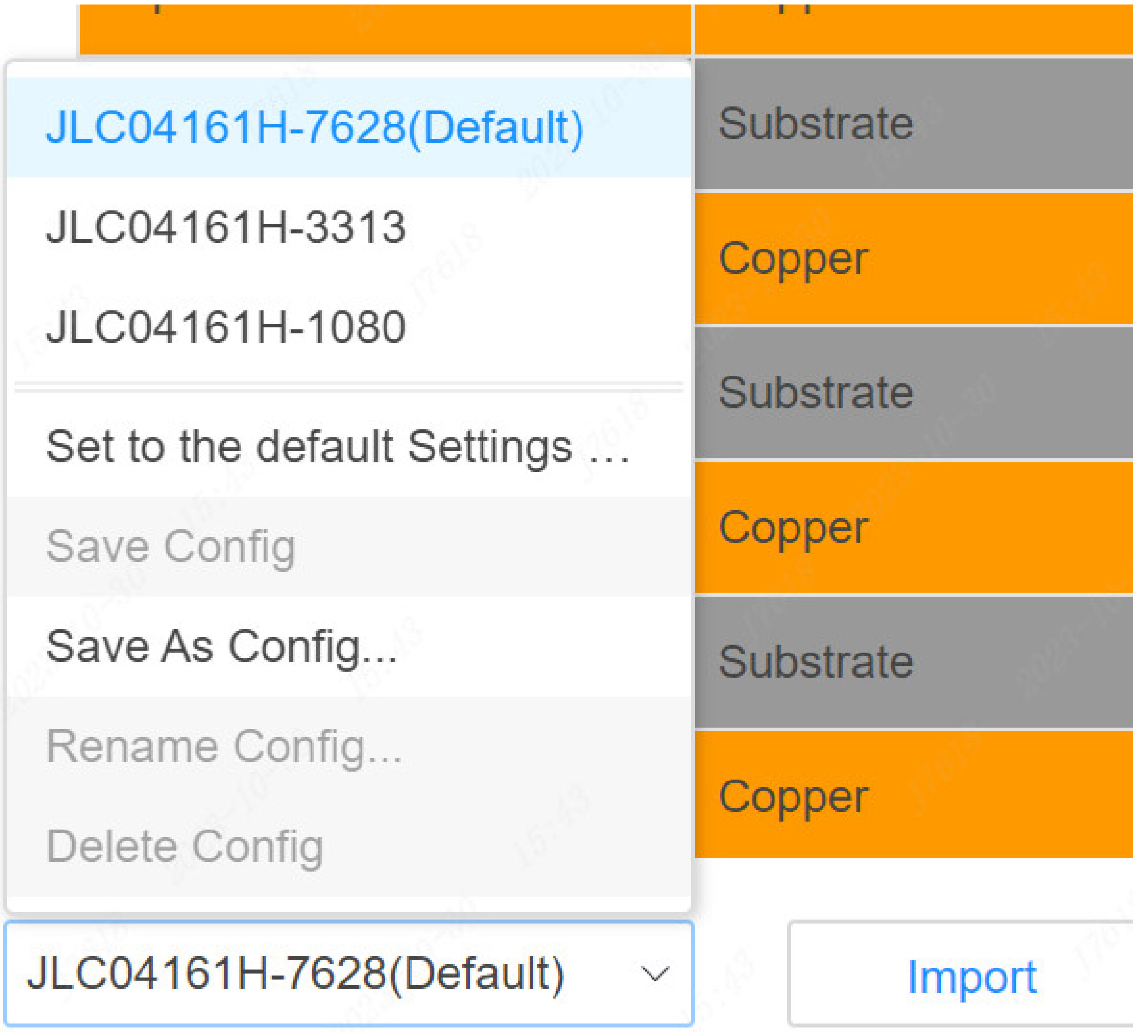
Physical Stacking

The physical stack parameters of the board can be set. At present, this parameter is only recorded, and the impedance calculation is not involved for the time being.The subsequent PCB wiring may use this parameter.

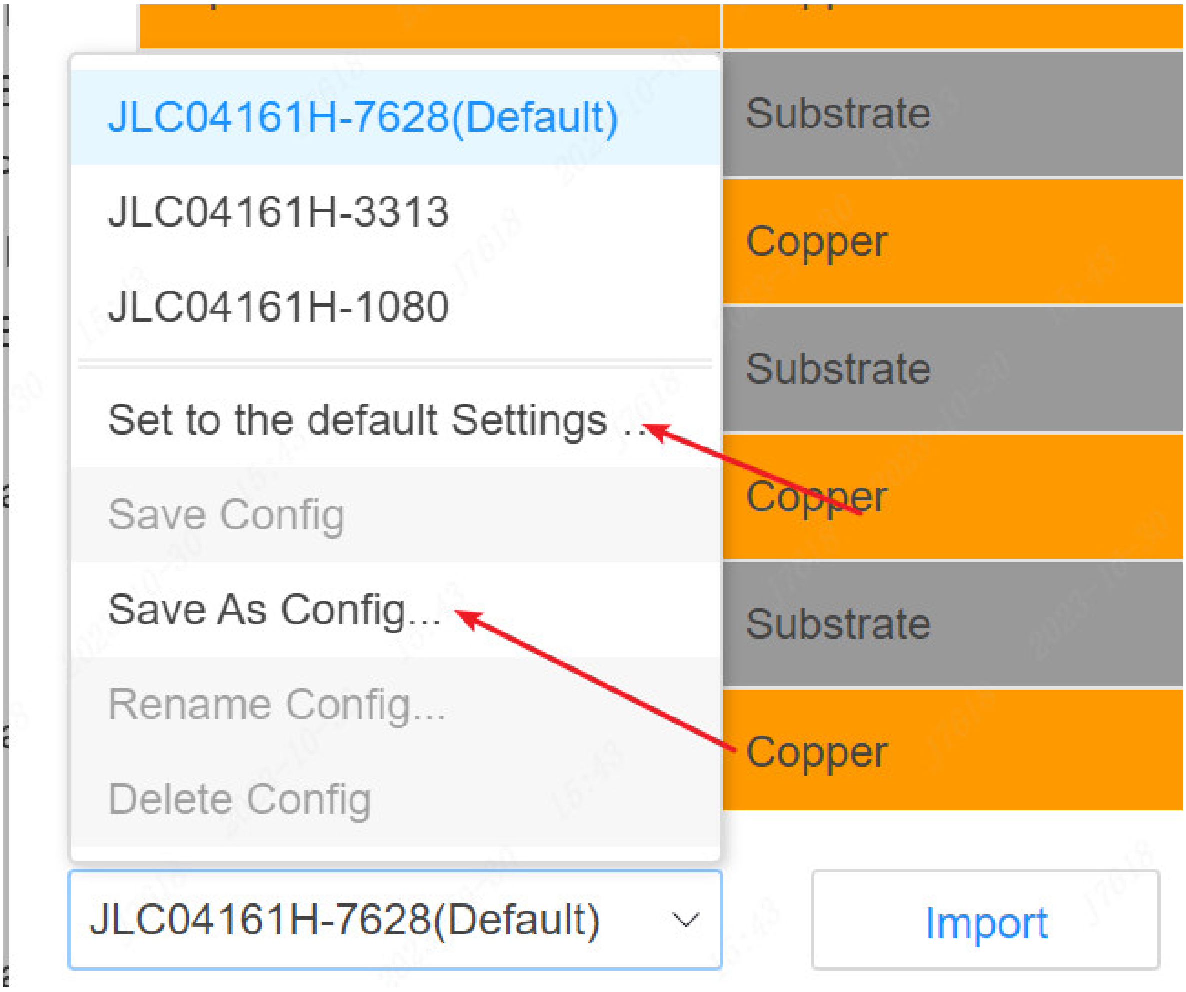
This physical stacking setting does not affect the export of Gerber. When placing an order, you need to re -select the stack parameters for ordering, but the setting can affect the thickness of the substrate medium of the 3D preview, the exported 3D file and ODB++ file.



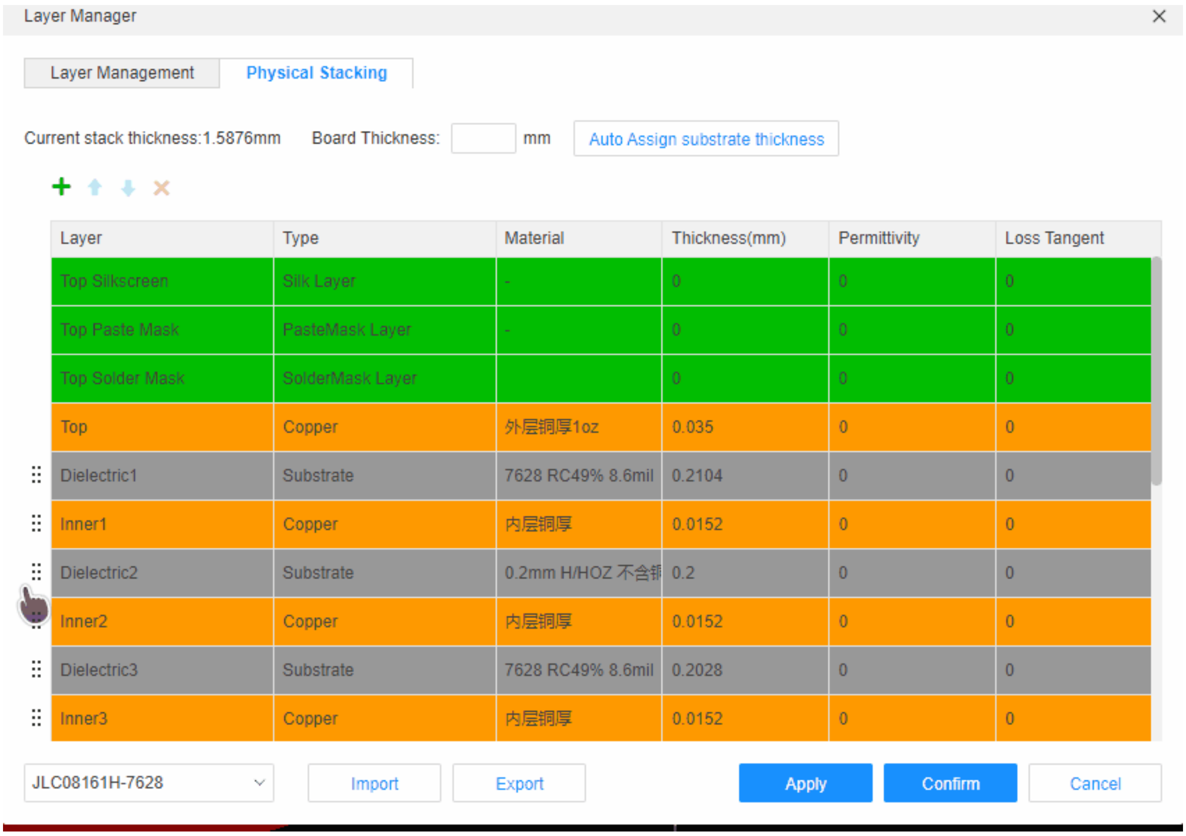
Click the drop -down menu in the lower left corner to switch the layer pressure configuration of different layers. The system provides a commonly used layer under the default of several commonly used layers.



You can also set your own common layer pressure configuration under different layers of boards, and the default configuration when setting a new PCB.



You can also drag the order of the inner layer



Drawing a single layer PCB

The copper foil layers of EasyEDA Pro are double numbers. It does not support direct drawing of a single-layer PCB. You can achieve the purpose of drawing a single-layer by two methods:

method 1:

- Place and route directly on a single layer (top or bottom) without placing vias.
- If you have a footprint with multiple pads, there will be copper on the top and bottom layers. At this point, you can find out the footprints corresponding to all the multi-layer pads, right-click to edit the footprint - apply the current PCB, change the metallization (copper plating) property of the multi-layer pads to No, and then change the size of the multi-layer pads Set to "Custom" and set the diameter of the top or bottom to 0.
- After generating the Gerber, check whether the Gerber meets the design requirements.

Method 2:

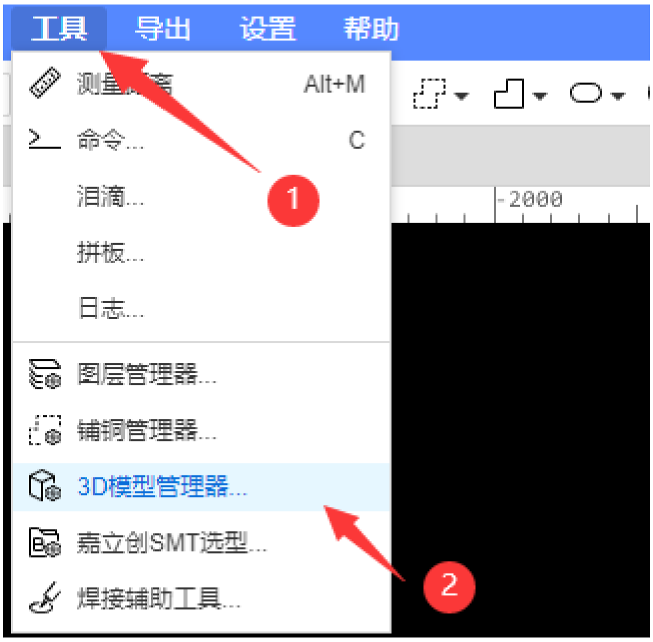
- The PCB design is completed on one side, and the pads on the other side do not need to be processed to generate Gerber.
- Communicate well with the board factory when placing an order for PCB with Gerber, and make a note to only make a single-layer board.

3D Model Manager

The 3D model manager is similar to the footprint manager, the symbol manager supports single and batch addition of 3D models and adjustments

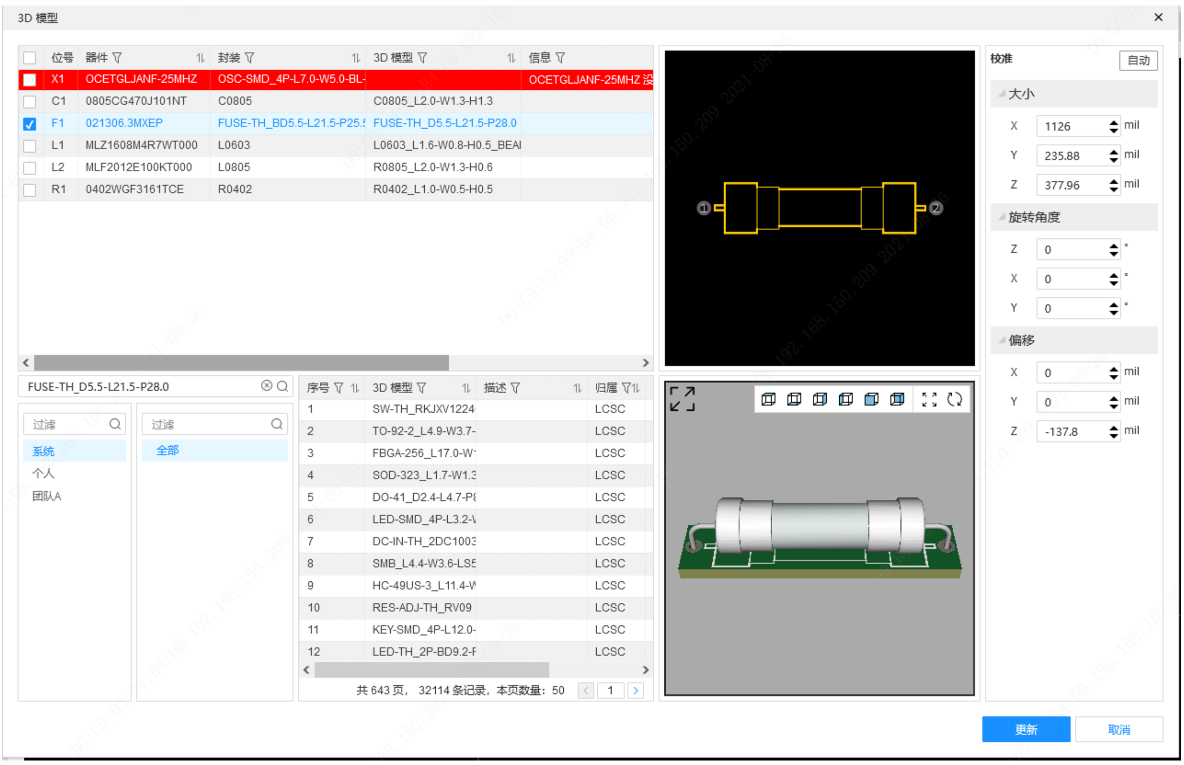
Operation entry:

- Top Menu - Tools - 3D Model Manager

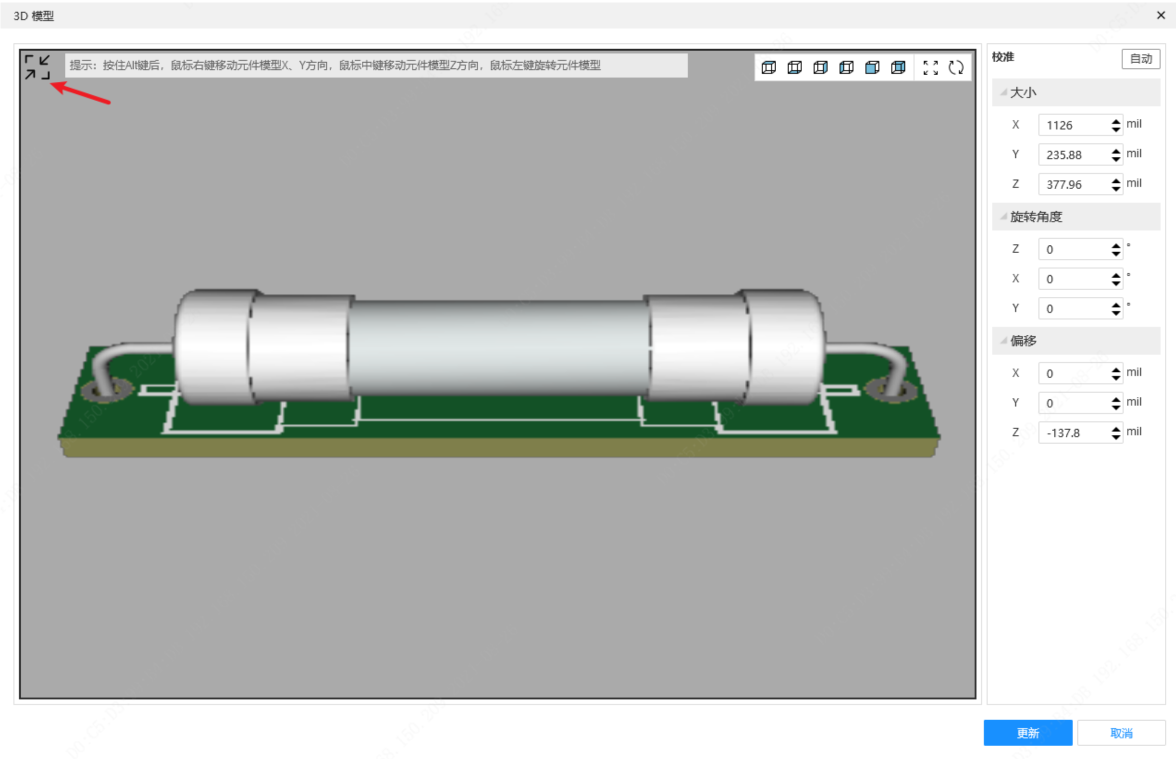


Or select a component, in the property panel on the right, click Properties **3D Model** to open it.

3D Model Manager interface:



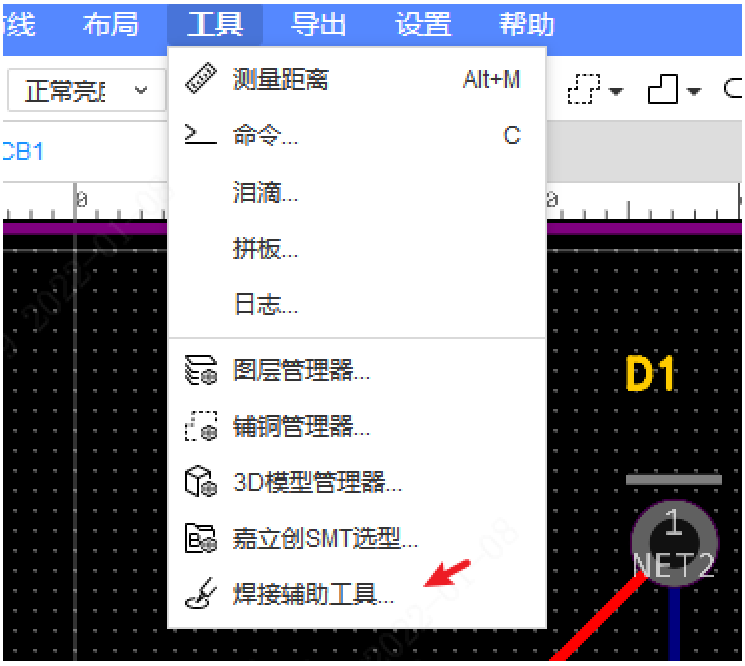
- **Device List**: The device list displays all the devices of the current PCB, and the selected device will be bound when binding the 3D model.
- **Search and selection list**: The library list in the lower left corner makes it easy to search for 3D models and click to select by category.
- **Preview area**: Preview the footprint above, and preview the footprint and 3D model below. Both support scrolling and zooming, right-click and long-press to move. The 3D model preview area can also be enlarged for preview and operation, and supports adjustment after holding down the shortcut key ALT.
- Calibration area: Control the relative position and angle of the 3D model and footprint by adjusting parameters. The Auto button is to automatically adjust the size and position of the model according to the size of the top view of the 3D model and the size of the footprint of the footprint. After clicking Auto, it still needs to be fine-tuned manually.

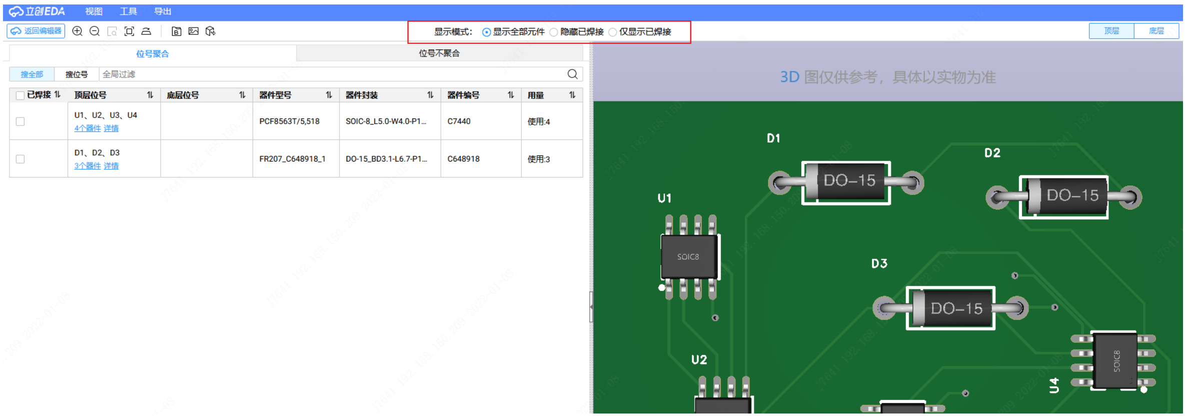


Soldering Assitance Tool

EasyEDA Pro provides a simple soldering aid to facilitate component positioning while soldering.

Entry: Top Menu - Tools - Soldering Assitance Tool





Check the welded part in the component list on the left, and support component tag aggregation or non-aggregation. When you move the mouse over the component list, the 3D preview on the right will highlight the model of the component at the same time, which is convenient for positioning.

The top supports displaying all components by category, or displaying components that need to be soldered, etc.

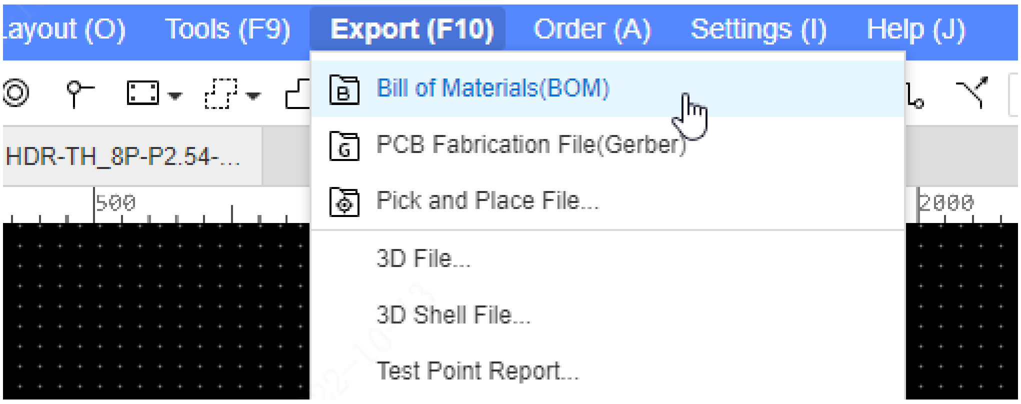
Export BOM

###Export BOM

EasyEDA Pro supports separate PCB export BOM (Bill of Materials) so that you can purchase the required parts.

Using Zoom:

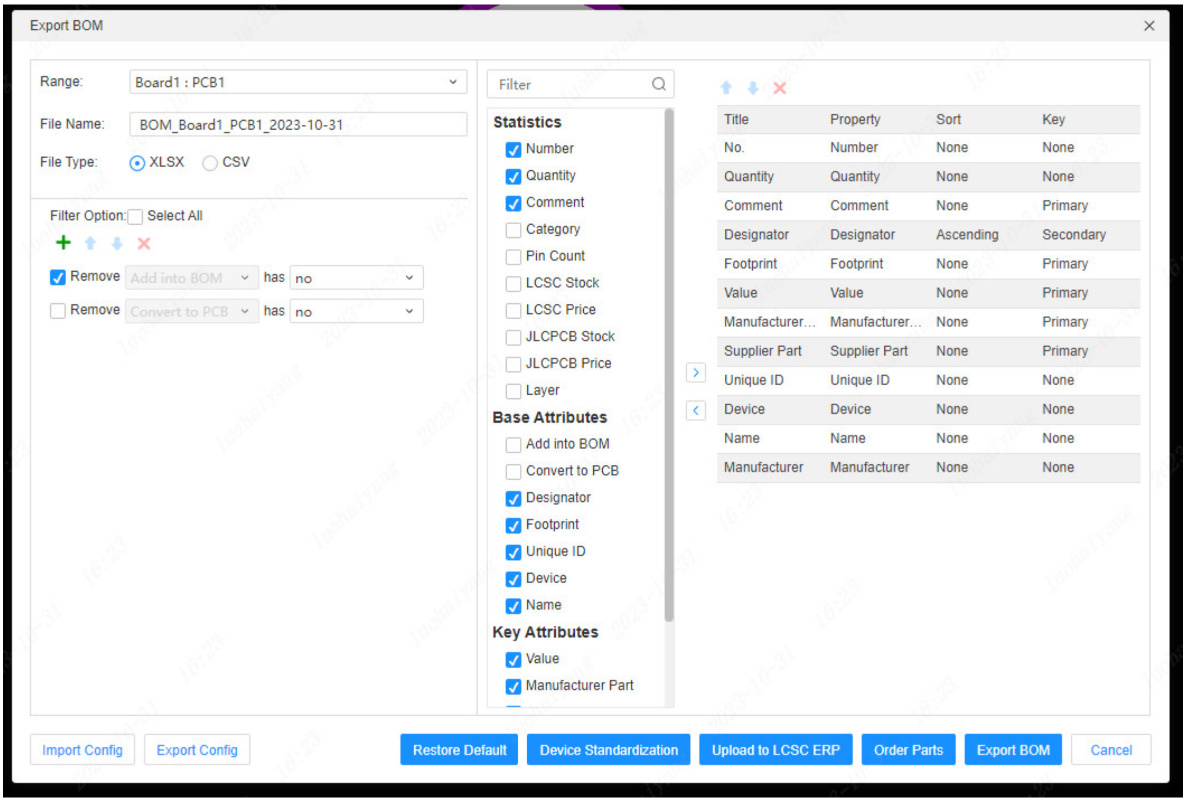
- Top Menu - File - Export - Bill of Materials BOM
- Top Menu - Export - Bill of Materials BOM



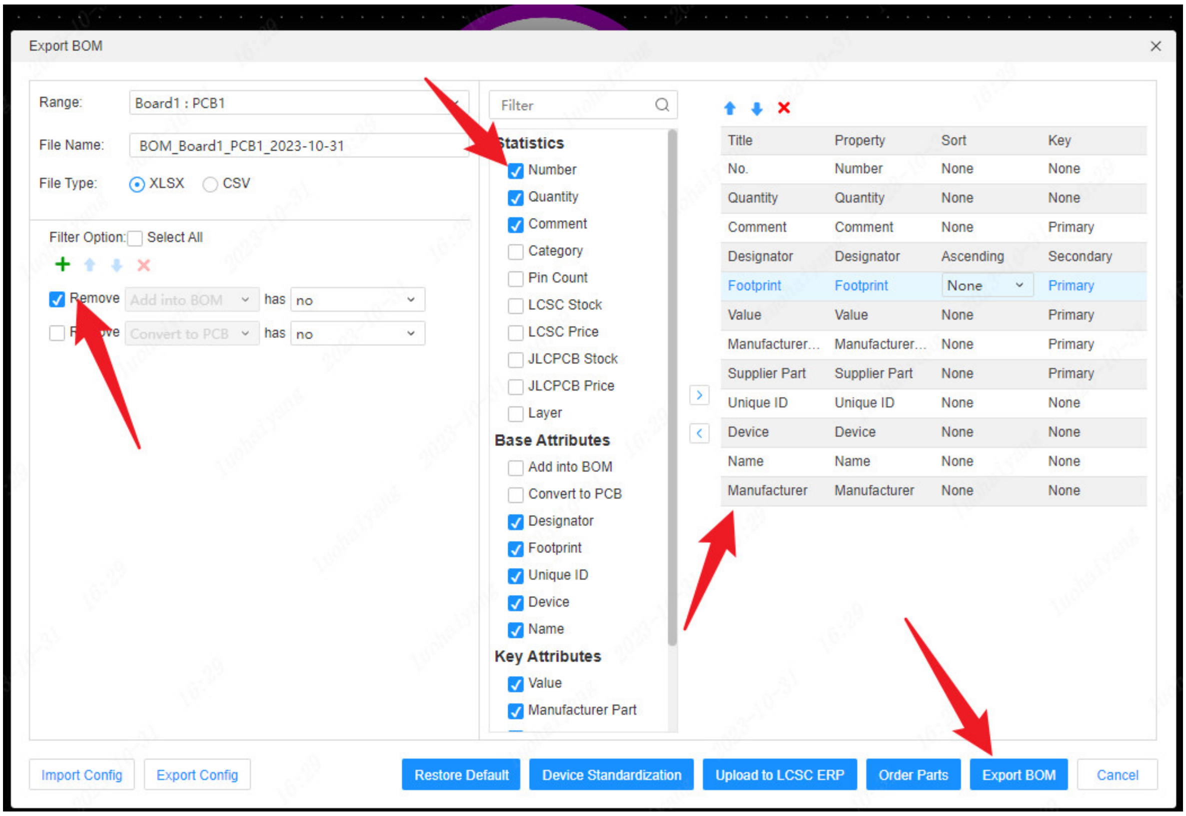
Scope: You can choose to export the PCB of the project or the BOM of the schematic diagram;

File name: The file name of the exported BOM;

File Type: Only XLSX and CSV formats are supported;

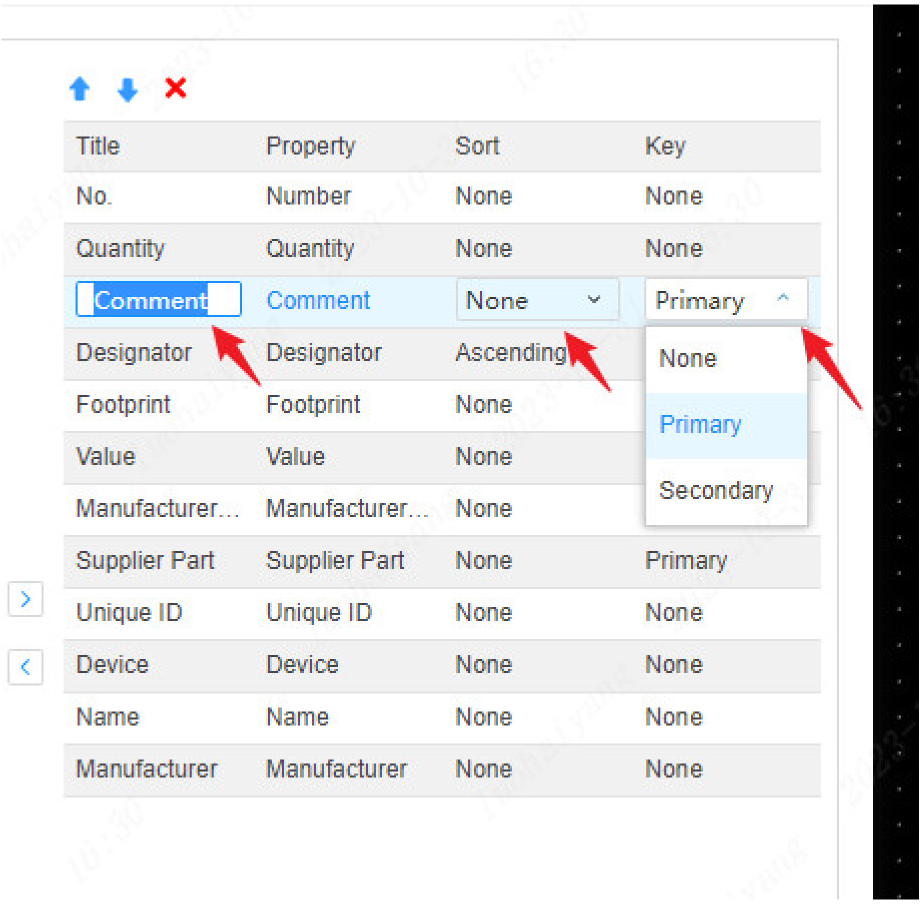


you can set the filter option and select the export properties



To add a type of sorting, you need to double-click the content to be changed on the right, or select the type and click the up and down arrows at the top to change the order.

- **Title**: The title of the exported BOM;
- **Property**: the relevant attribute type of the device;
- **Sort**: export the sorting order of BOM properties;



Primary key: When exporting the same attribute to the BOM table, the values are arranged separately;

Cluster key: When exporting the same attribute to the BOM table, the values are merged and arranged;

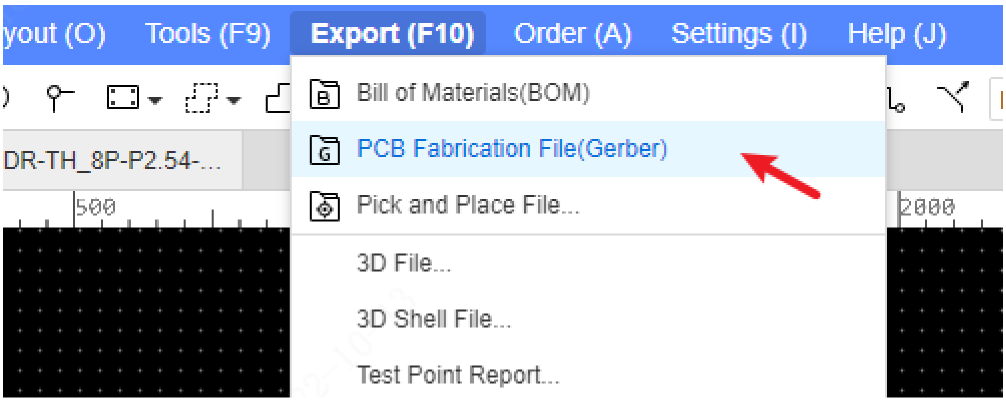
Exported bom table

No.	Device	Designator	Footprint
1	0.5a/6v	F1	0805_L_JX_1
2	100nf/50v	C2, C4, C5, C6, C7, C8, C9	0603_C_JX_13
3	bl1117-3.3_jx	U1	SOT223_JX_1
4	mini-usb_jx	P1, P2	USB_Mini_B_Fe_JX_2
5	1.5k/1%	R1	0603_R_JX_11
6	red/led	LED0	0603_D_JX_3
7	4.7k/1%	R2, R3, R4	0603_R_JX_11
8	green/led	LED1	0603_D_JX_3
9	blue/led	LED2	0603_D_JX_3
10	0r/1%	R5	0603_C_JX_13
11	114148	D1, D2	LL-34_JX_2
12	0r/1%	R6	0603_R_JX_11
13	8mhz/20pf	Y1	HC-49S_JX_1
14	10pf/50v	C14, C13	0603_C_JX_13
15	32.768khz/6pf	Y2	SMD-3215_2P_JX_1
16	k2-1101ut-b4sw-01_jx	WKUP1, KEY1, KEY2, RESE	SW_SMD_3*6_JX_3

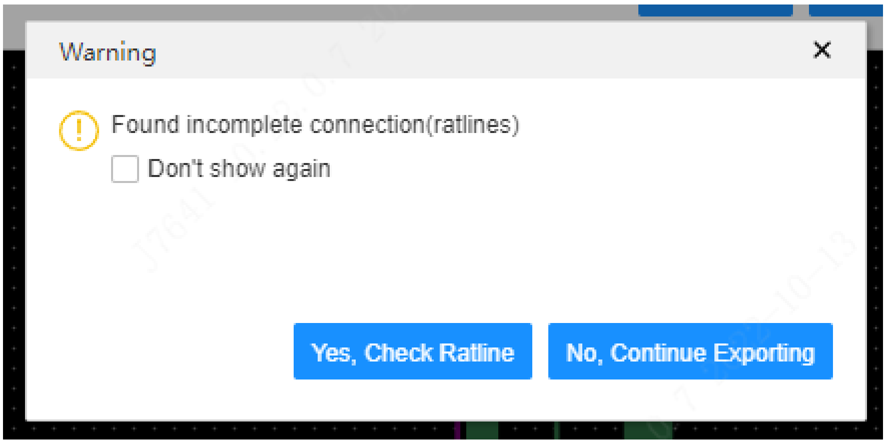
Export PCB Fabrication File(Gerber)

Export method:

- Entry: Top Menu - File - Export - PCB Fabrication File (Gerber)
- or: Top Menu - Export - PCB Fabrication File (Gerber)



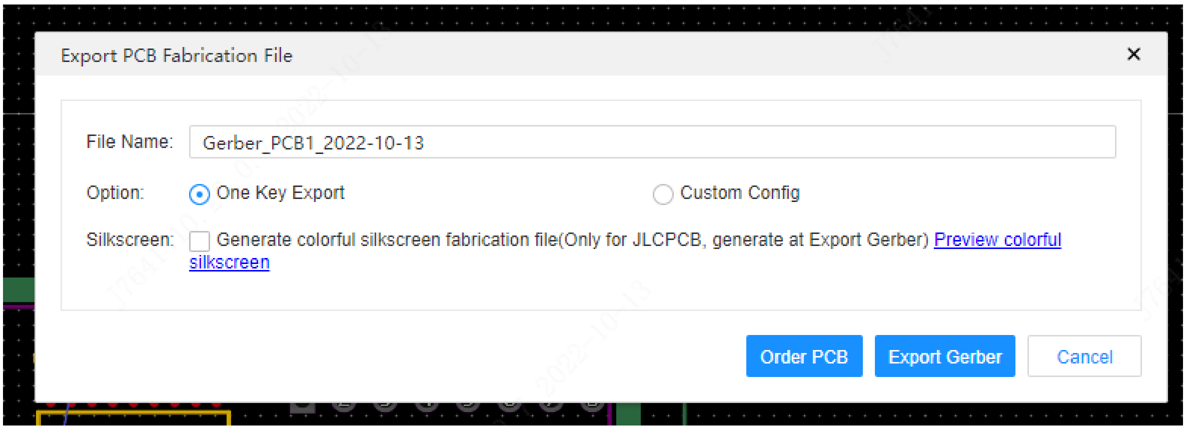
After clicking, the system will detect whether there are flying wires in the PCB. If so, the following pop-up window will pop up.



Click "Yes" to check the flying leads. It will automatically locate the first flying line, and switch to the flying line tree on the left panel to help you check. If you click "No" to continue exporting, the gerber file will be exported directly ignoring the existing flying lines.

If you don't want to automatically detect flying leads every time you export gerber, you can turn off this configuration in system settings.

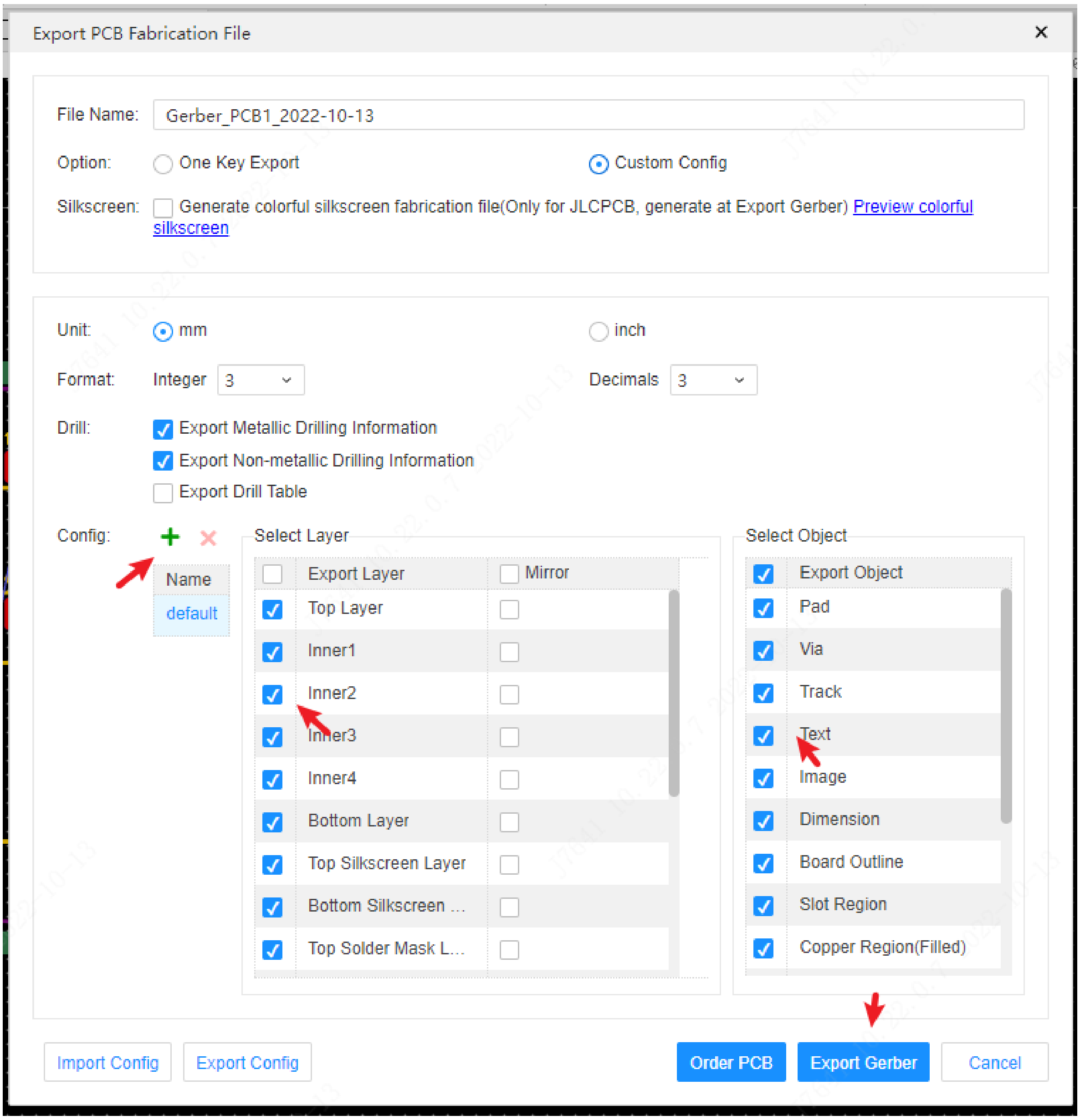
Export the Gerber popup:



File name: Support to modify the file name and then export.

One-click export: According to the default settings, all layers and primitives are exported, excluding the drilling table and independent drilling information files.

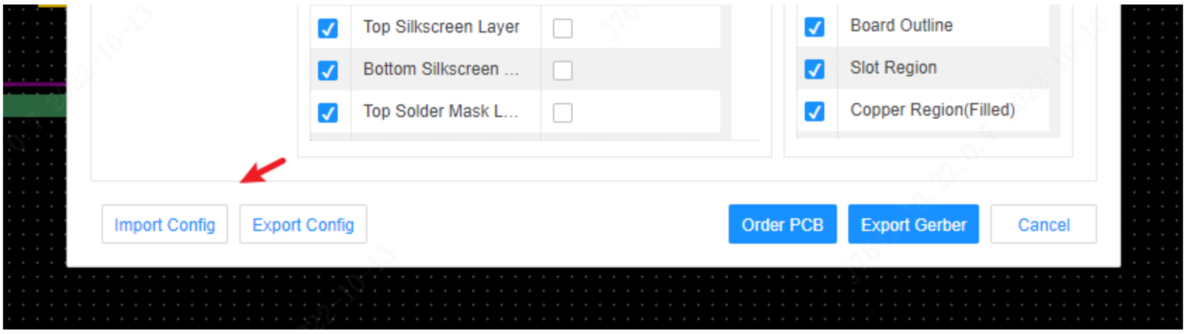
Custom configuration: Modify the configuration according to your own needs. Support drilling information and drilling table; support adding different configurations on the left list; support selecting exported layers; layer mirroring; support selecting exported primitives. When exporting, select a configuration for Gerber export. Up to 20 configurations can be created, double-click to modify the configuration name.



Unit: The unit of the exported Gerber file and drilling file, the default is mm.

Format: The numerical format of the exported drilling file, the number of integer digits and decimal digits, affects the expression of numerical precision. This setting may affect the alignment of the drill file viewed by the Gerber Viewer. If the Gerber viewer previews the Gerber and drilling files and finds that the drilling files are inaccurate, you can reset the numerical format of the drilling files in the Gerber viewing tool to 3:3, or other formats.

Export/Import Configuration: Supports importing and exporting Gerber custom configurations, which is convenient for configuration reuse. Configuration information is stored in personal preferences for cloud synchronization.



Gerber Instructions

Gerber compressed package: After exporting Gerber, it is a zip compressed package, which can be directly uploaded when the board factory places an order. Those with editing needs (such as CAM engineers) can use third-party CAM tools to edit Gerber after decompression.
Description of file names in Gerber:

Gerber filename:

The generated Gerber file is a compressed package. After decompression, you can see the following files:

File Name	Type	Remarks/Description
Gerber_BoardOutline.GKO	Outline file	The PCB board factory cuts the shape of the board according to this file, and the digging area drawn by EasyEDA is reflected in the outline file after Gerber is generated
Gerber_TopLayer.GTL	PCB Top Layer	Top Copper Foil Layer
Gerber_BottomLayer.GBL	PCB bottom layer	Bottom copper foil layer
Gerber_InnerLayer1.G1	Inner Copper Foil Layer	Signal Layer Type
Gerber_InnerLayer2.GP2	Inner copper foil layer	Inner electrical layer type
Gerber_TopSilkLayer.GTO	Top silkscreen layer	
Gerber_BottomSilkLayer.GBO*	Bottom silkscreen layer	
Gerber_TopSolderMaskLayer.GTS	Top Solder Mask Layer	It can also be called the copper expose layer. By default, the board is covered with oil, and the elements drawn on this layer correspond to the area of the top layer, which is not covered with oil
Gerber_BottomSolderMaskLayer.GBS	Bottom Solder Mask	It can also be called the copper expose layer. The board is covered with oil by default, and the elements drawn on this layer correspond to the area of the bottom layer without oil
Gerber_TopPasteMaskLayer.GTP	Top layer flux layer	For stencil fabriaction
Gerber_BottomPasteMaskLayer.GBP	Bottom layer of soldering flux	For stencil fabriaction

File Name	Type	Remarks/Description
Gerber_TopAssemblyLayer.GTA	Top assembly layer	Only for reading, does not affect PCB manufacturing
Gerber_BottomAssemblyLayer.GBA	Bottom assembly layer	Only for reading, does not affect PCB manufacturing
Gerber_MechanicalLayer.GME	Mechanical Layer	The information recorded in the mechanical layer in the PCB design is only used for information recording. The shape of this layer is not used for manufacturing by default during production. This layer is only used for text identification. For example: process parameters, V-cut path, etc.
Gerber_DocumentLayer.GDL	Document Layer	Used to record PCB remarks, not involved in manufacturing
Gerber_CustomLayer1.GCL	User-defined layer	User-defined layer generally does not belong to the layer required for generation. If you need to use it in production, you can communicate with the board factory
Gerber_DrillDrawingLayer.GDD	Drilling layer	This layer does not participate in manufacturing, it is used for comparison and identification of the position of the generated via hole
Gerber_TopStiffenerLayer_xx_xx.GTSL	Top Stiffener Layer	This file is stiffener object layer, only for JLCPCB use
Gerber_BottomStiffenerLayer_xx_xx.GBSL	Bottom Stiffener Layer	This file is stiffener object layer, only for JLCPCB use
Drill_PTH_Through.DRL	Drilling layer for metallized multilayer pads	This file shows the drill hole position that needs metallization on the inner wall, such as multi-layer pads and through-hole vias
Drill_PTH_Through_Via.DRL	Drilling layer of metallized through-hole type via	This file shows the drill hole position that needs metallization on the inner wall. This file use at JLCPCB

File Name	Type	Remarks/Description
Drill_PTH_Inner1_to_Inner2.DRL	Drilling Layer for Metallized Blind and Buried Via	This file shows the position of the drill holes that need metallization on the inner wall. Inner1 and Inner2 are automatically changed according to the layer type of the blind and buried via
Drill_NPTH_Through.DRL	Non-metallized drilling layer	This file shows the drilling position where the inner wall does not need metallization, such as through holes (circular trenching areas)
Fabrication_ColorfulTopSilkscreen.FCTS	Top layer color silkscreen file	This file only exists when the color silkscreen process is checked when exporting Gerber
Fabrication_ColorfulBottomSilkscreen.FCBS	Bottom color silkscreen file	This file only exists when the color silkscreen process is checked when exporting Gerber
jlcpcb.json	Gerber config file	Only for JLCPCB use, it save some info for ordering, such as stiffener location etc.

Notice:

- *Before generating a manufacturing file, be sure to do a 2D or 3D preview to check the Design Manager's DRC error entry to avoid generating defective Gerber files. *
- *The generated Gerber is generated by the browser, so it must be downloaded through the browser's own download function, and cannot use any third-party downloader*
- *The coordinates of the Gerber file follow the canvas coordinates*
- *When exporting Gerber, the default accuracy of the drill file coordinate format is 3:3, and when the size exceeds the range, the 4:2 format is automatically used. If you find the drill hole offset in the viewing tool such as CAM350, please adjust the drill hole coordinate format. . You can also choose to customize the output when exporting, and set the format precision. *

Gerber Preview

Before sending the Gerber file to the manufacturer, use the Gerber Viewer to double-check that the Gerber meets the design requirements and has no design flaws.

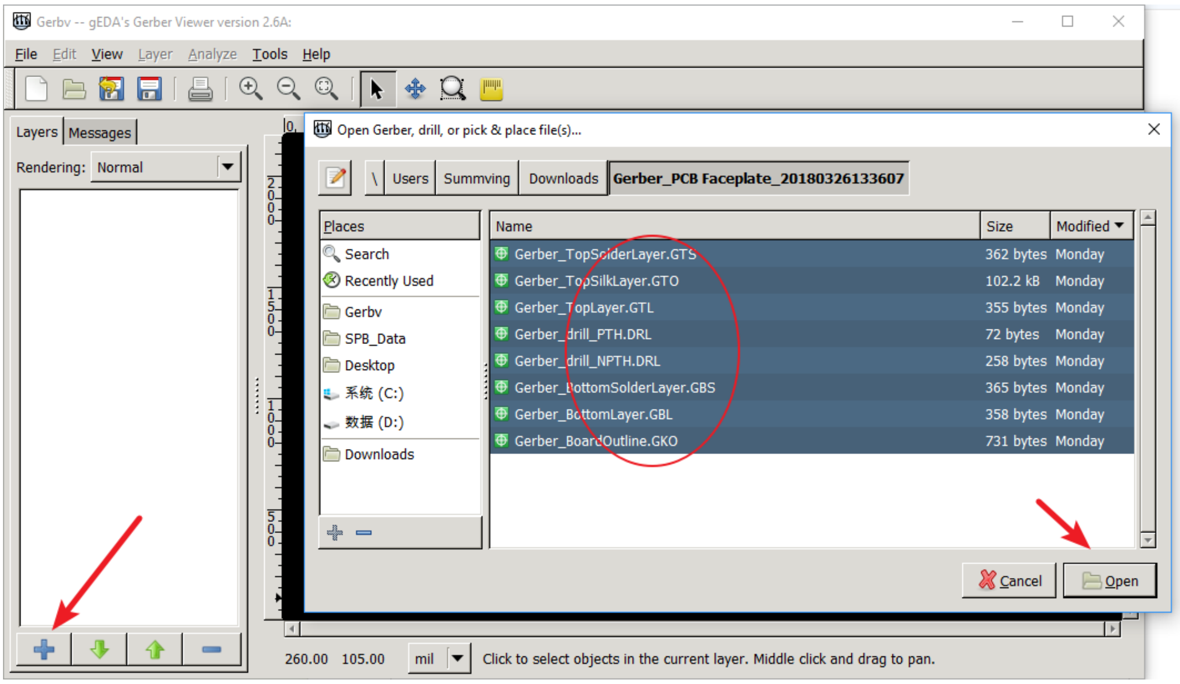
Gerber viewers include: Gerbv, FlatCAM, CAM350, ViewMate, GerberLogix and some DFM inspection tools.

Recommended free Gerbv:

Official website homepage: http://gerbv.geda-project.org/
Download address: https://sourceforge.net/projects/gerbv/files/
Download address 2: Gerbv-2.6.0.exe

How to use Gerbv:

1. Download Gerbv, and open it; unzip the downloaded Gerber archive.
2. Click the plus sign + in the lower left corner to open the Gerber folder, and **SHIFT+select all** or **CTRL+A select all** the unzipped Gerber file.



3. Then zoom, measure, change layers, check whether drilling, copper plating, etc. meet the design and production requirements.

Other free Gerber preview tools:

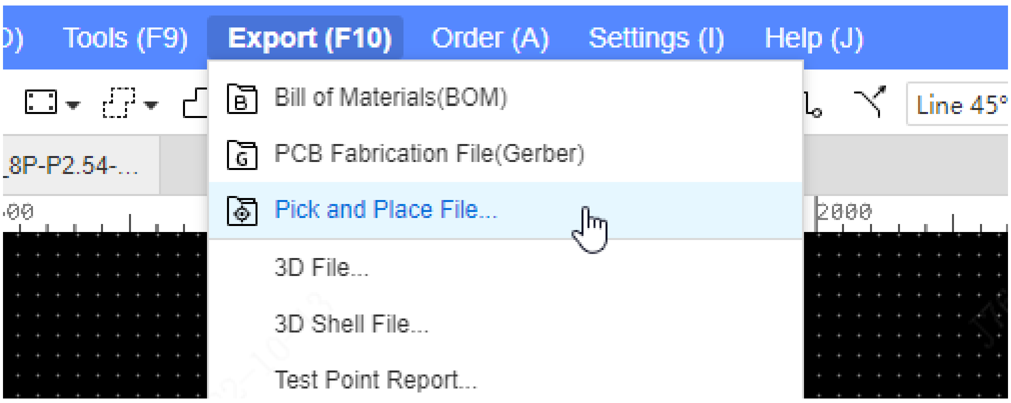
online use: [jlcpcb.com(after import gerber)]
online use: [tracespace.io/view]
online use: [gerber.ucamco.com]
desktop client: [http://flatcam.org/]

export coordinate file

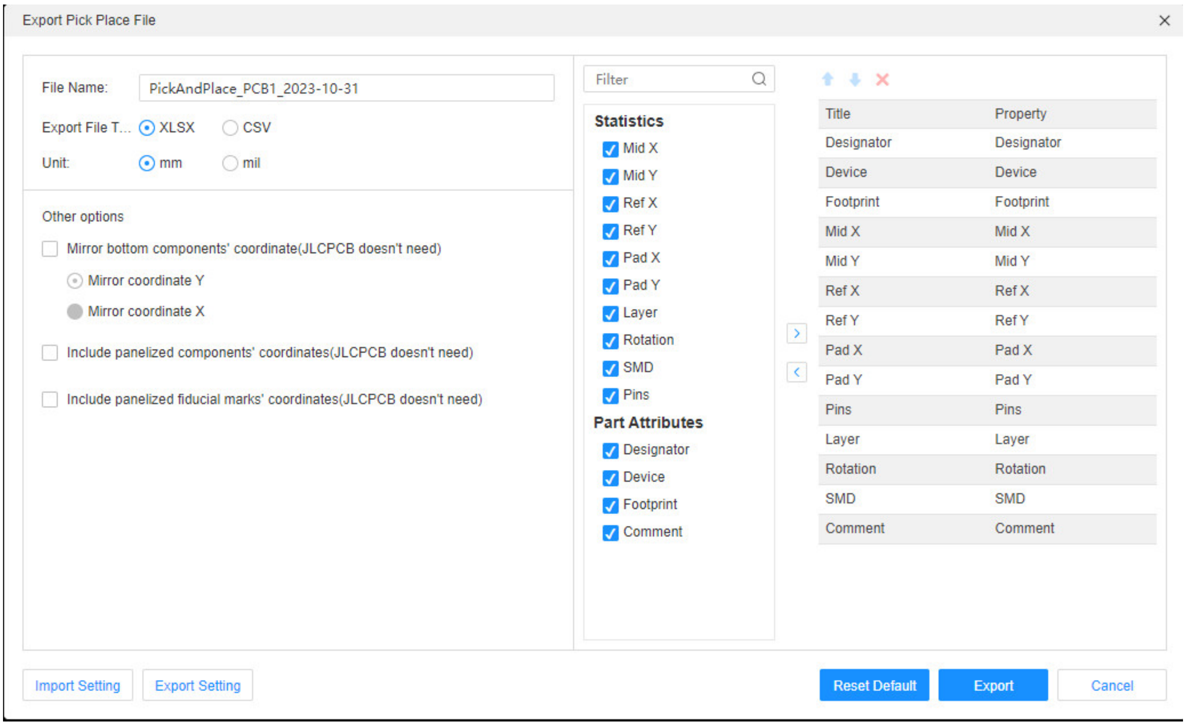
EasyEDA Pro supports exporting SMT coordinate information to facilitate SMT Place in factories. Coordinate files can only be exported in PCB.

Export method:

- Top Menu - File - Export - Pick and Place File



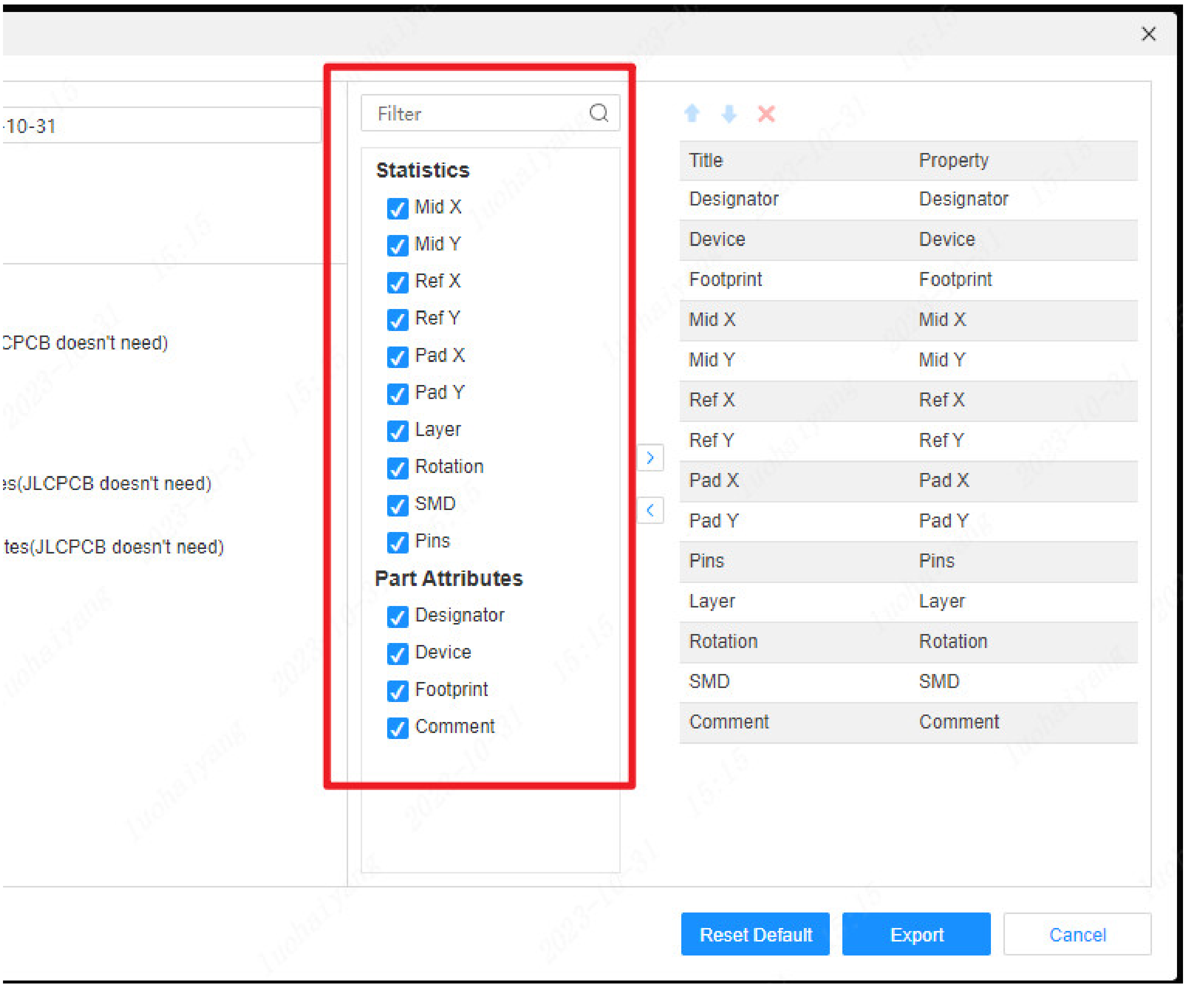
Options:



- Filename: supports custom exported filename
- Export file type: support XLSX and CSV two types of files
- Unit: Can be set to mm or mil
- Mirror the coordinates of the underlying components: Some patch manufacturers need the coordinates of the underlying components after mirror, you can check this option, generally do not need to check. In JLC proofing do not need to tick
- The component coordinates of the palette after the collage (not required by JLC) : output all the coordinates of the split
- Include the coordinates of the Mark point after the puzzle board (not required by JLC) : Check and output the coordinates of the Mark point after the puzzle board

statistics

The Statistics Office can select the content that needs to be exported,You can adjust the order of the output in the table on the right



The exported coordinate file looks like this:

	A	B	C	D	E	F	G	H	I	J	K	L
1	Designator	Comment	Footprint	Mid X	Mid Y	Ref X	Ref Y	Pad X	Pad Y	Layer	Rotation	SMD
2	R1	1k	R0805	43.992mm	42.672mm	43.992mm	42.672mm	43.026mm	42.672mm	T	0	Yes
3	R2	1k_R0805	R0805	31.8mm	45.466mm	31.8mm	45.466mm	30.834mm	45.466mm	T	0	Yes
4	R5	1k_R0805	R0805	42.672mm	34.798mm	42.672mm	34.798mm	43.638mm	34.798mm	B	180	Yes
5	R6	1k_R0805	R0805	25.096mm	36.322mm	25.096mm	36.322mm	24.13mm	36.322mm	T	0	Yes
6	C1	C_Ele_SMD_3x5.4mm	CAP-SMD_BD3.0-L3.3-W3.3-RD	38.1mm	21.844mm	38.1mm	21.844mm	39.469mm	21.844mm	T	0	Yes
7	J2	C_Ele_SMD_3x5.4mm	CAP-SMD_BD3.0-L3.3-W3.3-RD	47.752mm	18.288mm	47.752mm	18.288mm	49.121mm	18.288mm	T	0	Yes
8	C3	C_Ele_SMD_3x5.4mm	CAP-SMD_BD3.0-L3.3-W3.3-RD	55.803mm	16.48mm	55.803mm	16.48mm	57.172mm	16.48mm	T	0	Yes
9	Q1	S8050_NPN	SOT-23_L2.9-W1.3-P1.90-LS2.4-BR	32.766mm	7.366mm	32.766mm	7.366mm	33.916mm	6.416mm	T	0	Yes
10	USB1	USB_ Type-C-6P	USB-SMD_U262-061N-4BVC11	12.446mm	22.352mm	12.446mm	22.352mm	10.307mm	18.032mm	T	270	No
11	U2	CJA1117B-5.0	SOT-89-3_L4.5-W2.5-P1.50-LS4.2-BR	27.01mm	27.94mm	27.01mm	27.94mm	28.366mm	26.44mm	T	0	Yes
12												

Table description:

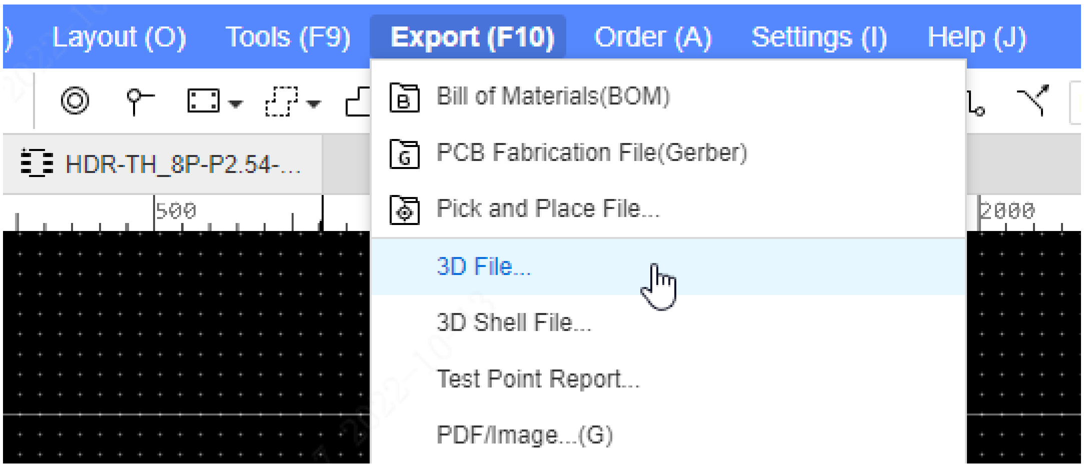
- Designator: The component's designator
- Comment: The name of the device, usually the manufacturer's number of the component.
- Footprint: The component's footprint.
- Mid X, Mid Y: The center coordinate of the footprint.
- Ref X, Ref Y: The origin of the footprint.
- Pad X, Pad Y: The first pad coordinates of the footprint.
- Layer: The footprint layer.
- Rotation: The Rotation angle of the footprint.
- SMD: The footprint is SMD or not.

Export 3D file

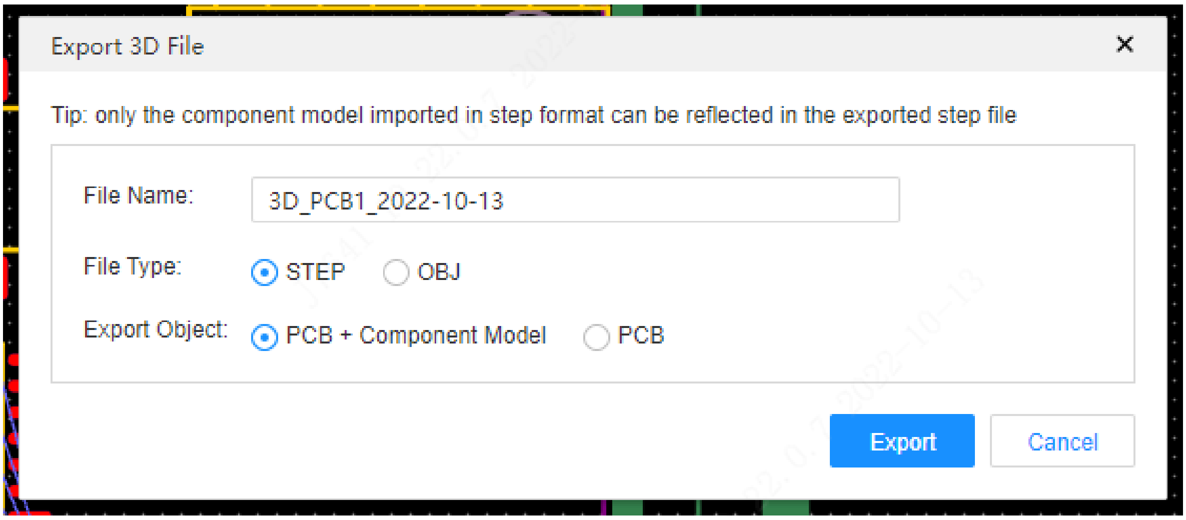
EasyEDA Pro currently only supports exporting 3D files in two formats.

Export method:

- Top Menu - Export - 3D file

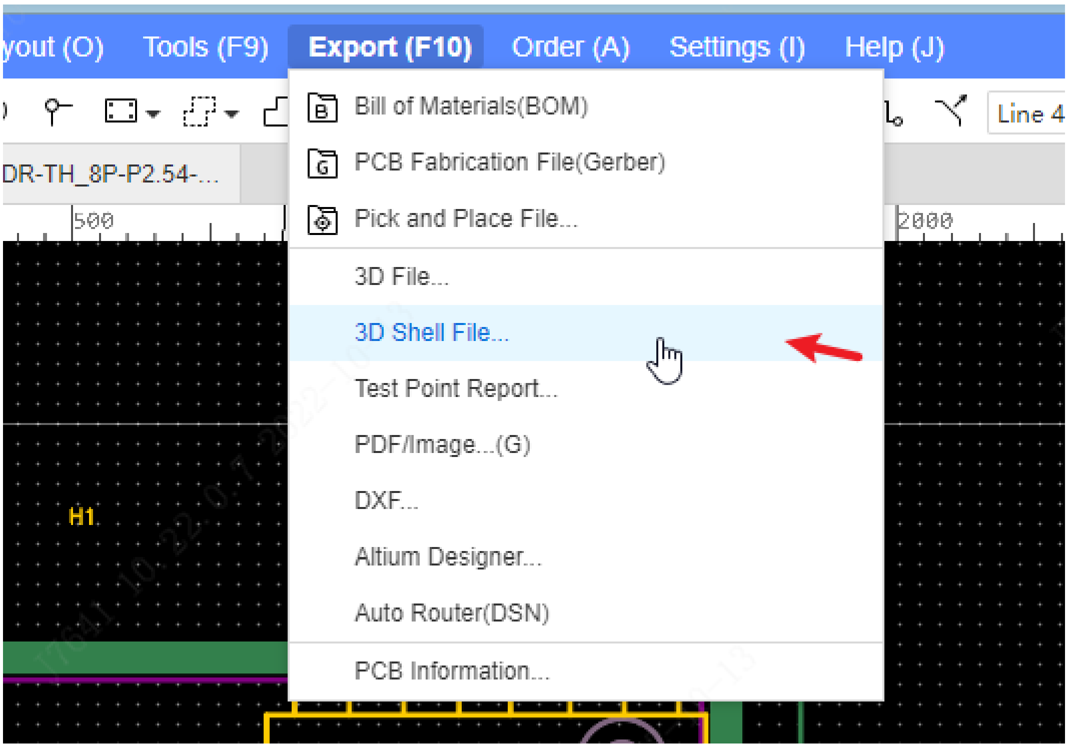


STEP: The exported STEP format must be that the 3D model bound by the PCB device is also in STEP format, otherwise the exported STEP will not have the model.
OBJ: The export OBJ format is a compressed footprint that contains material files. You can use the 3D viewer that comes with window10 to preview.

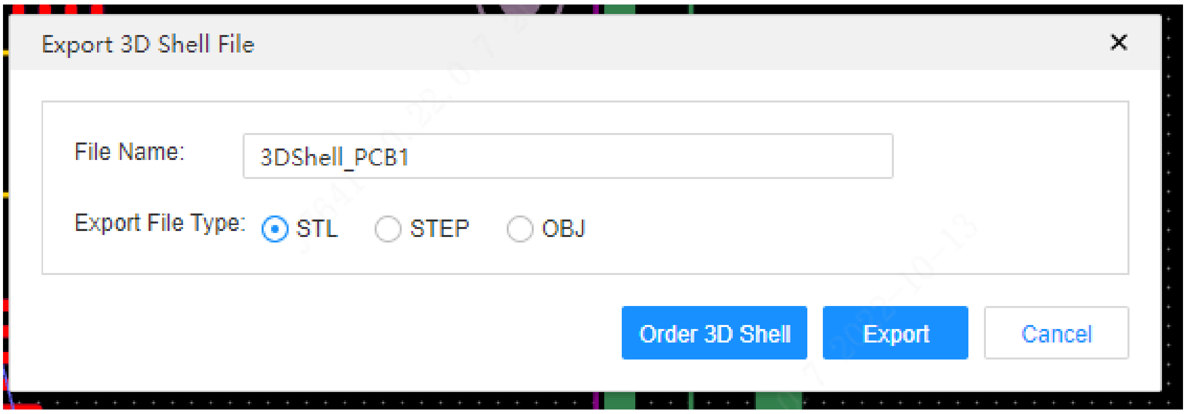


Export 3D Shell File

EasyEDA Pro currently supports exporting simple 3D shell files. Before exporting shell files, please draw 3D shell primitives in the "Place" menu.



It is recommended to export STL for 3D printing, because usually 3D printing manufacturers use STL for production.



If the simple 3D shell designed in EasyEDA Pro does not meet the needs, you can export the STEP or OBJ format and continue the design in other professional 3D design tools.

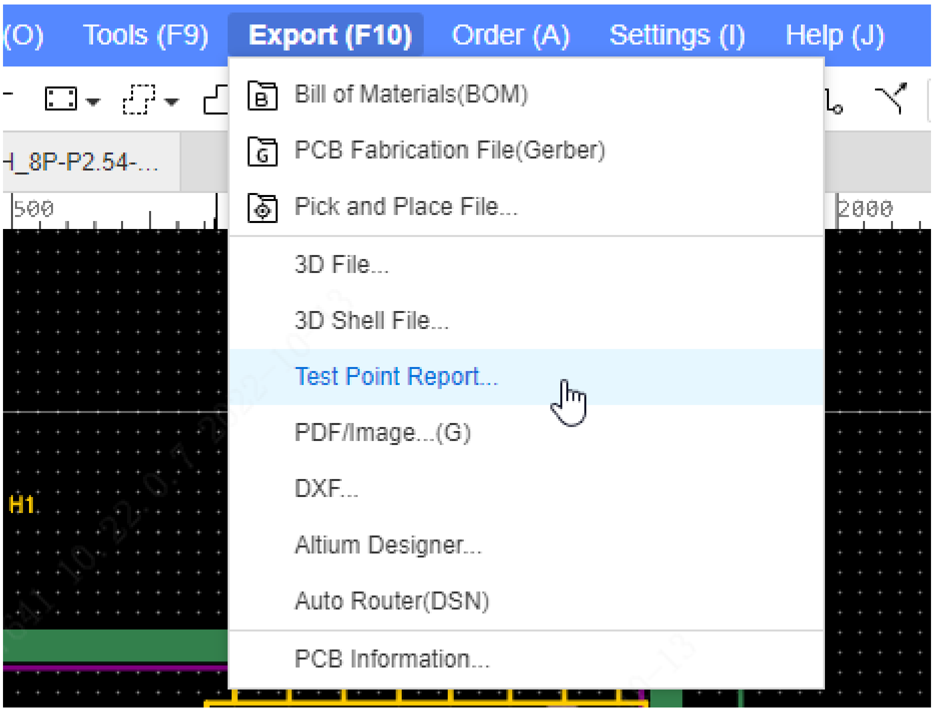
After exporting STL, you will get a compressed footprint, which contains two STL files of the upper cover and the lower cover. To place an order, please visit: [JLCPCB 3D Printing]

Export Test Point Report

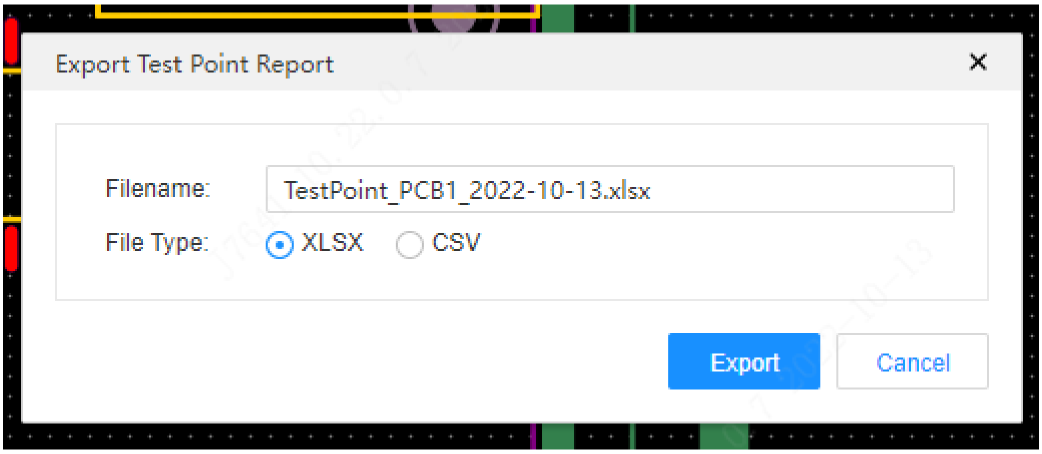
EasyEDA Pro supports exporting the test point report. Before exporting the test point report, you need to place the corresponding test point in the Place menu.

Export method:

- Top Menu - Export - Test Point Report



Support custom file names, and file types.



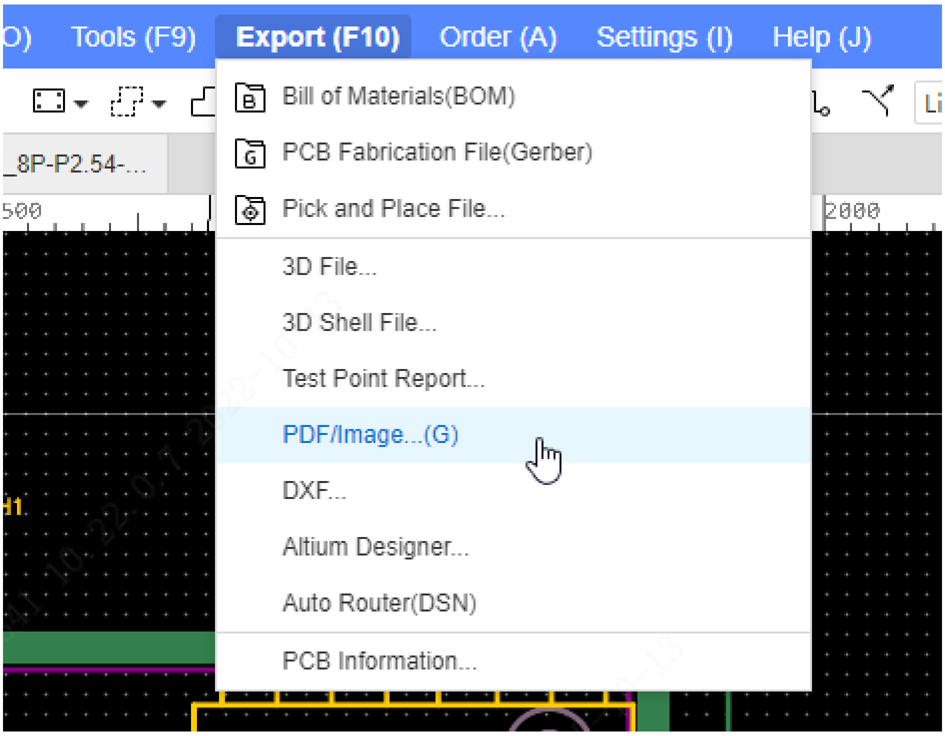
Open test point report:

	A	B	C	D	E	F
1	Number	Net	Center X	Center Y	Layer	
2	TP1		800 mil	-650 mil	Top Layer	
3	TP2		725 mil	-845 mil	Top Layer	
4	TP3		485 mil	-620 mil	Top Layer	
5	TP4		607.6 mil	-726.5 mil	Top Layer	
6	TP5		433.6 mil	-726.5 mil	Top Layer	
7	TP6	$1N1565	246.3 mil	-946 mil	Top Layer	
8						
9						

Export PDF/Image

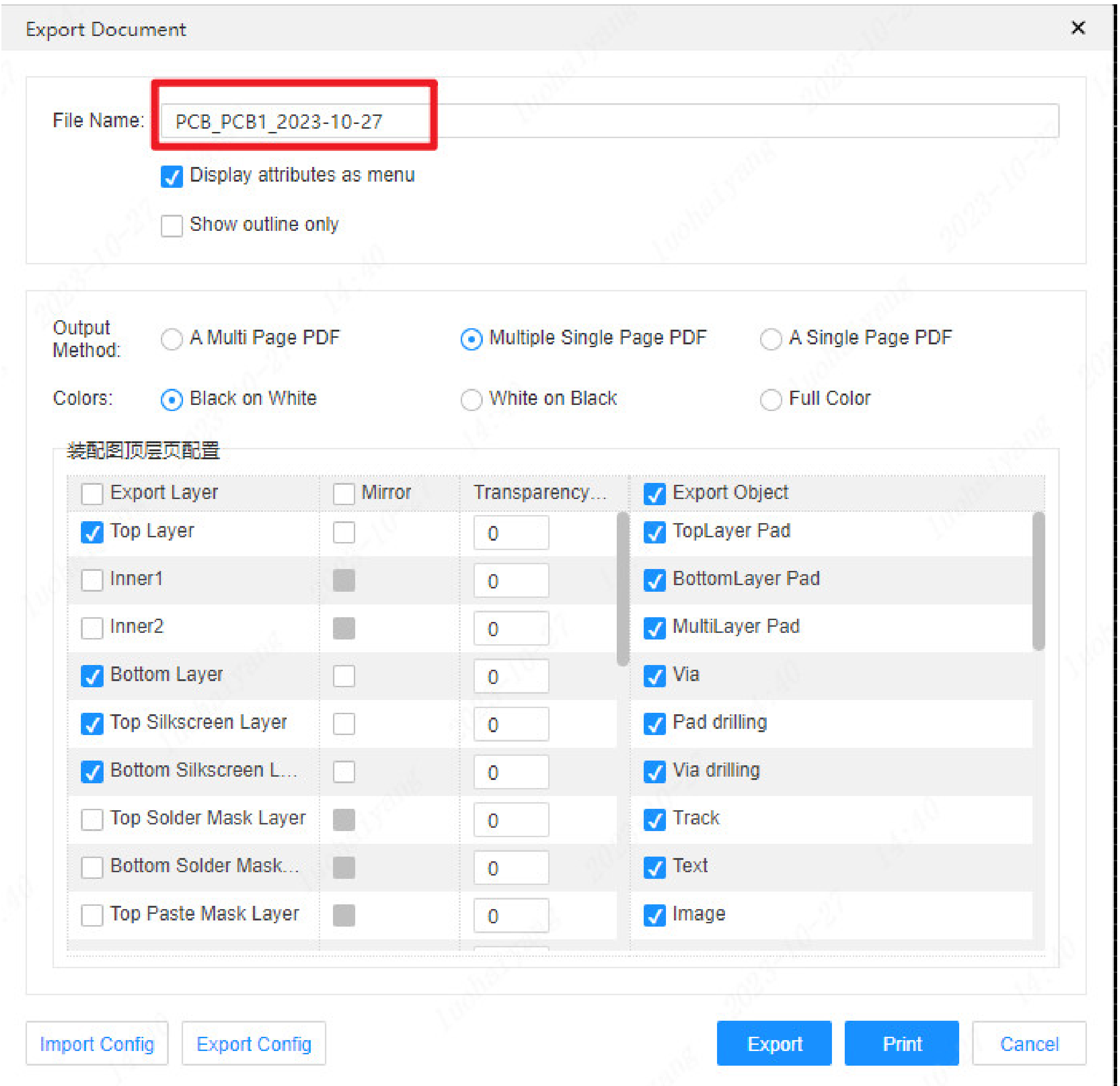
The export PDF functionality for schematics and PCBs is different, but the operation is similar. It can support exporting layers and objects separately, and set export mirroring and transparency.

- **Top Menu - Export -PDF/Image**



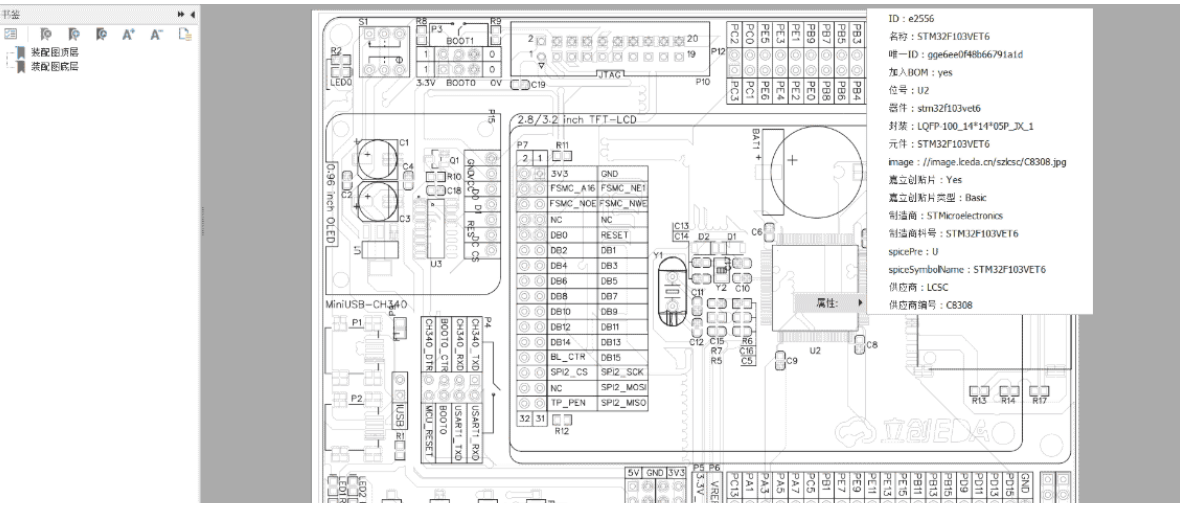
The Settings window will pop up after clicking

The name of the exported file can be set at the top of the window



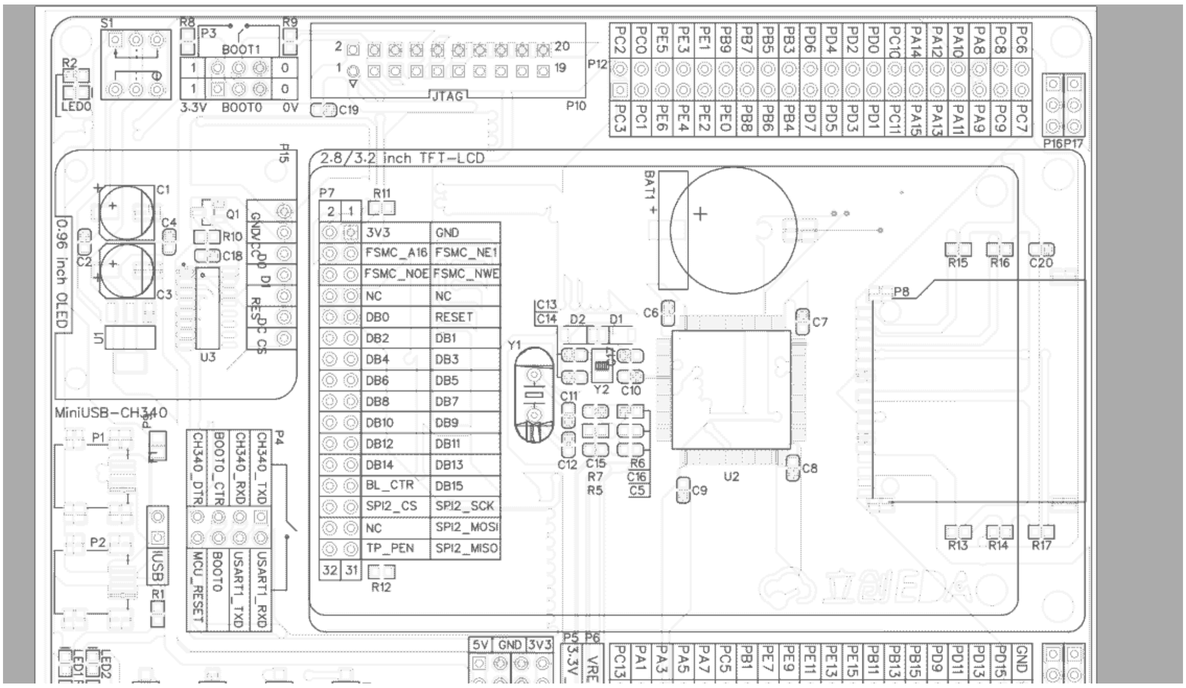
Display attribute as menu

After selection, the exported PDF is opened in a PDF reader, click Device Yes to view the properties of the device.



Show outline only

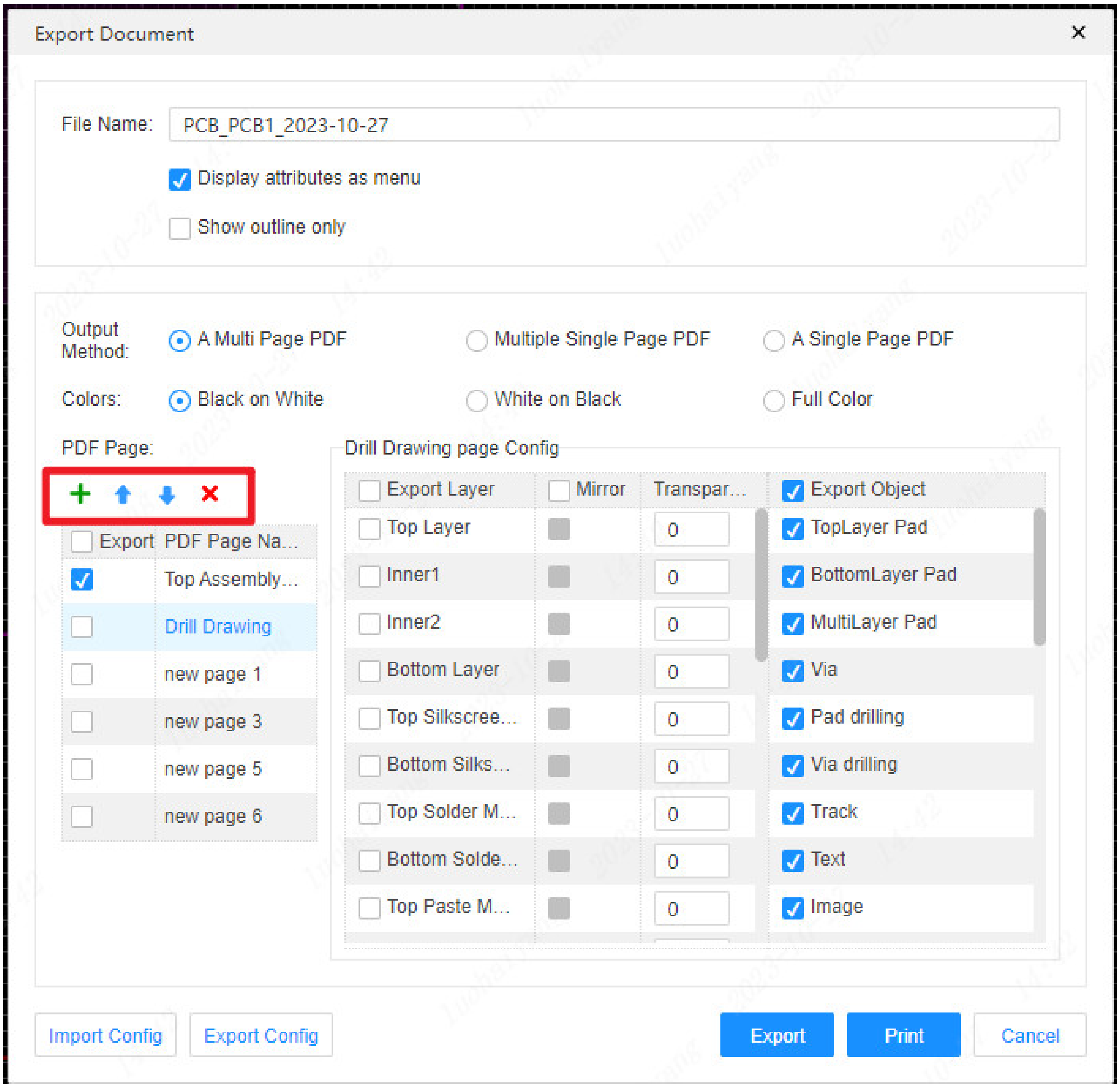
After selecting, the pads, wires, and outline primitives of the exported PDF will only display outlines.



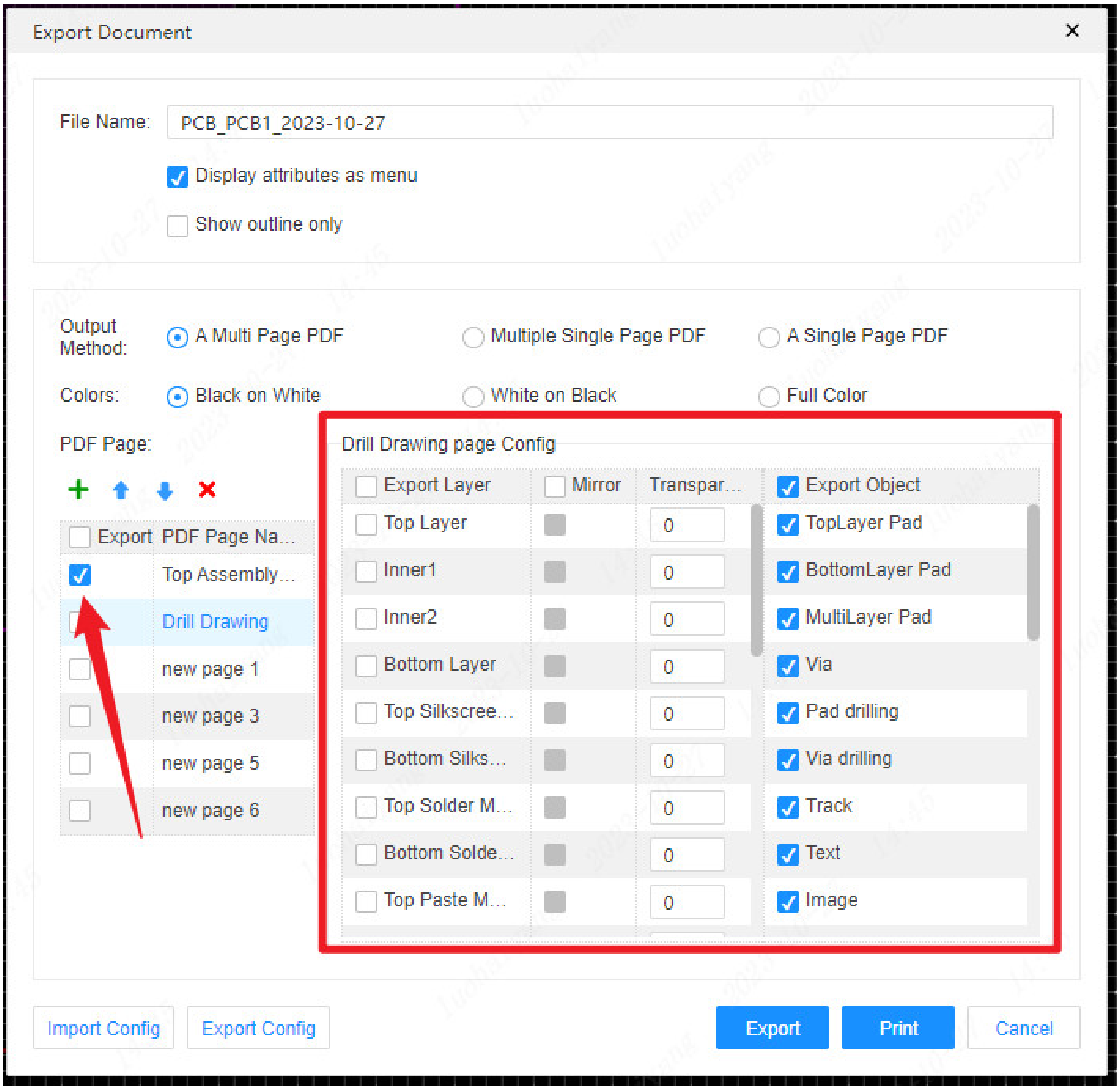
output method

Single Multi-Page PDF: You can export multiple pages of different content in the same PDF file

Click the green plus sign to add a PDF, click the up or down arrow to adjust the order of the PDF, and click the red × to delete the PDF.



You can choose whether to export the page or not by configuring the checkbox. After clicking the name of the PDF page, the right side can configure the layers and objects that the page needs to export



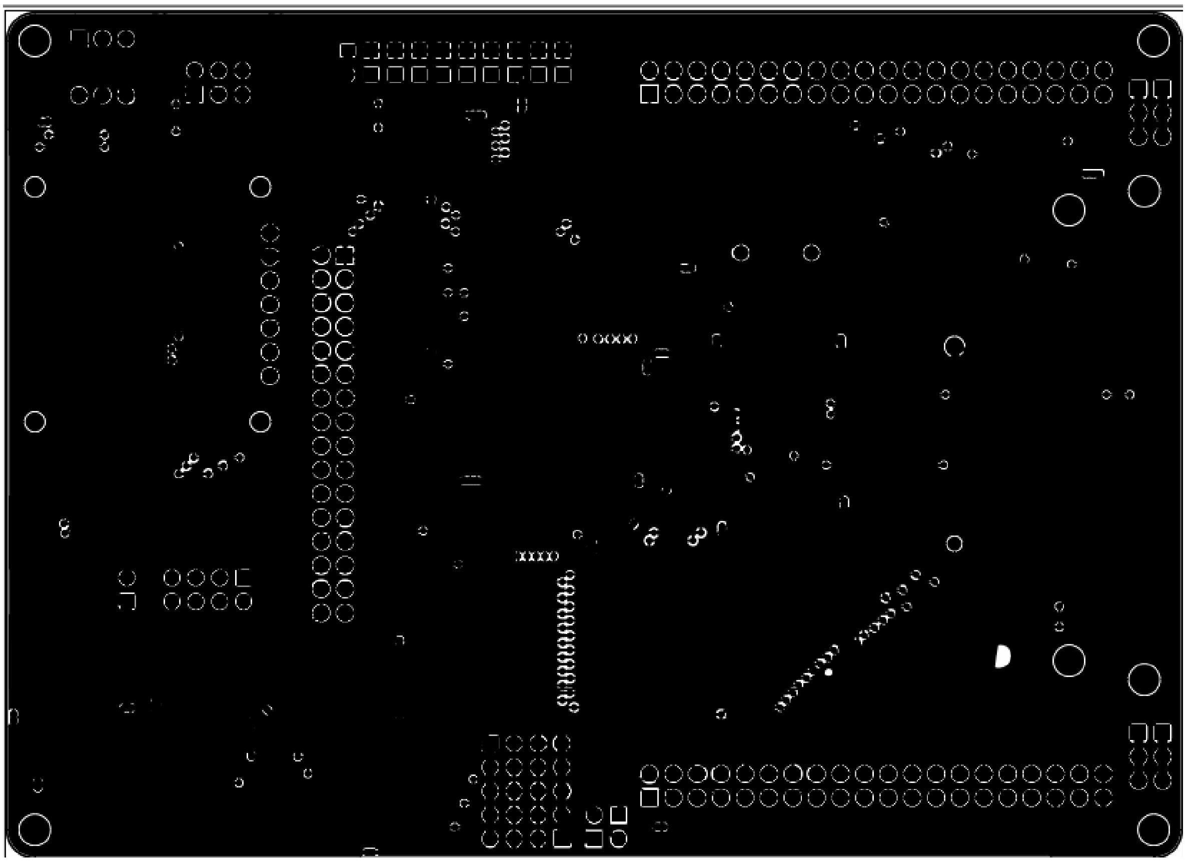
Multiple single-page PDFs: Selecting this option will export each layer as a separate PDF and then merge it into a single package. Below you can choose which layer to export, whether to mirror it or not, and adjust the opacity. Exported object selection.

Single single-page PDF: Export all layers as a drawing page, you can also choose the exported layers, transparency adjustment, and exported object selection.

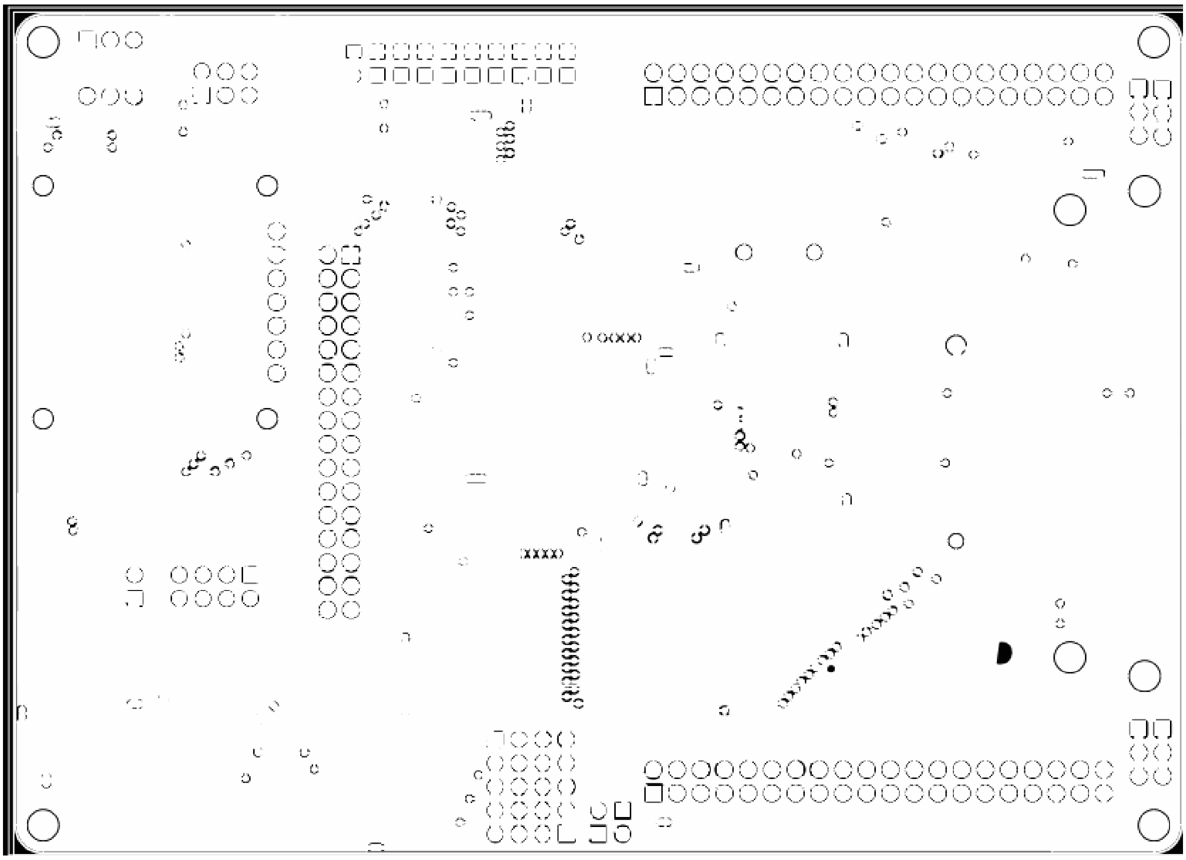
Colors

You can select the color as you want, support three colors: Black on White, White on Black, and Full Color

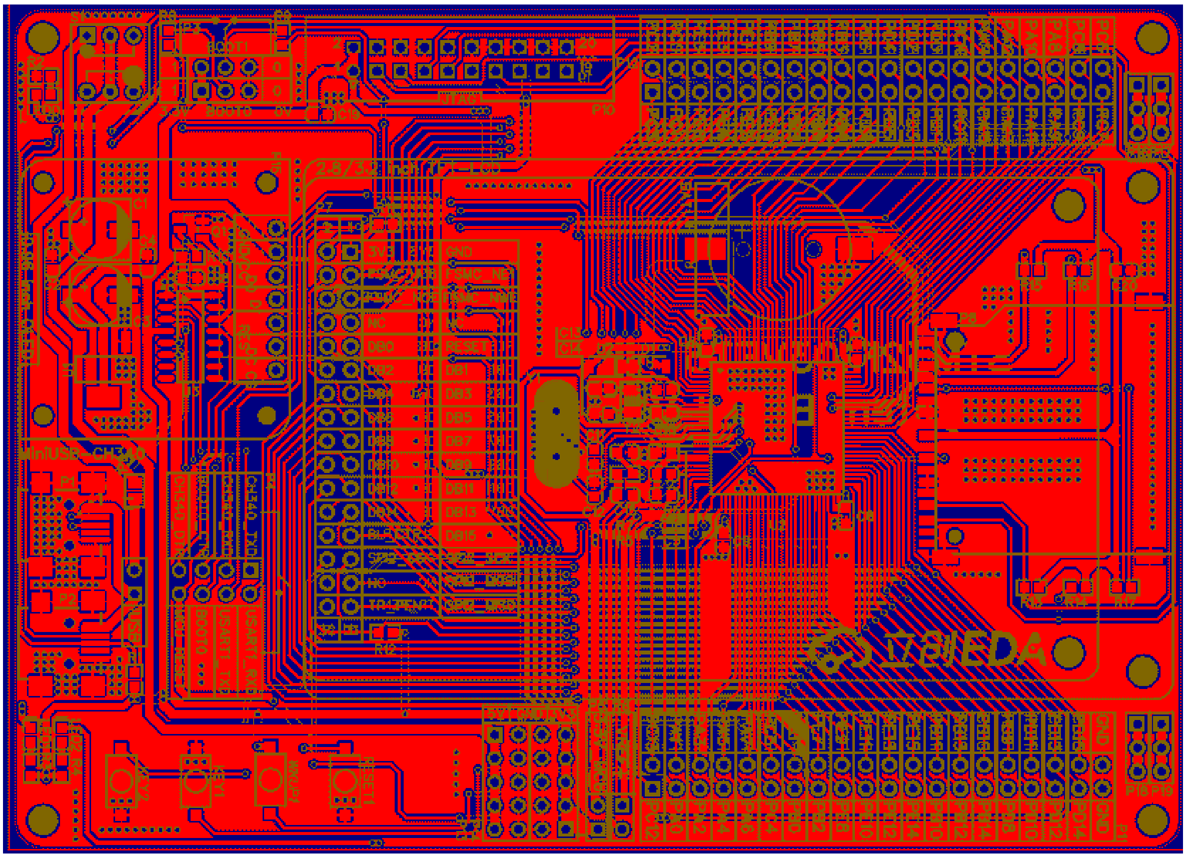
Black on White



White on Black



Full Color



PDF Page Config

EasyEDA support to config many configs to export the PDF, you can make each config to decide to export which layer which object.

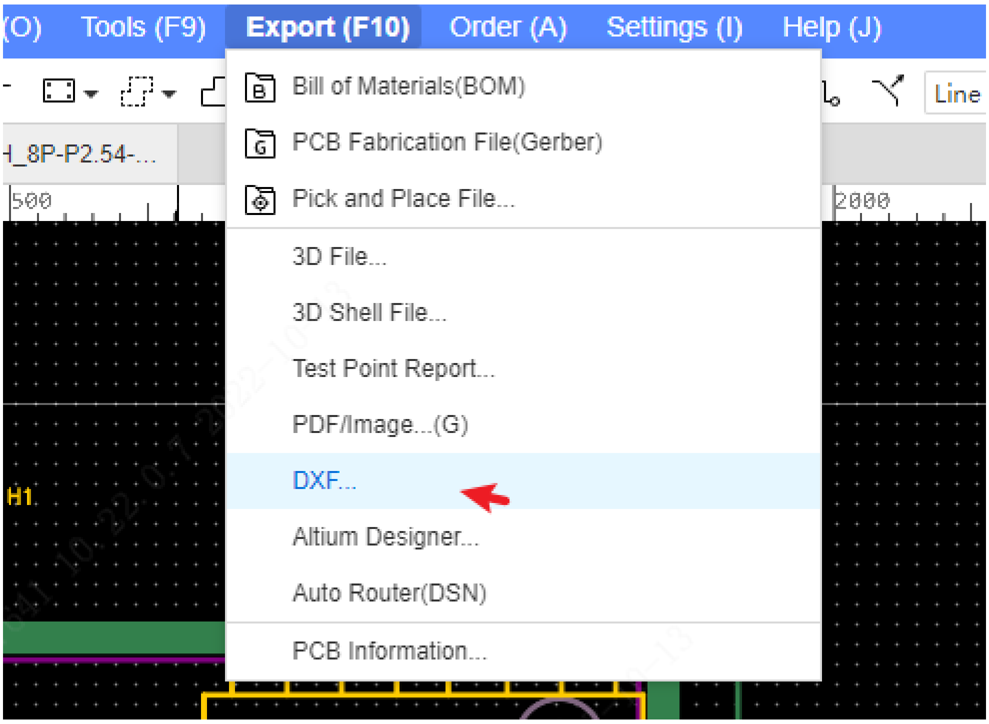
You need to know, one config one page, if you select multiple configs, then export multiple pages

Export DXF

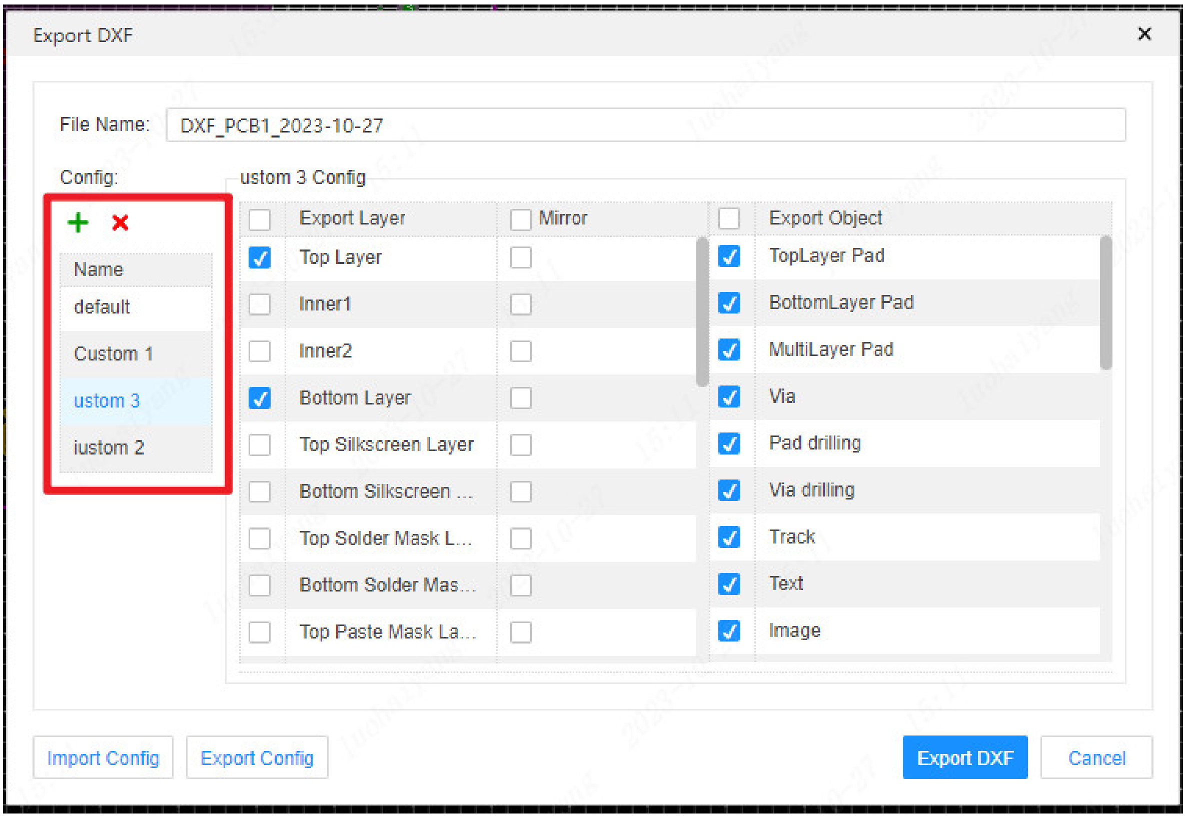
EasyEDA Pro supports powerful export DXF function.

Steps:

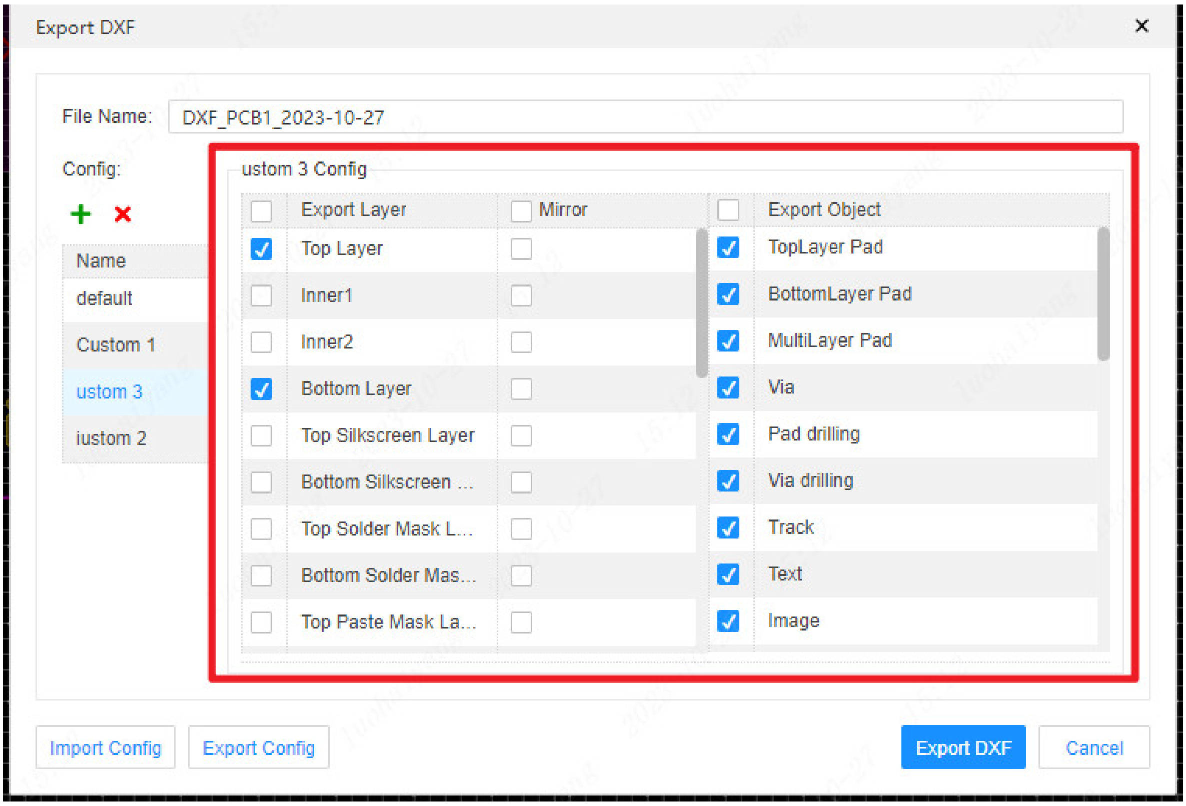
- Click on the top menu: File - Export - DXF or Export - DXF



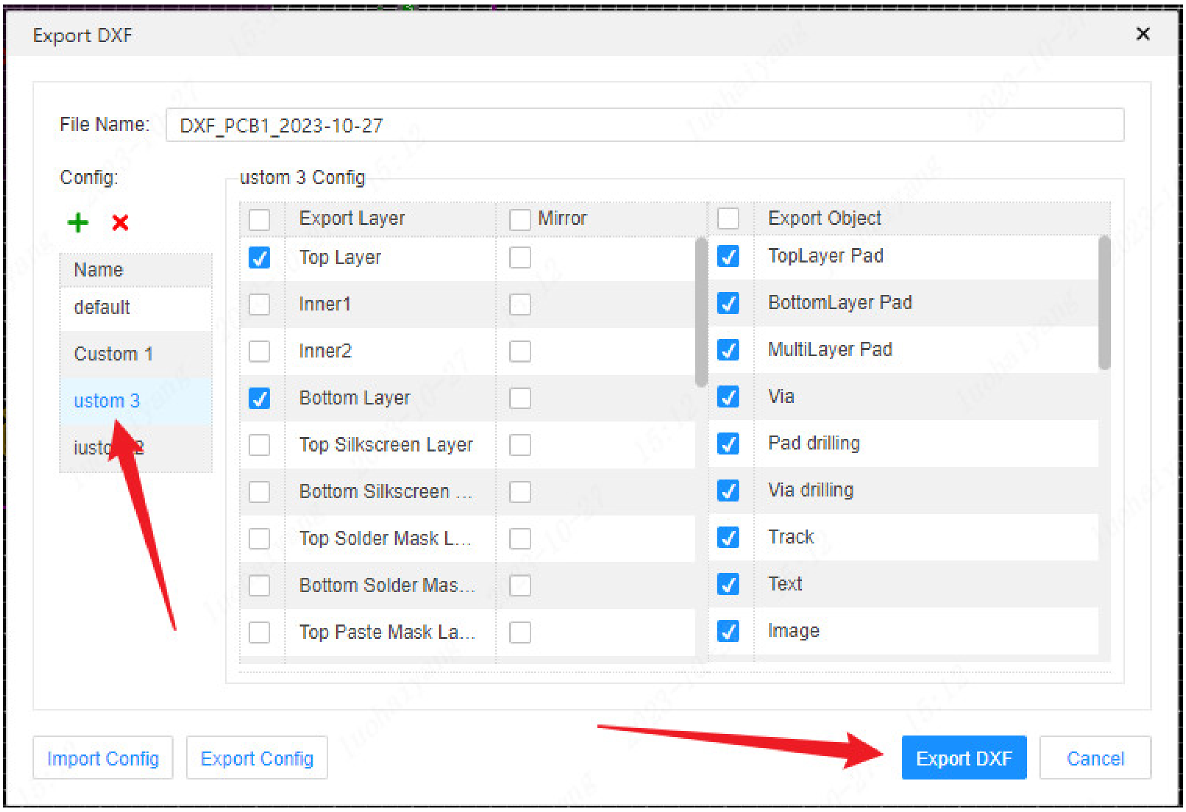
On the left you can set some presets, which are exported once and then saved



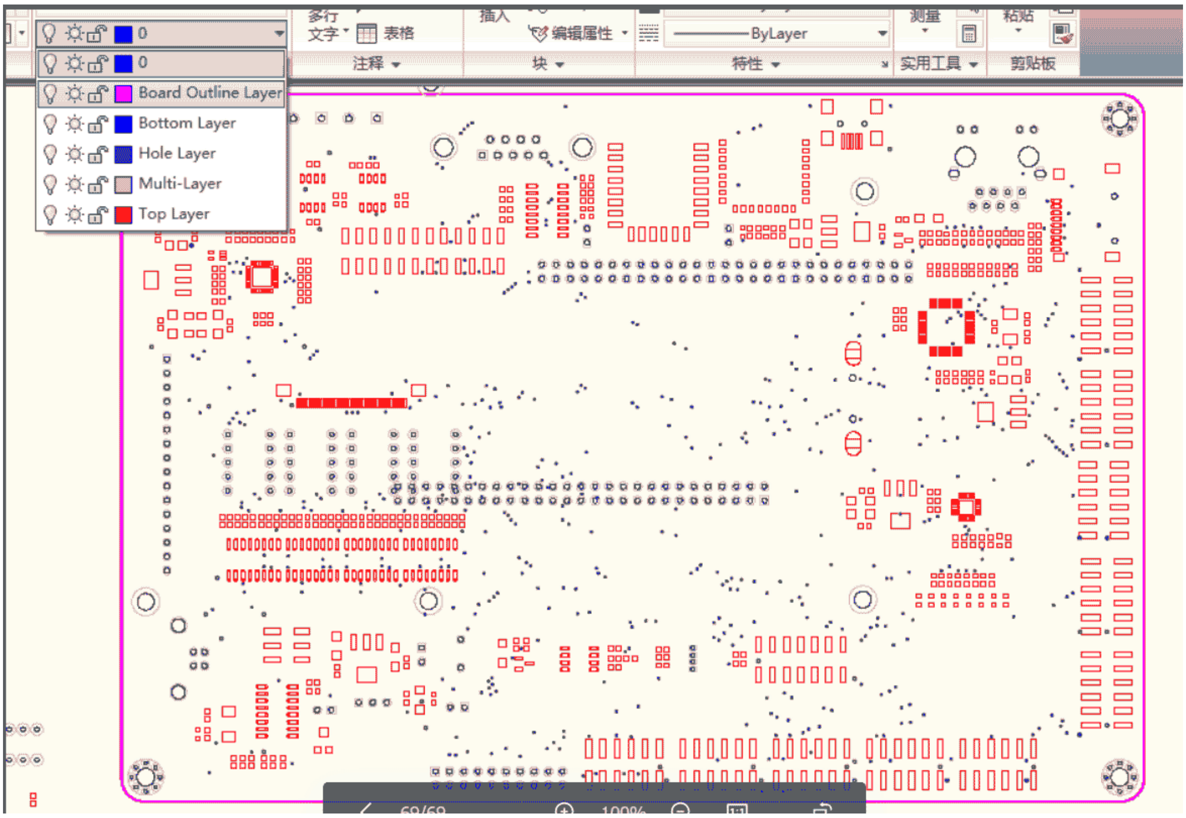
The bottom right allows you to configure what layer your preset should export, whether that layer should be mirrored, and what objects should be exported



Select one of your preset configurations and click Export to export a corresponding DXF file



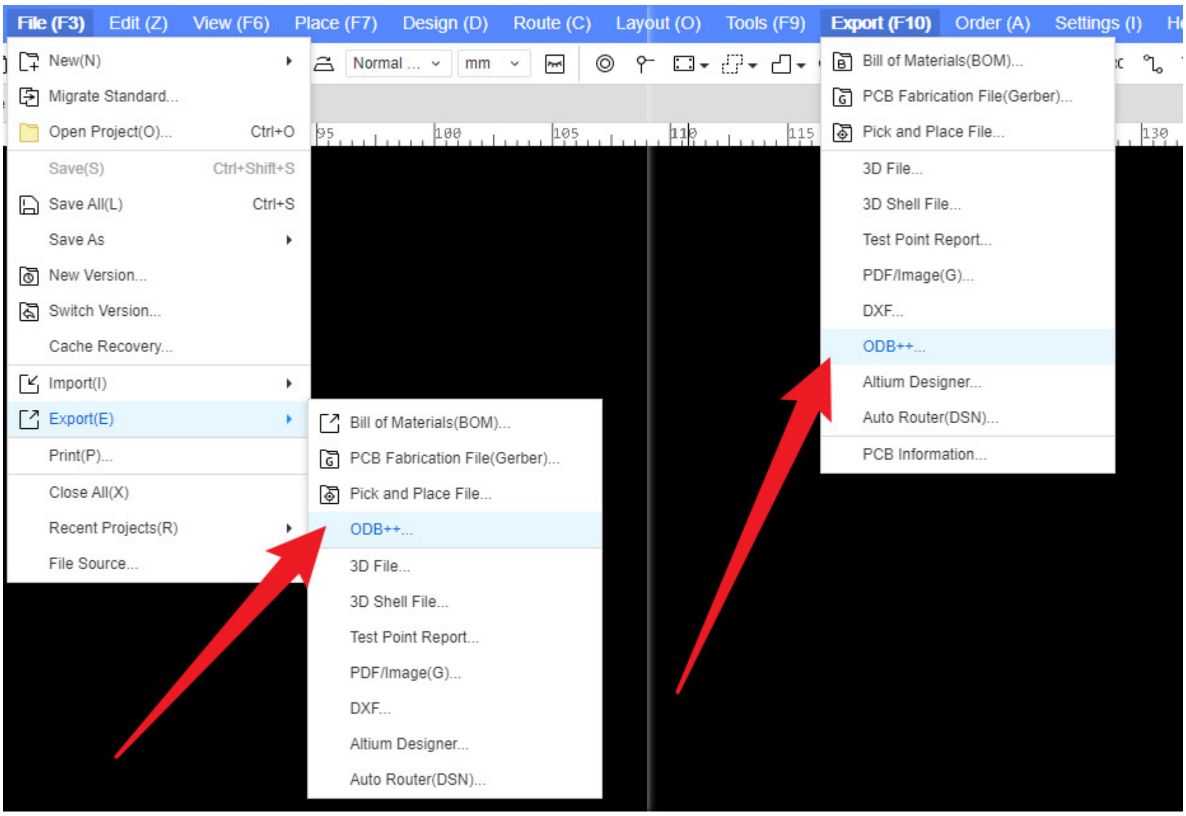
The exported DXF will be layered according to the layers you choose



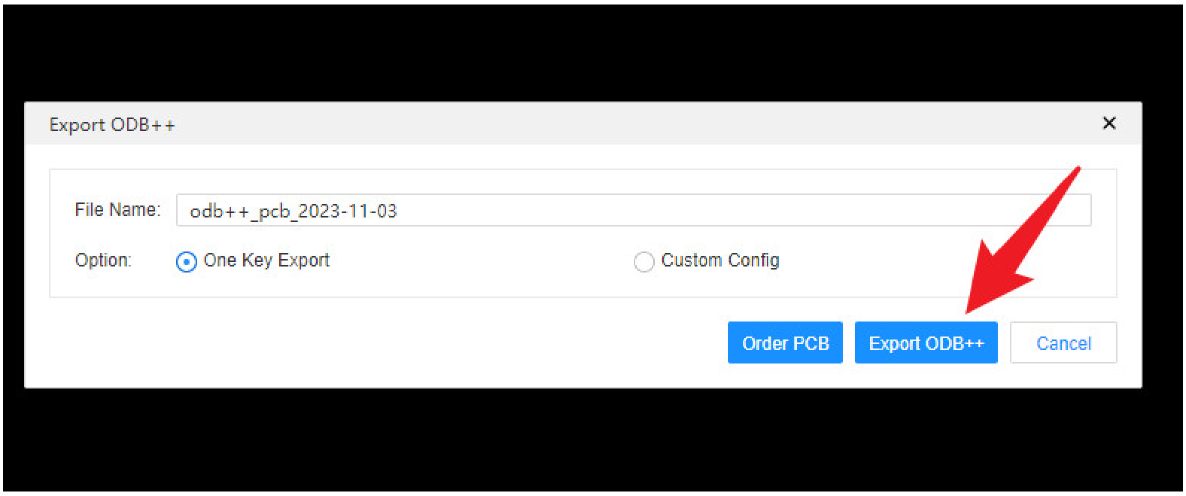
Export ODB++

Function entry

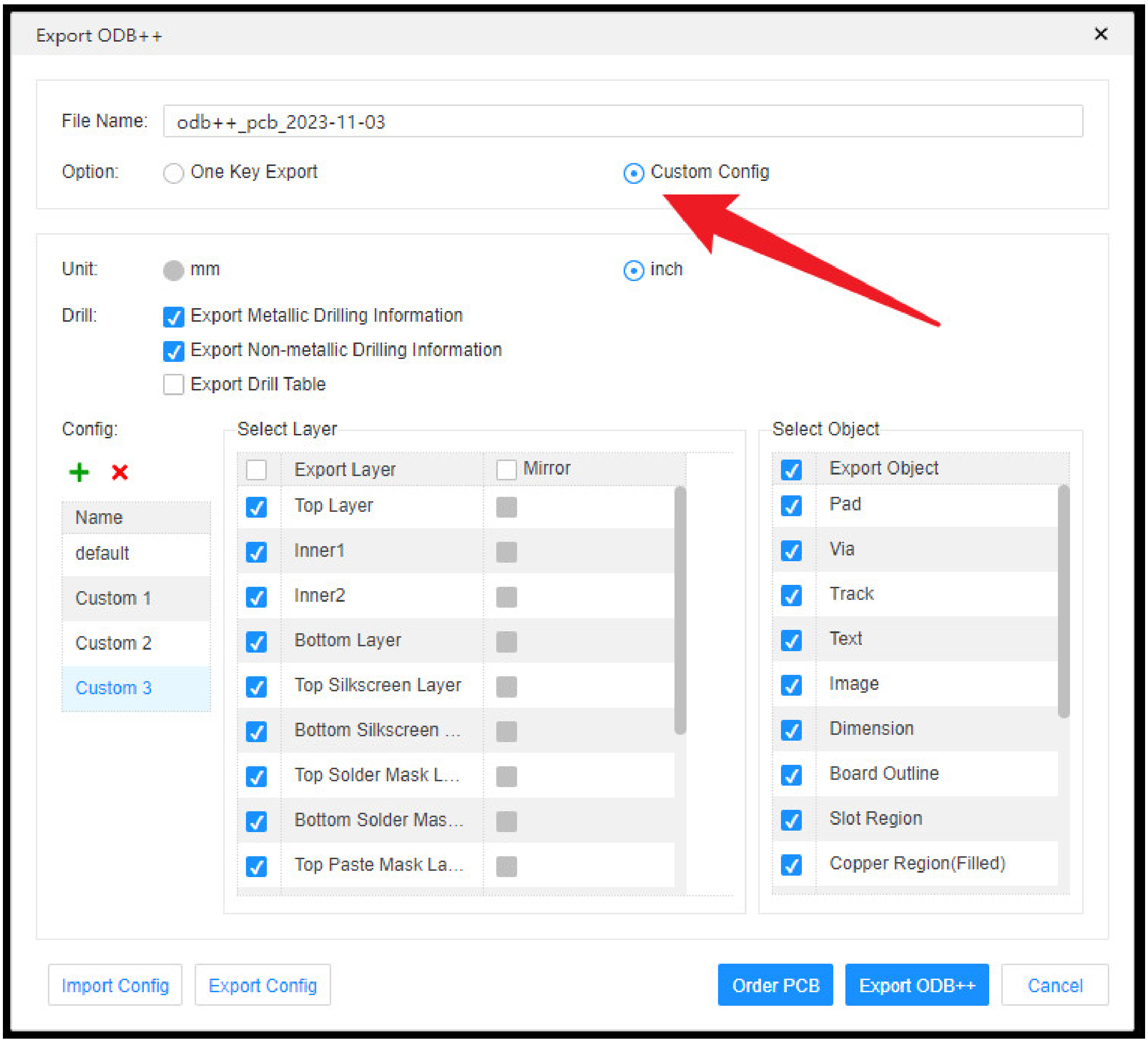
- On Menu - File - Export - ODB++
- On Menu - Export - ODB++



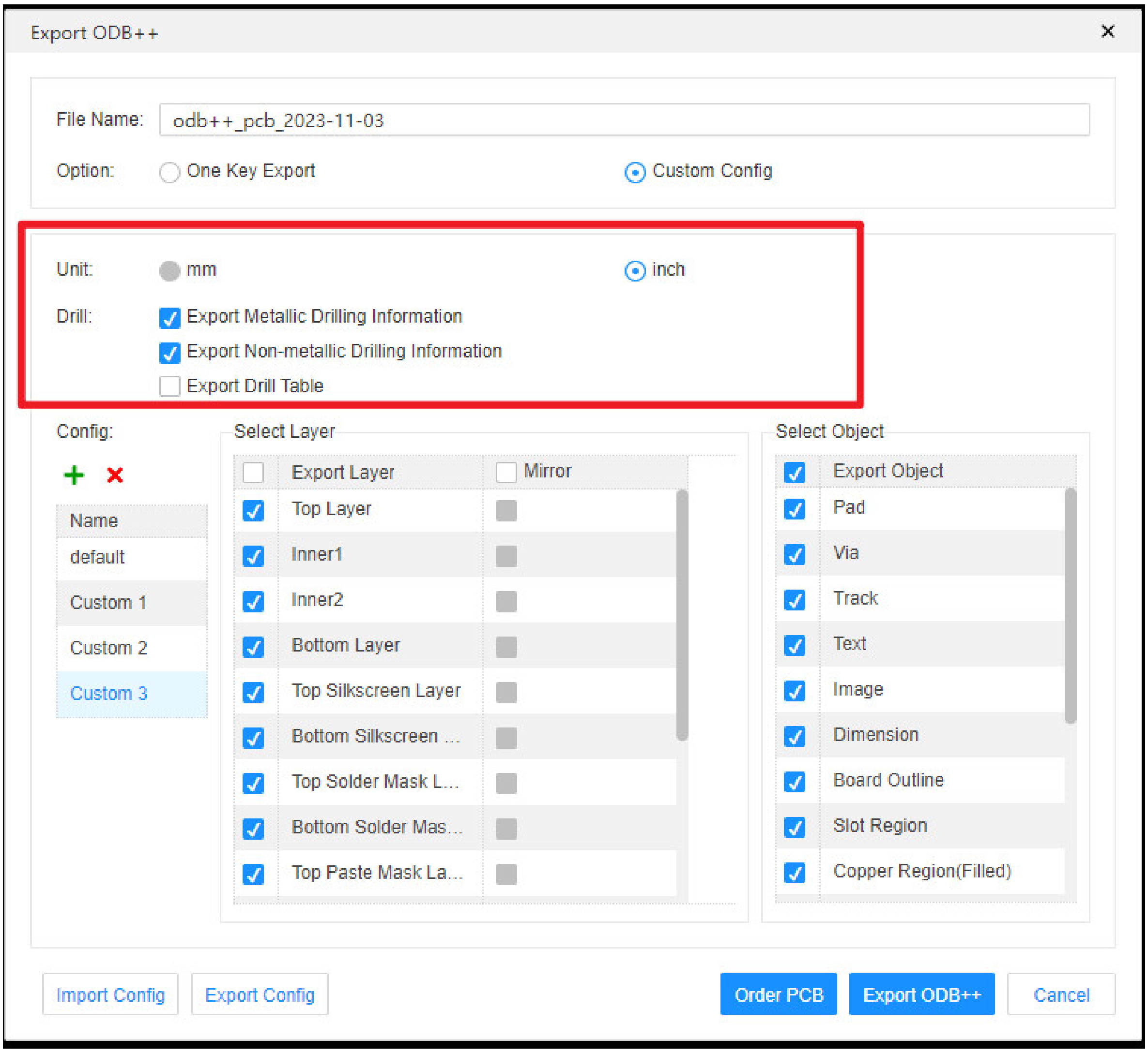
After opening the window, click Export ODB++ button to export the ODB++ file



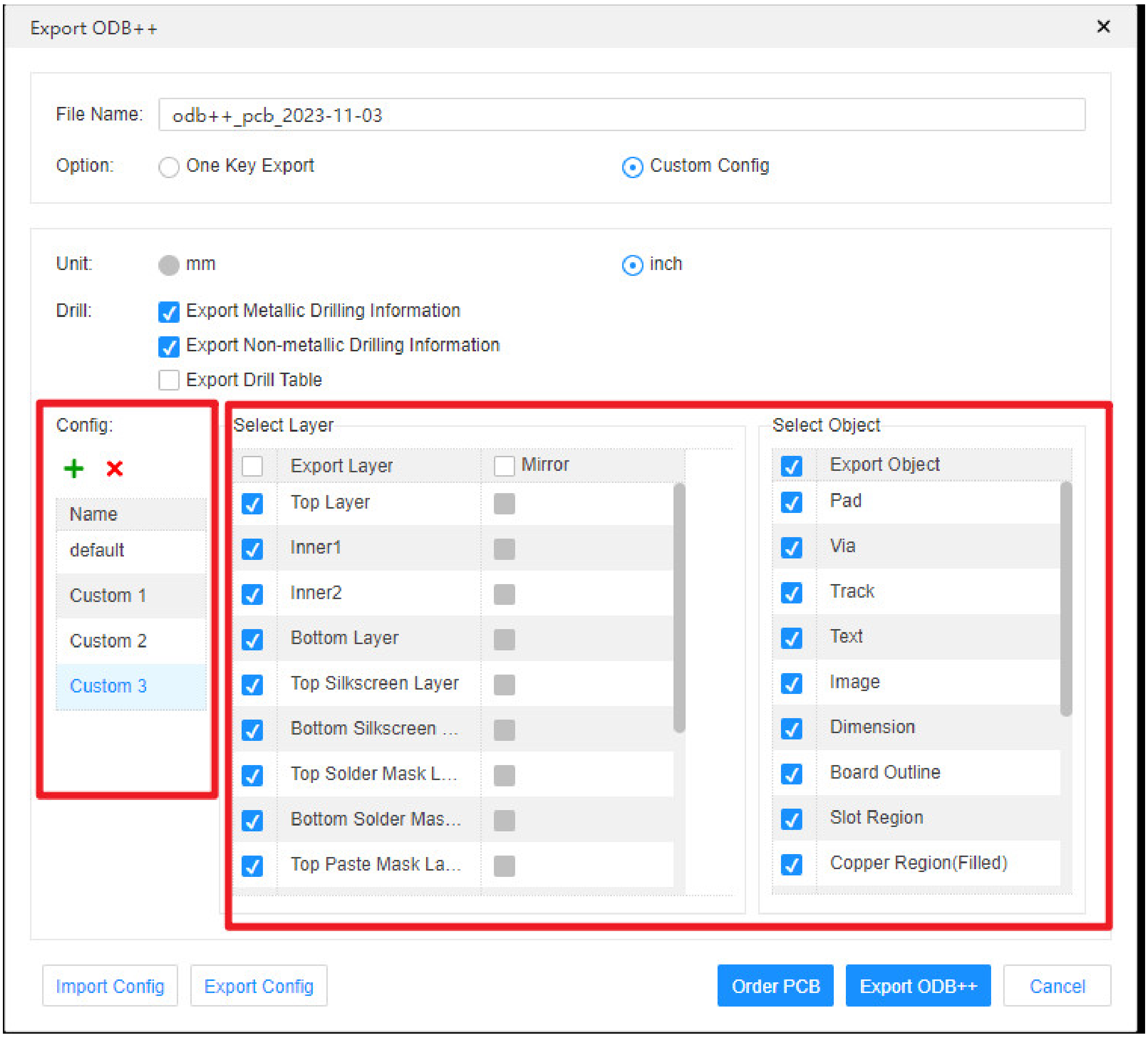
Or customize the exported content



Here you can customize the exported unit and the relevant content of the drilling.



You can add different configurations below. Click on the left to modify or export the configuration



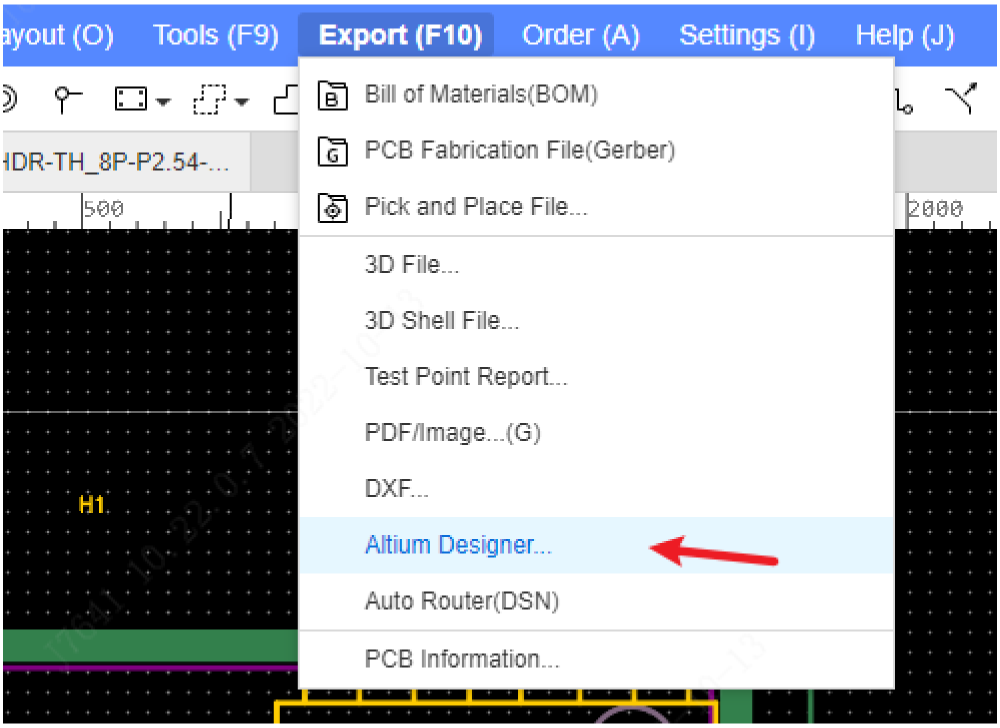
Export Altium Designer

EasyEDA Pro currently supports exporting to Altium Designer.

Due to differences in file formats, types and properties of primitives, etc., exporting to Altium Designer cannot ensure 100% consistency. You need to open the Altium Designer file and check the details yourself.

Entry: Top Menu - Export - Altium Designer, or File - Export - Altium Designer.

Export Exports the entire project to a compressed footprint by default.



Known issues:

1. The copper filling is not exported
2. The font of the text class cannot be completely matched, and the position of the text will be slightly offset
3. Design rules are not yet supported
4. The layer stack is not support yet

If you find some differences in exporting Altium files that affect usage, please contact us to fix it.

Export Autorouter DSN

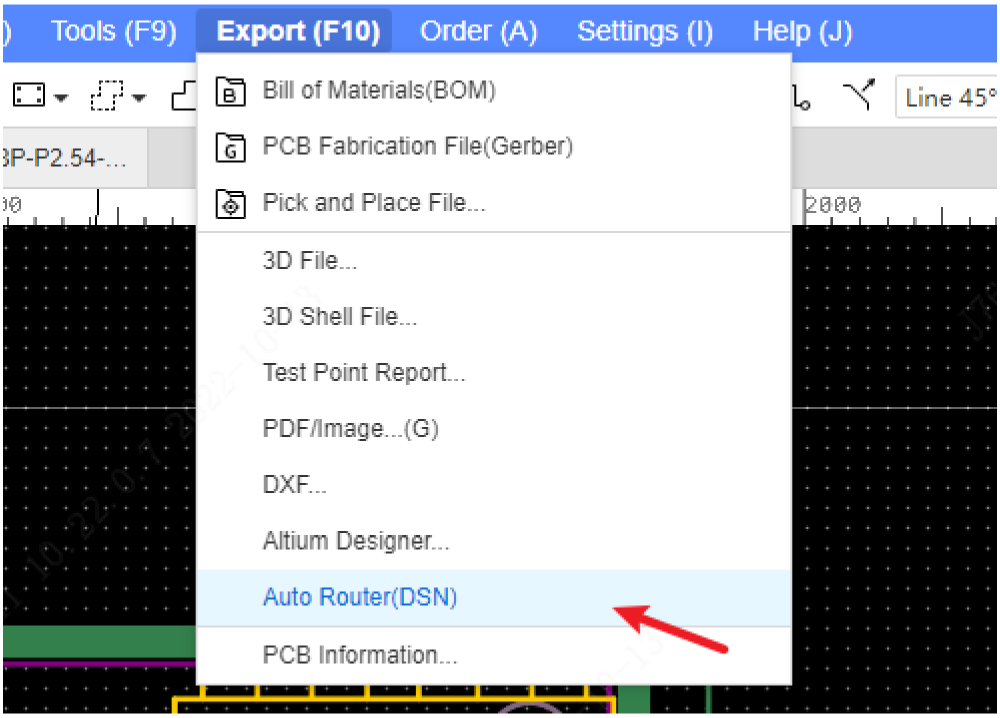
Autorouting files are intended for use by third-party autorouting tools.

If the built-in auto-router cannot meet the needs of auto-routing, Jiali Chuang EDA supports exporting the auto-routing file dsn and importing the auto-routing session file ses. You can use the third-party auto-routing tool for wiring by exporting the auto-routing file, and then import the ses file. That's it.

Steps:

1.Open the PCB to export the automatic routing file dsn, it is recommended to remove the unnecessary vias and wires first.

Via: Top Menu - Export - Autoroute (DSN), or Top Menu - File - Export - Autoroute (DSN)



2.Open the third-party auto-routing tool and open the auto-routing file DSN

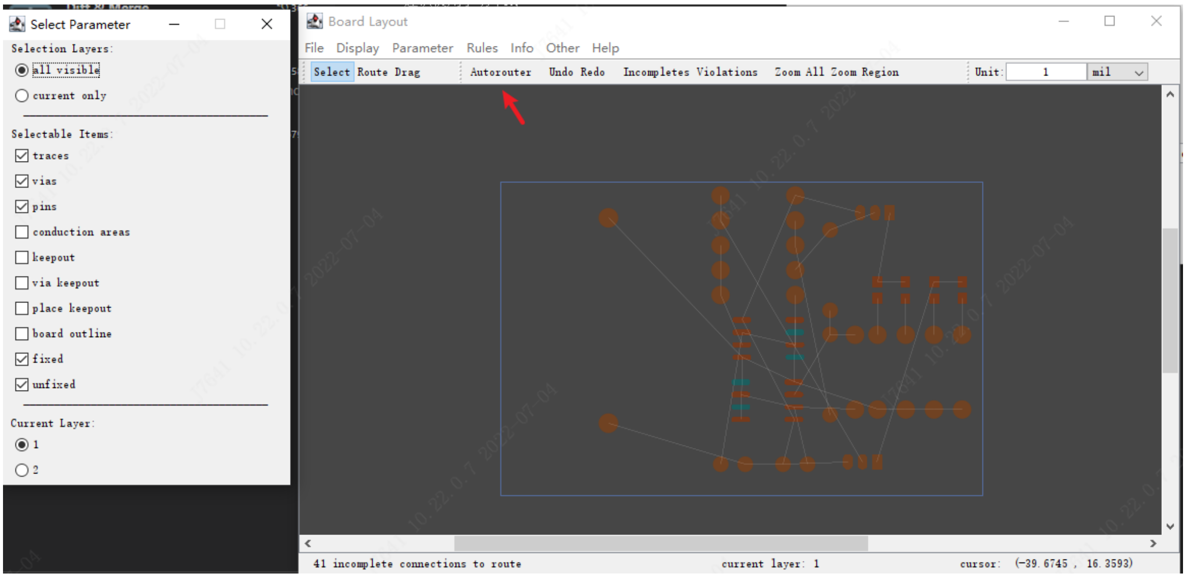
Third-party auto-routing tools include: Freerouting, ELECTRA, TopoR, etc.

Freerouting download address: https://github.com/freerouting/freerouting

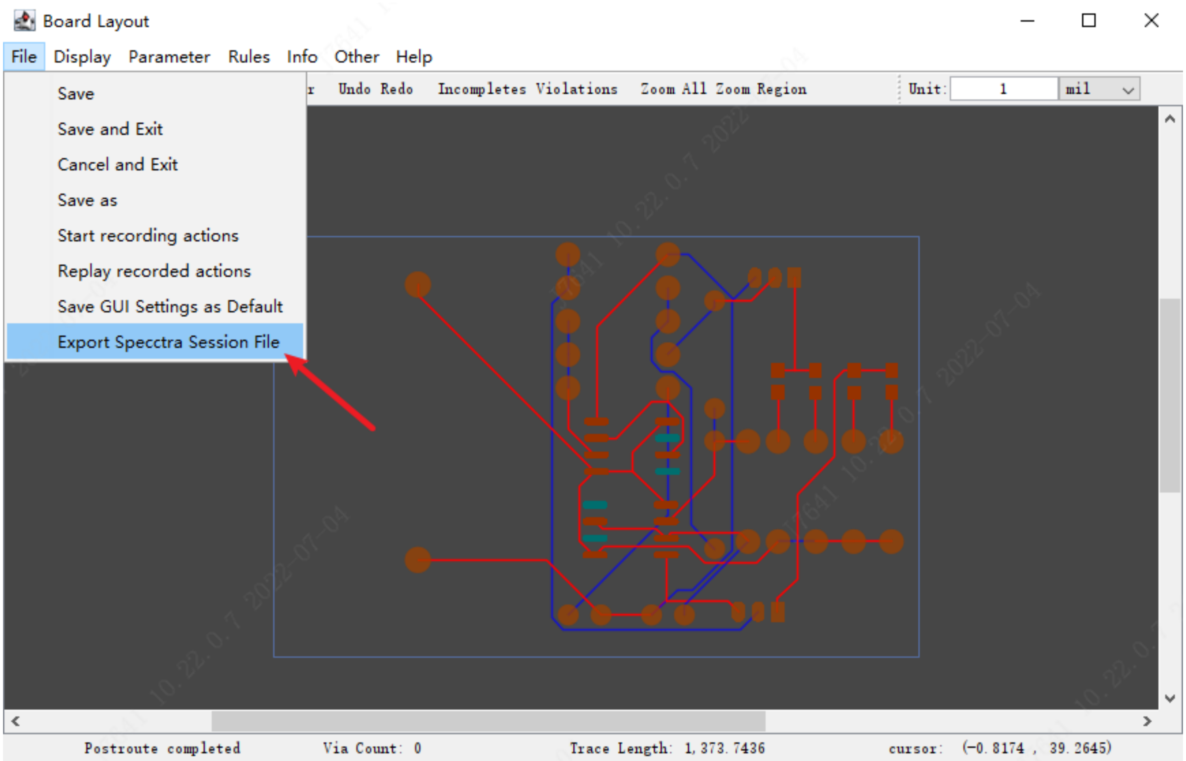
Take Freerouting as an example:

1) Open Freerouting, open the automatic routing file SES

2) Set the automatic routing rules, if you do not need to set, you can directly point the routing

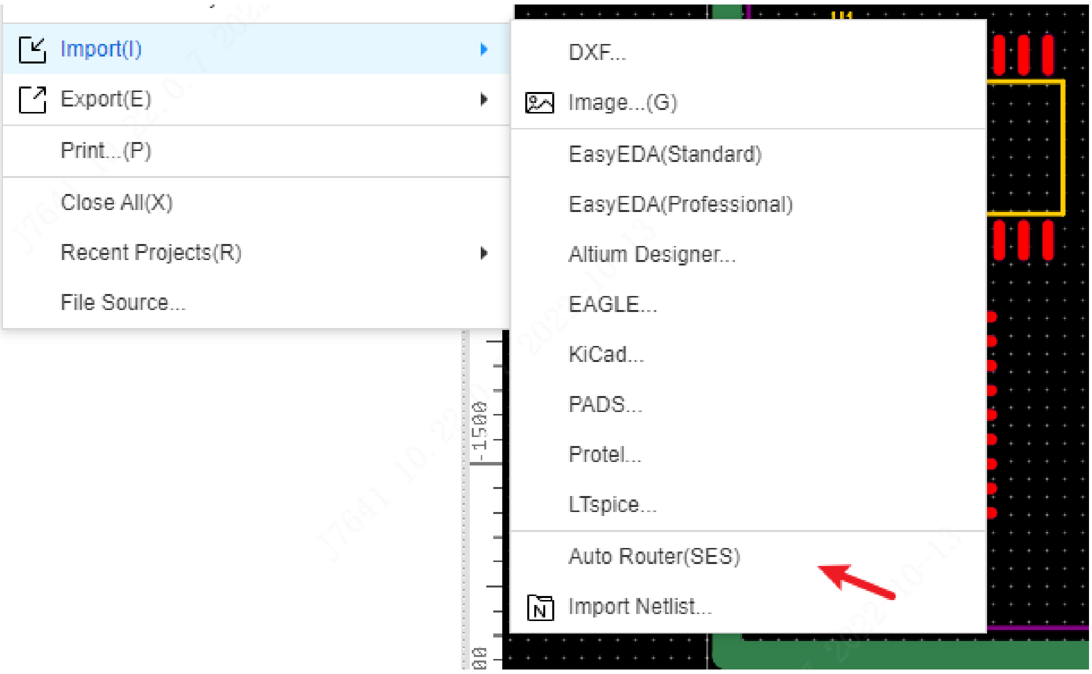


3) After completion, export the automatic routing session file ses, which will be saved in the same directory as dsn



3.Import the auto-routing session file into the PCB and complete the import

Entry: Top Menu - File - Import - Autorouting (SES).

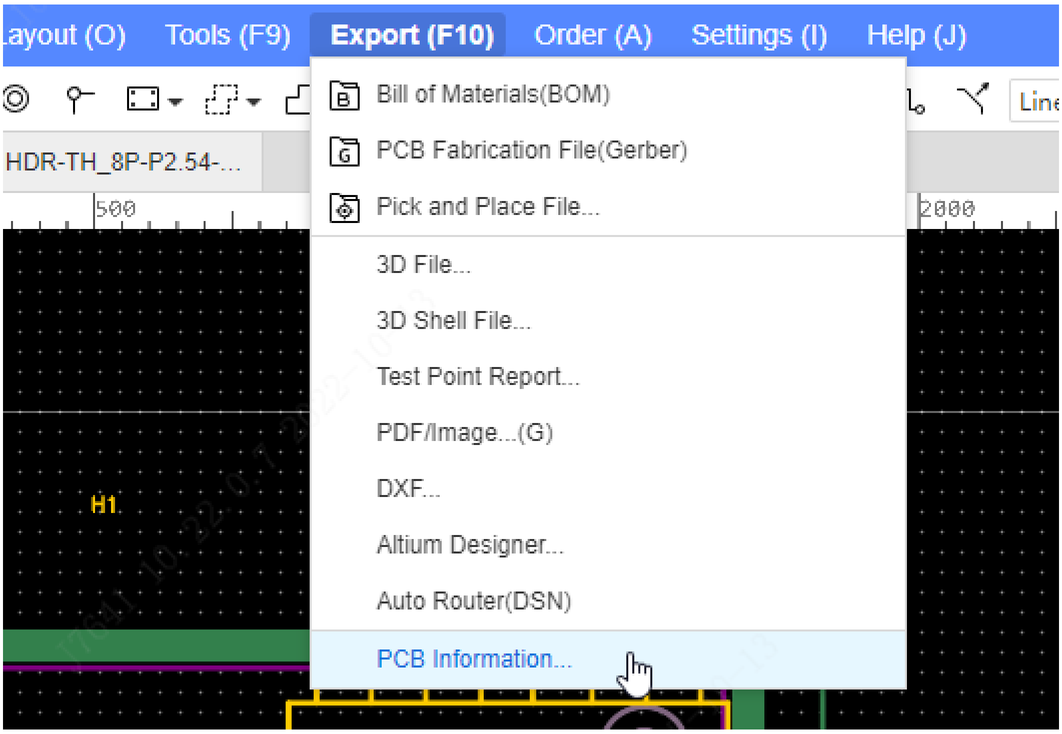


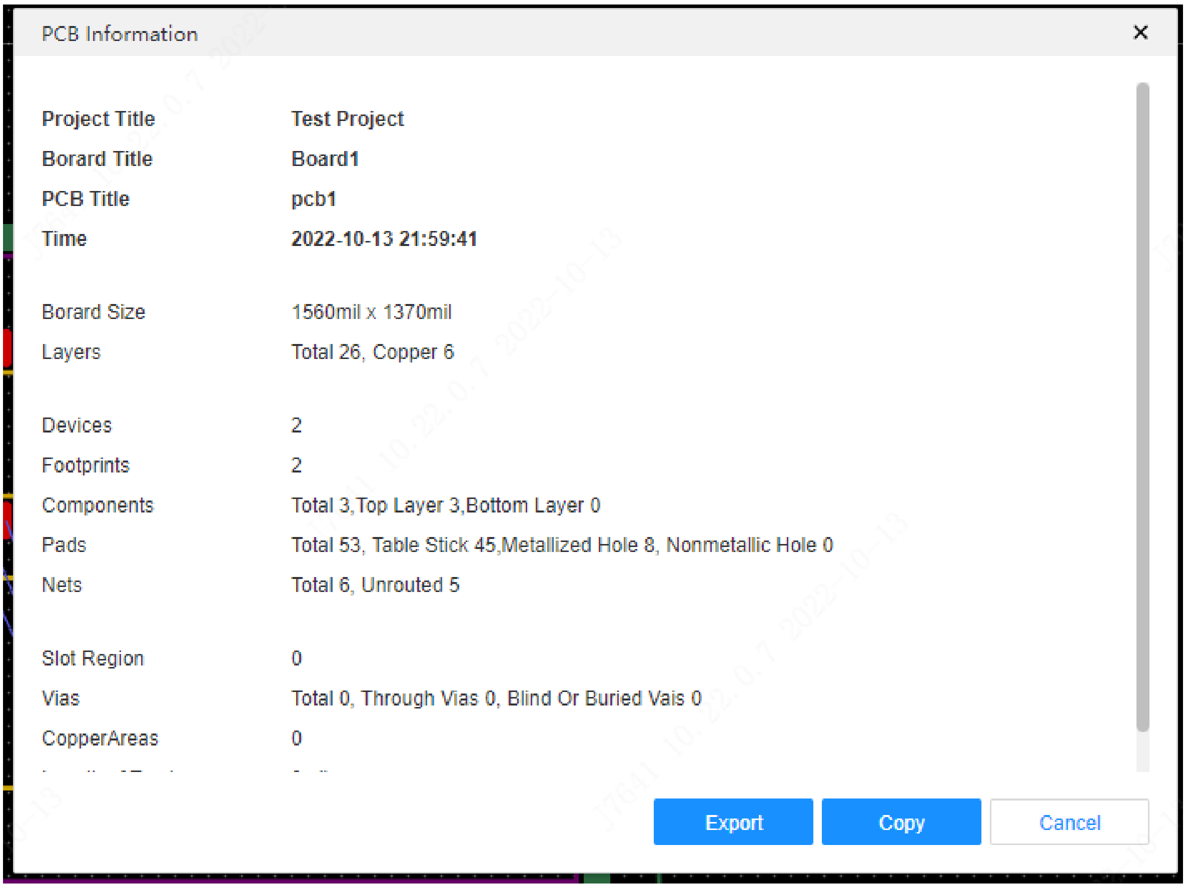
The editor will automatically generate wires and vias to complete automatic routing.

PCB Information

Support to view the relevant statistical parameters of the current PCB, such as the number of pads, the number of copper layers, etc.

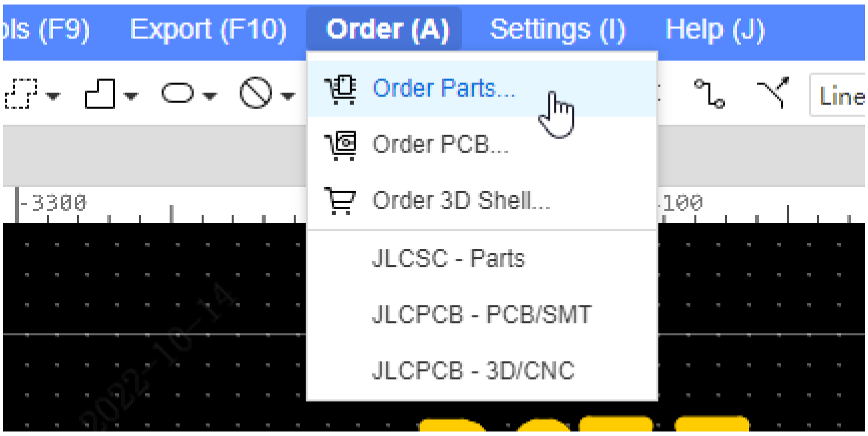
Entry: Top Menu - Export - PCB Information.



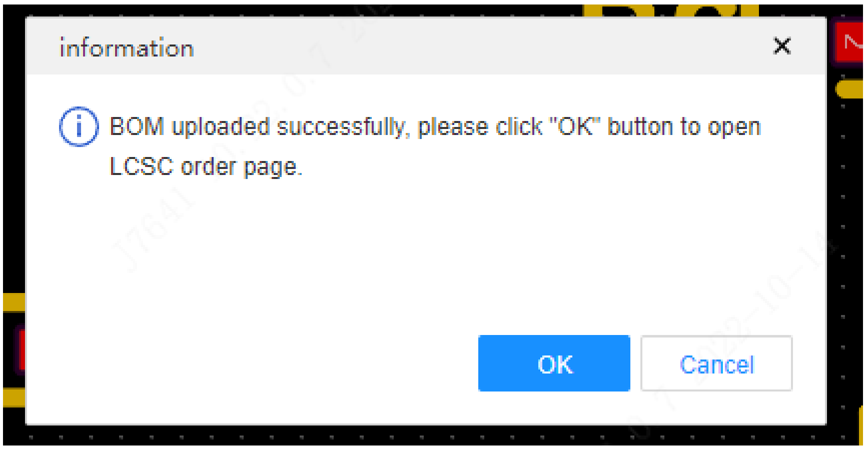


Order Parts

After clicking, the BOM will be uploaded to LCSC Mall according to the current PCB components.

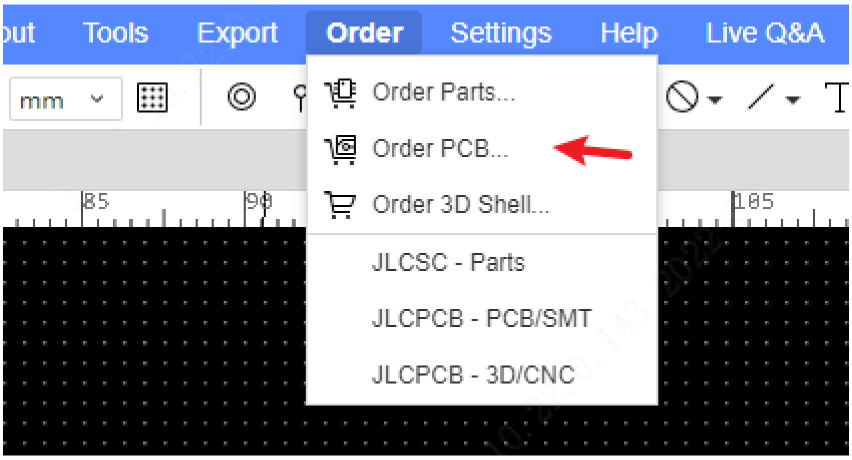


Click the pop-up button to open the single component under the mall page.

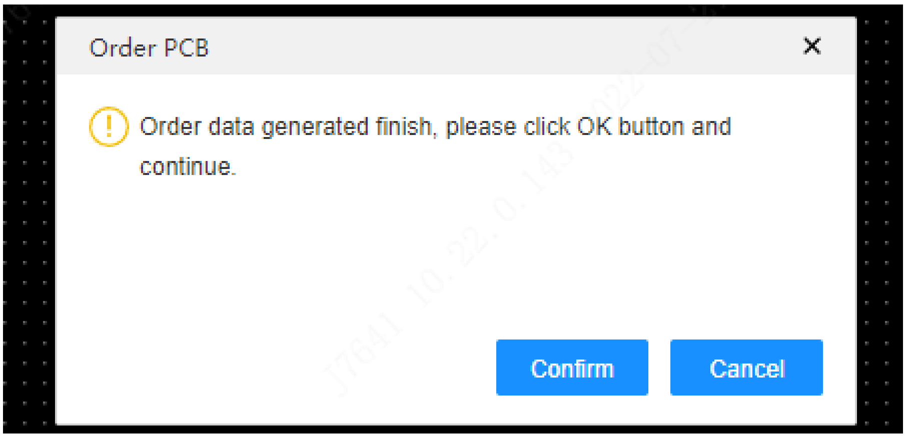


Order PCB

EasyEDA Pro will generate a Gerber file upload server based on the currently opened PCB.



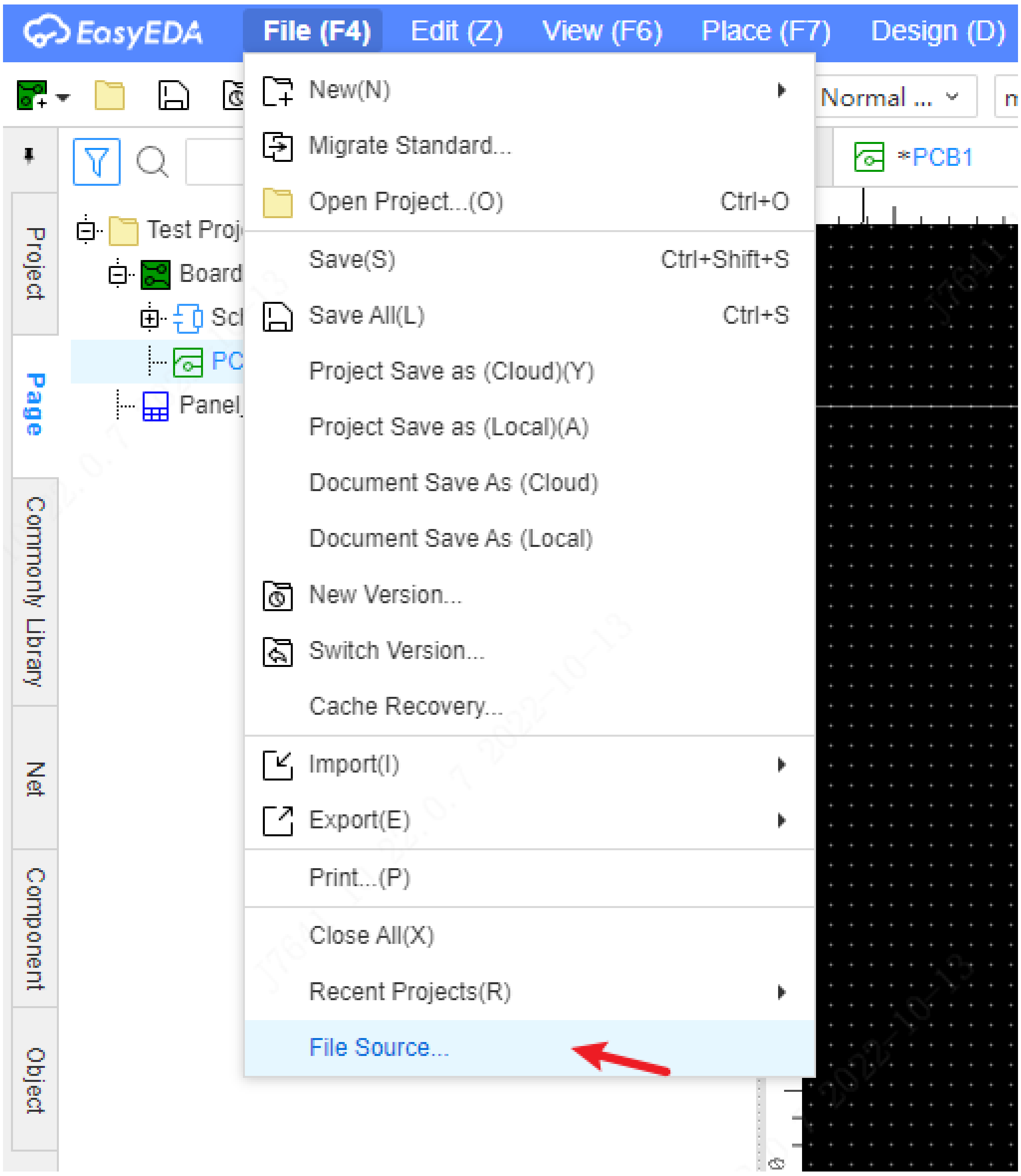
After the upload is successful, click the OK button in the pop-up window to open the order page of Jia Li Chuang to place an order.



Export File Source

EasyEDA Pro supports exporting or editing the source code of files.

Entry: Top Menu - File - File Source



After clicking Download, you can download the file source code of the project, and then you can open the file through a text editor or browser. You can also copy the code in the dialog to edit it in the editor.

After editing, click the Apply button to apply it to the canvas. If you do not know the corresponding relationship of the source code, please do not edit it, otherwise it will cause data errors and loss!

Note: If you export the source code of the schematic and PCB files separately, if they cannot be imported and opened by the editor, you need to export the entire project.

Drawing PCB without Schematic

In some small PCB fabrications, you may not need to draw schematics. EasyEDA Pro allows you to do PCB drawing directly.

1. New construction
2. New PCB
3. Select device Place in the bottom library
4. Click on the device pad to add a net
5. Add custom properties for each device to facilitate BOM export

Note: When drawing a PCB without a schematic diagram, only devices can be placed, symbols and footprints cannot be placed, only devices can be placed.

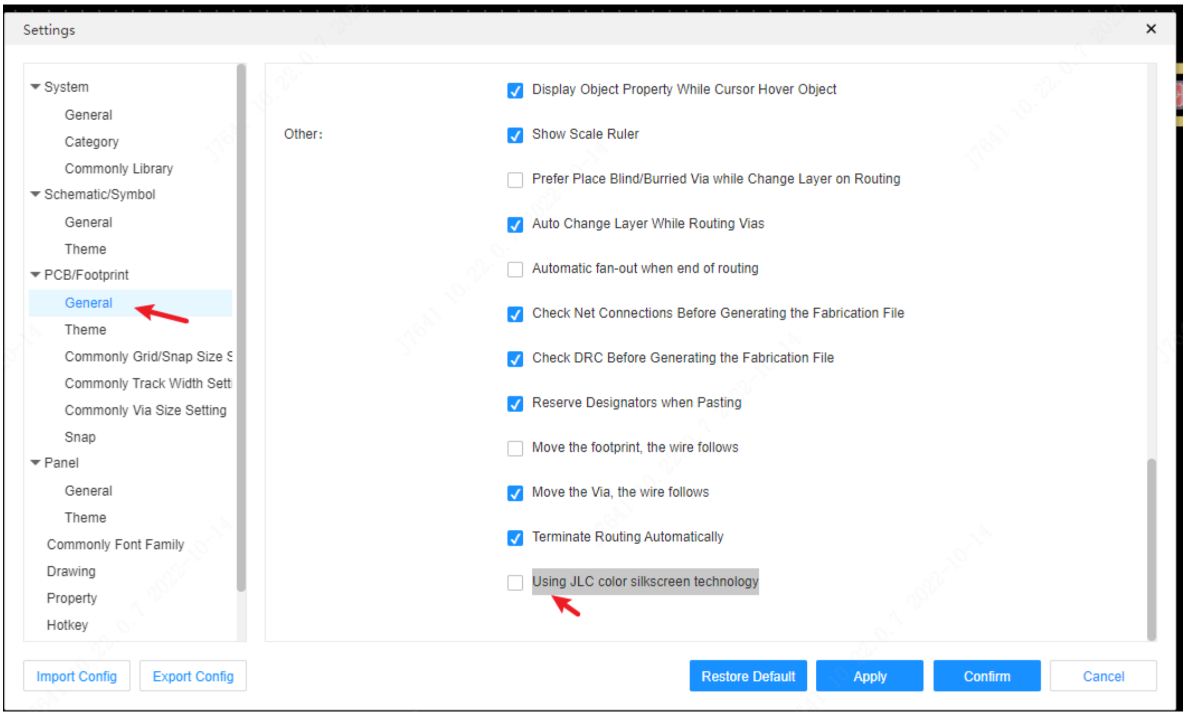
Draw Colorful silkscreen

Thanks to the JLCPCB process support, EasyEDA can be very convenient to support color silkscreen printing.

Support the import of color pictures, and character color Settings, export manufacturing files after the order in JLCPCB. Expected 2023 JLCPCB small batch open color silkscreen printing production.

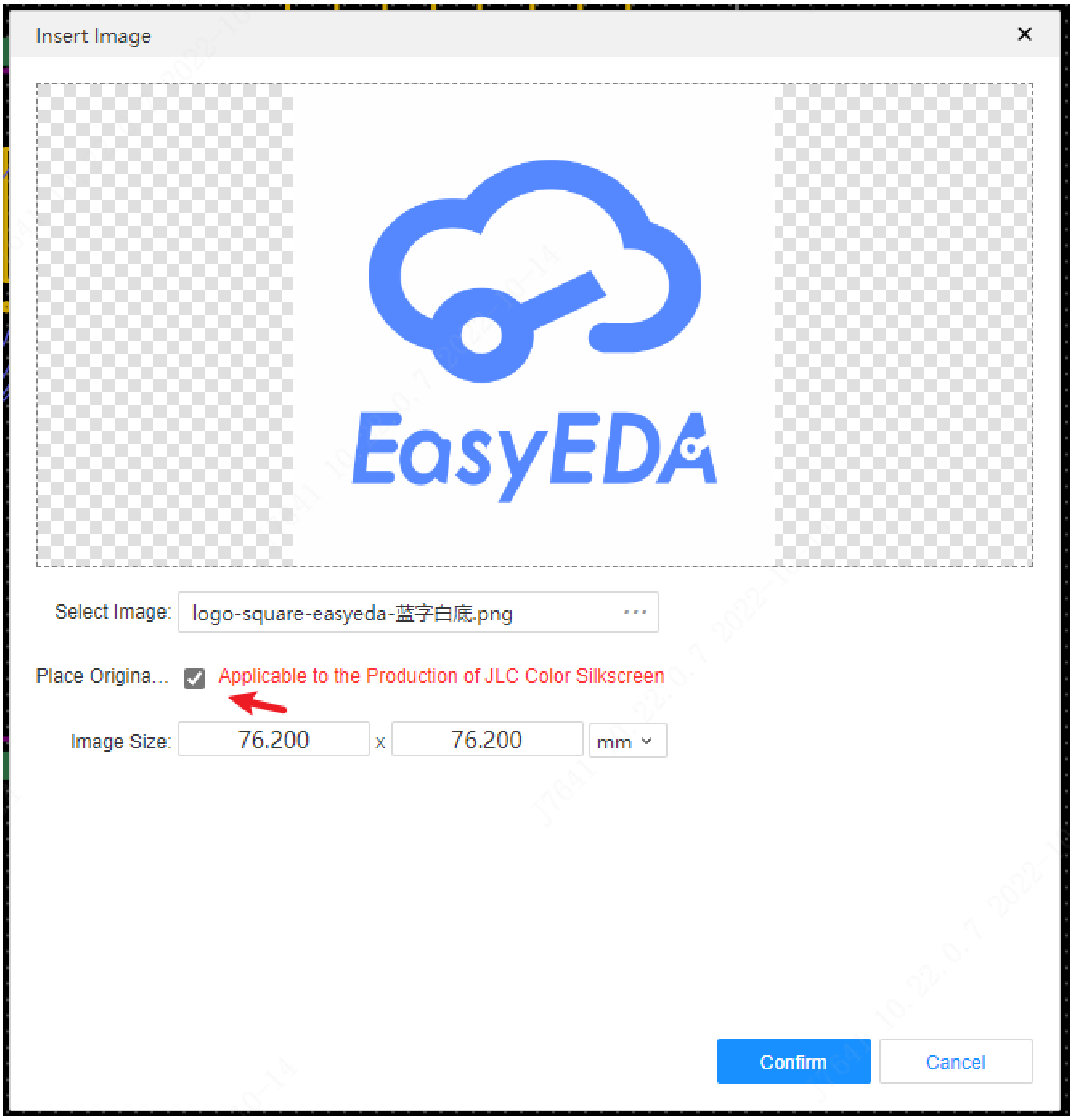
Turn on color silkscreen printing

You need to turn on the option of "Using JLC color silkscreen technology" in PCB Settings first.

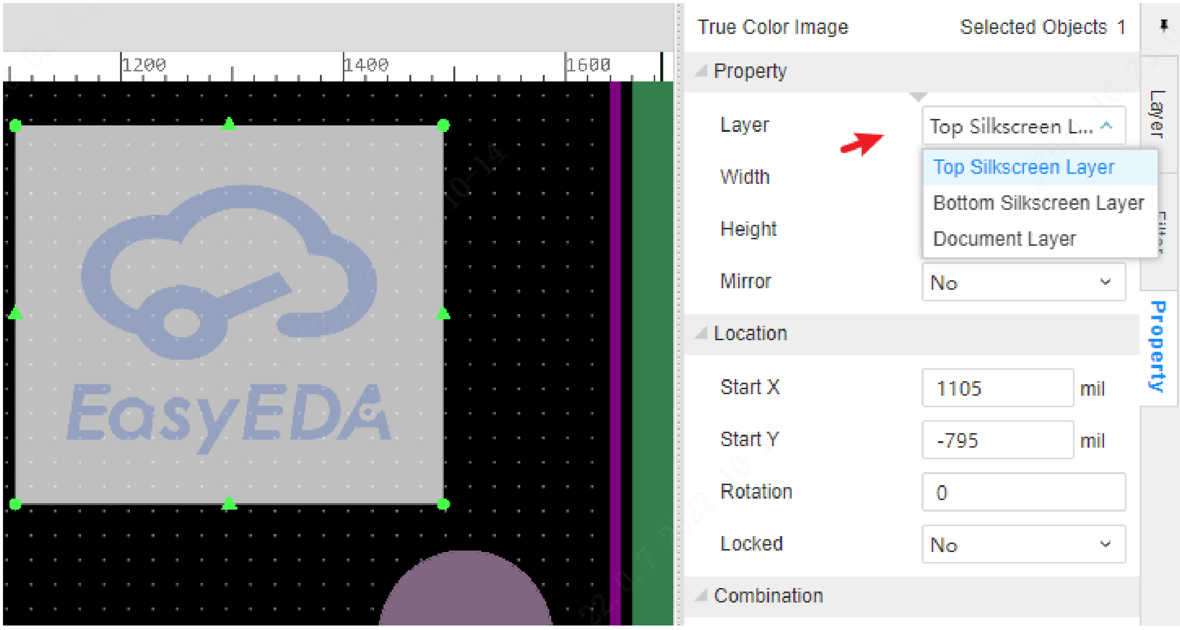


Import color images

Jia EasyEDA support to import color pictures, preview pictures, choose to place the original picture

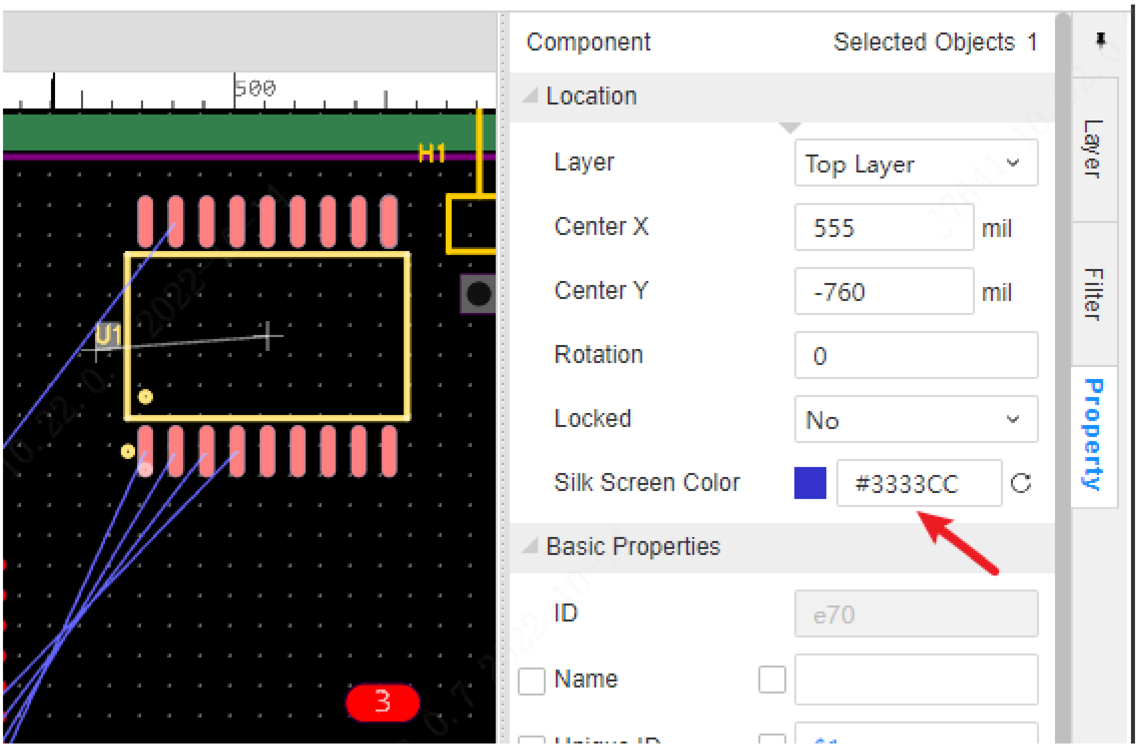


Will automatically be placed in the top silkscreen layer, you can also switch to the document layer or the bottom silkscreen layer.



Sets component silkscreen color

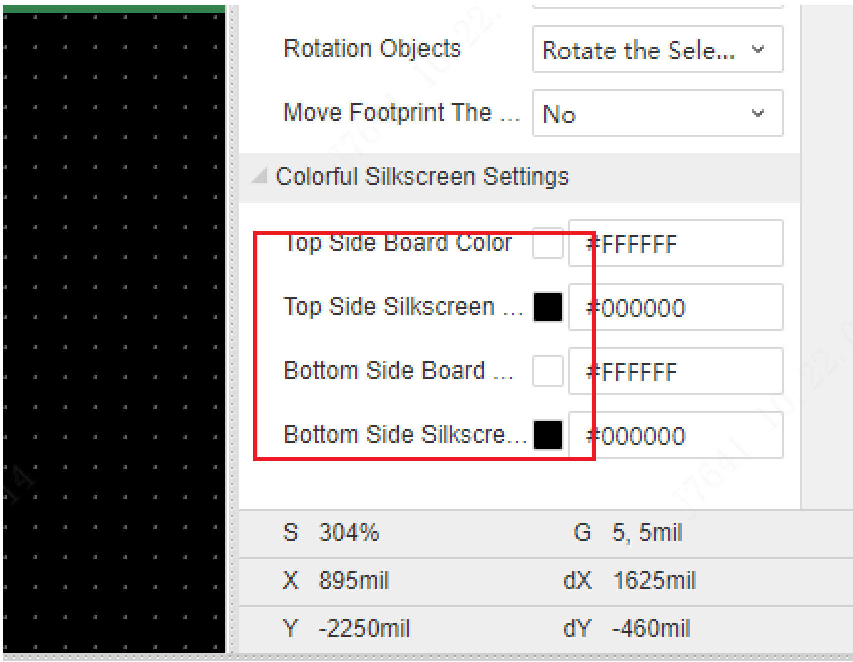
After selecting the component, you can set the silkscreen color in the right property panel



The internal line text, bit number and other attributes of the component will be produced using this color during manufacturing.

Set the silkscreen color

After clicking on the blank area of the canvas, you can see the color silkscreen Settings in the right property panel



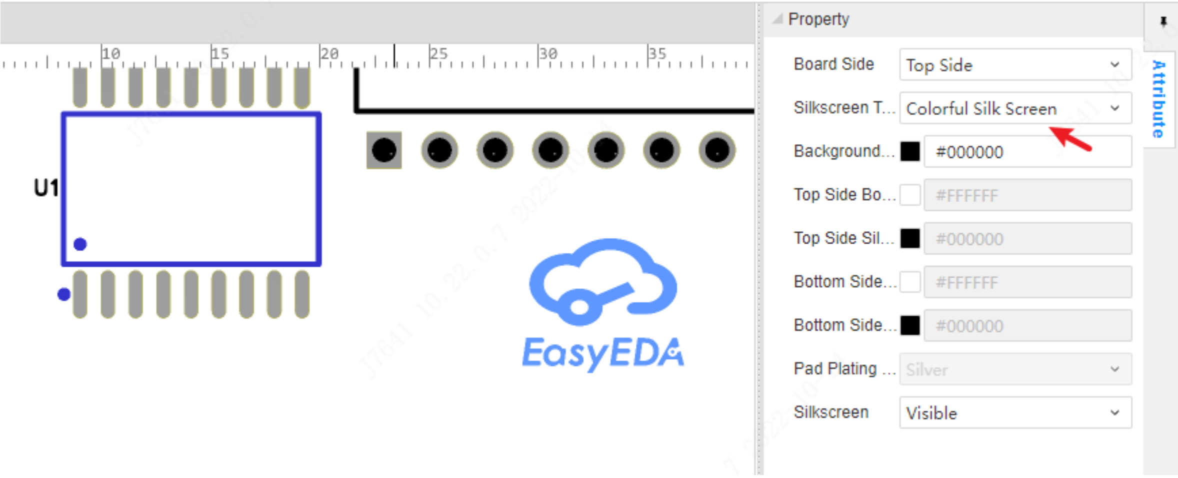
Top/bottom panel color: Default white. Because the board needs color printing, it needs to print a base color first, and the color setting is the color of the base color.

Top/BOTTOM silkscreen color: DEFAULT silkscreen color, such as bit number, component shape silkscreen color. If the component has a separate silkscreen color, the option will preferentially use a separate color.

Preview colorful silkscreen printing

In the top menu - View - 2D/3D Preview, you can preview color silkscreen printing and pictures.

In the right property panel of the preview interface, modify the silkscreen printing process to color silkscreen printing, and you can see the color printing effect.

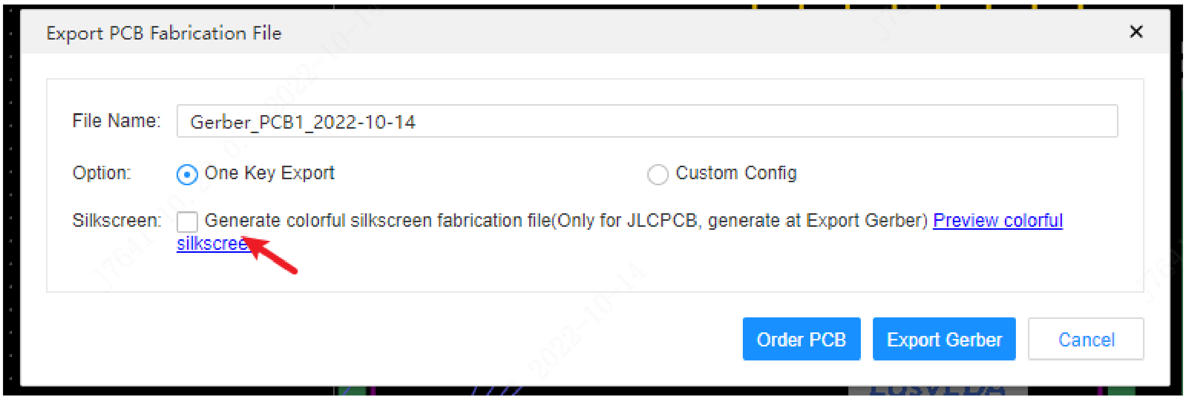


Export manufacturing files

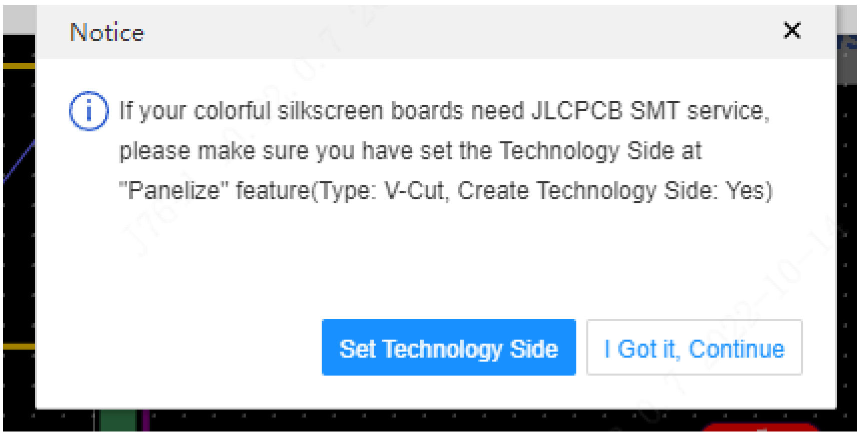
After setting and editing the color silkscreen printing, output the color silkscreen printing production file.

Entry: Top menu - Export - PCB Board file (Gerber)

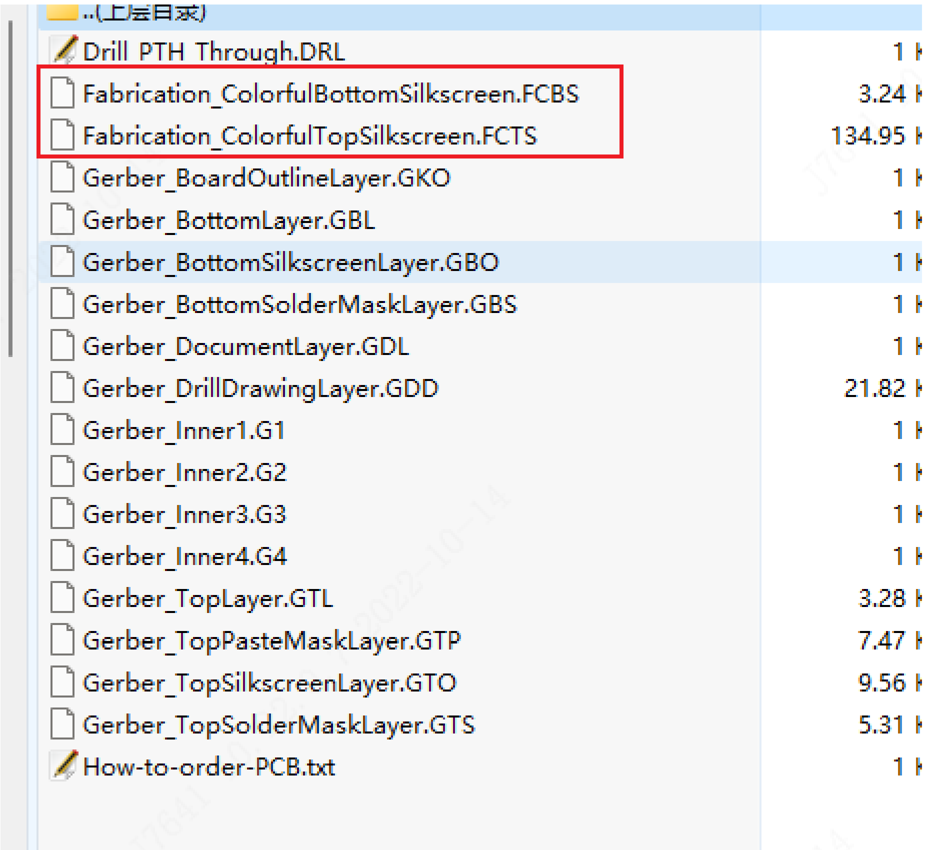
Select Export color silkscreen to export Gerber.



If you want to make SMT for your colorful PCB, you need to make a panelize option



After Gerber generated, there are two silkscreen fabrication files for JLCPCB to fabriaction in the Gerber zip file.



JLCPCB doesn't launch the colorful silkscreen ordering yet, need some time to finish the process.

Order Colorful Silkscreen PCB

(To be launched by JLCPCB color ordering function)

Panel Design

Panel Settings

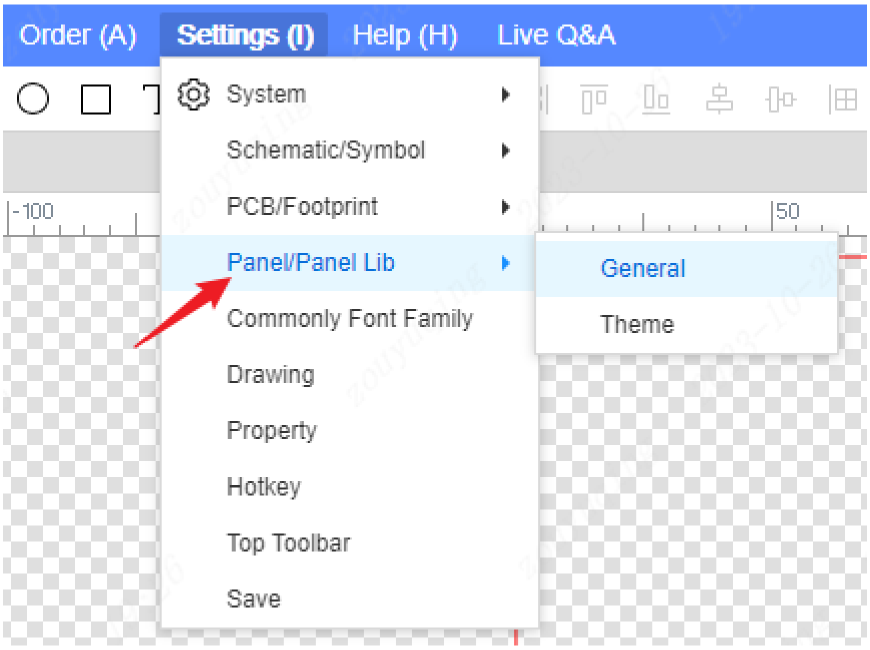
Panel Design

Panel Settings

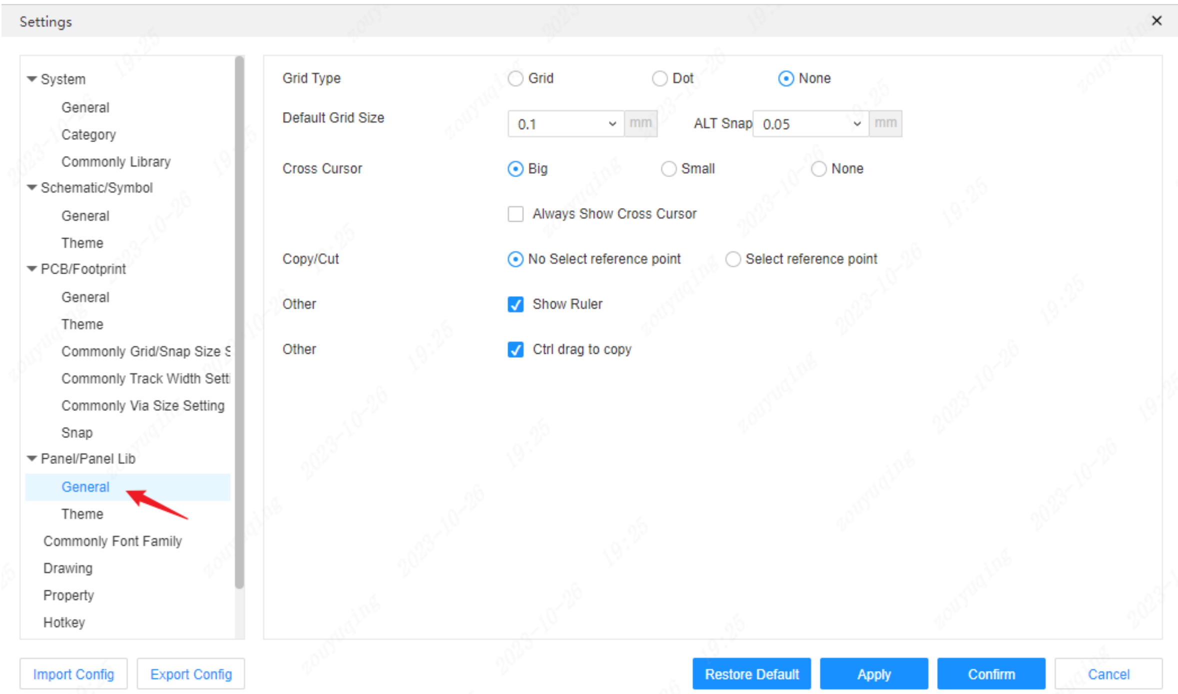
Panels can also support some general and thematic settings like schematics and PCBs .

Operation steps

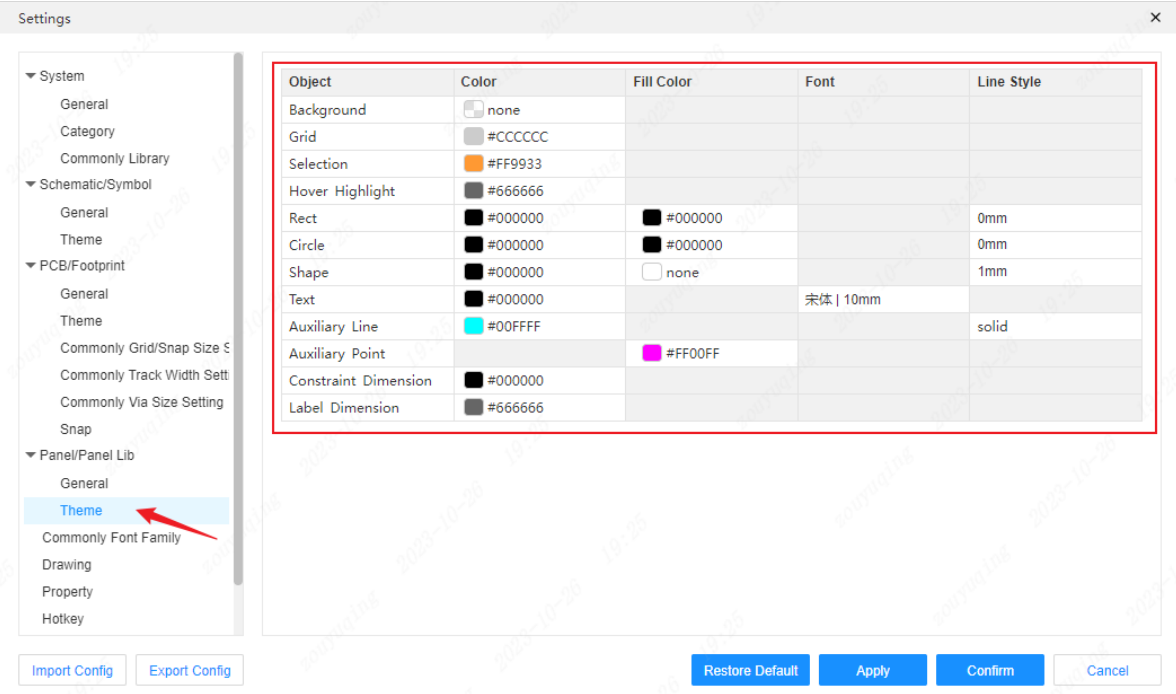
- Top menu - Settings - Panel/Panel Library, you can set some personal preferences for the panel/panel library.



Canvas mesh class, mesh size, ALT adsorption size can be set (this parameter is applied according to ALT adsorption during the drawing process), etc.



You can also set the theme, click to set font color, etc.



These settings will be applied to the current file after modification, and will also be applied when creating a new document next time.

New Panel

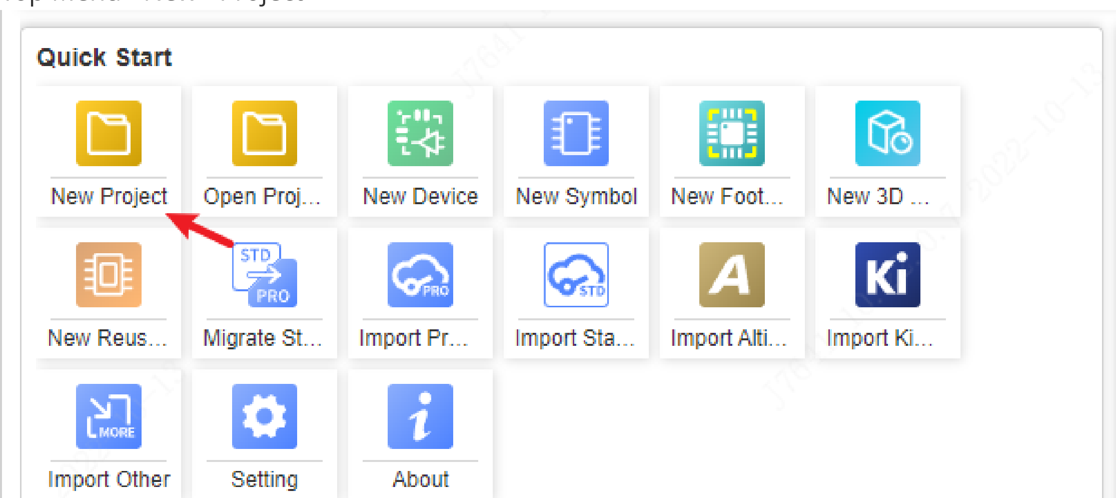
EasyEDA Pro supports a simple and easy-to-use panel design. It does not need to convert the format to process files, and does not need to communicate. It can be produced directly after drawing a picture, which saves the tedious communication process in the middle and avoids the problems caused by the conversion of formats and versions. , what you draw is what you get!

New operation

Notes for new creation: Panels cannot be newly created separately, and a new project needs to be created before creating panel design drawings.

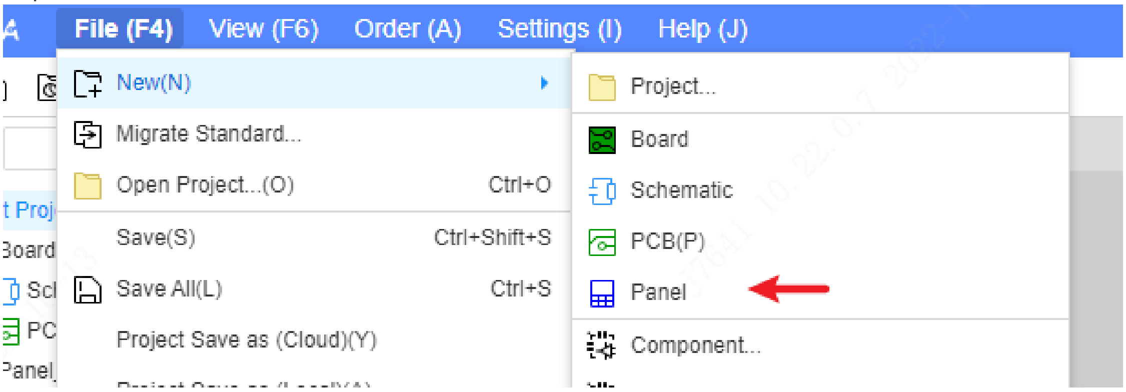
1. First create a new project in the editing area and fill in the project name to manage the panel file

- Top Menu - New - Project



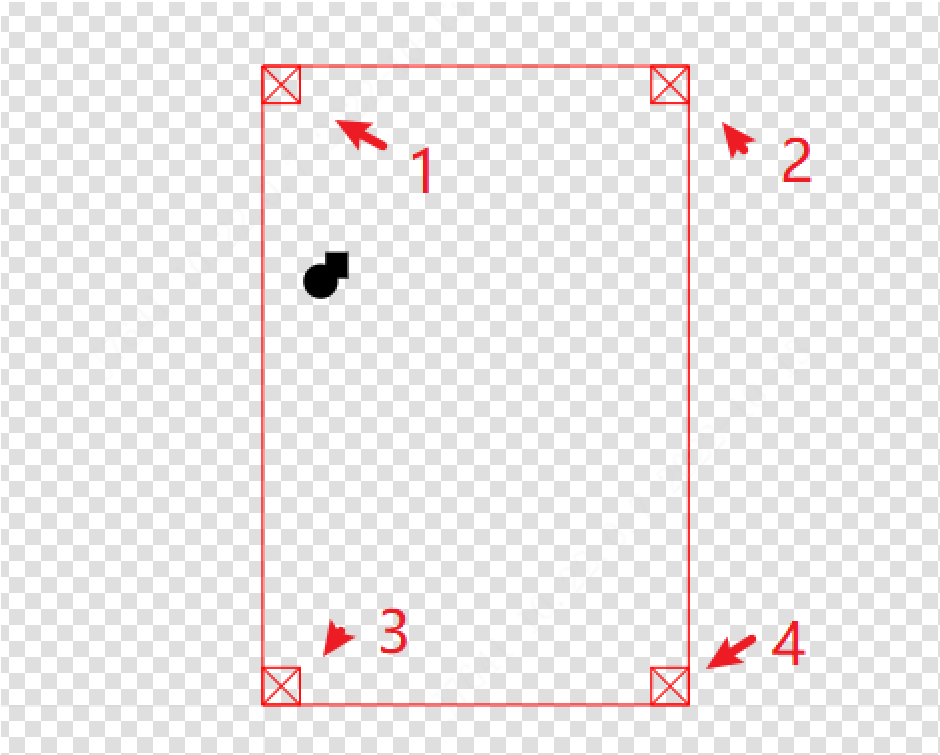
2. After the new project is completed, create a new panel

- Top Menu - New - Panel

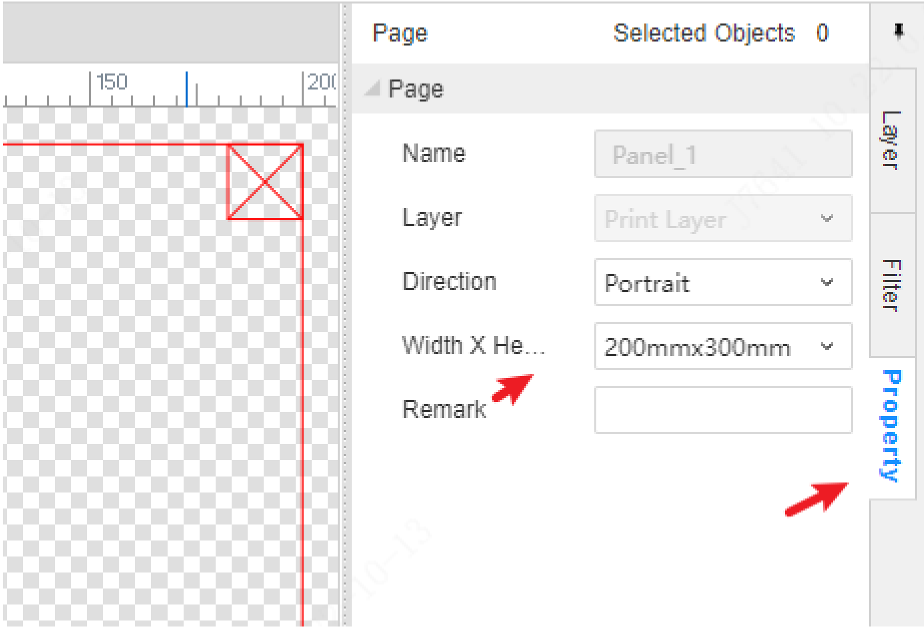


Design drawing description

- **Positioning point**: There is a square frame in the four corners (points 1, 2, 3, and 4) of the red wire frame on the design drawing, which is the positioning point of the sheet, which is used for positioning during production, cannot be modified by default.



- **Sheet frame**: The red rectangular frame in the design is the size of the panel printing material of LCSC Mall. The default size is: 200x300mm



New Panel Library

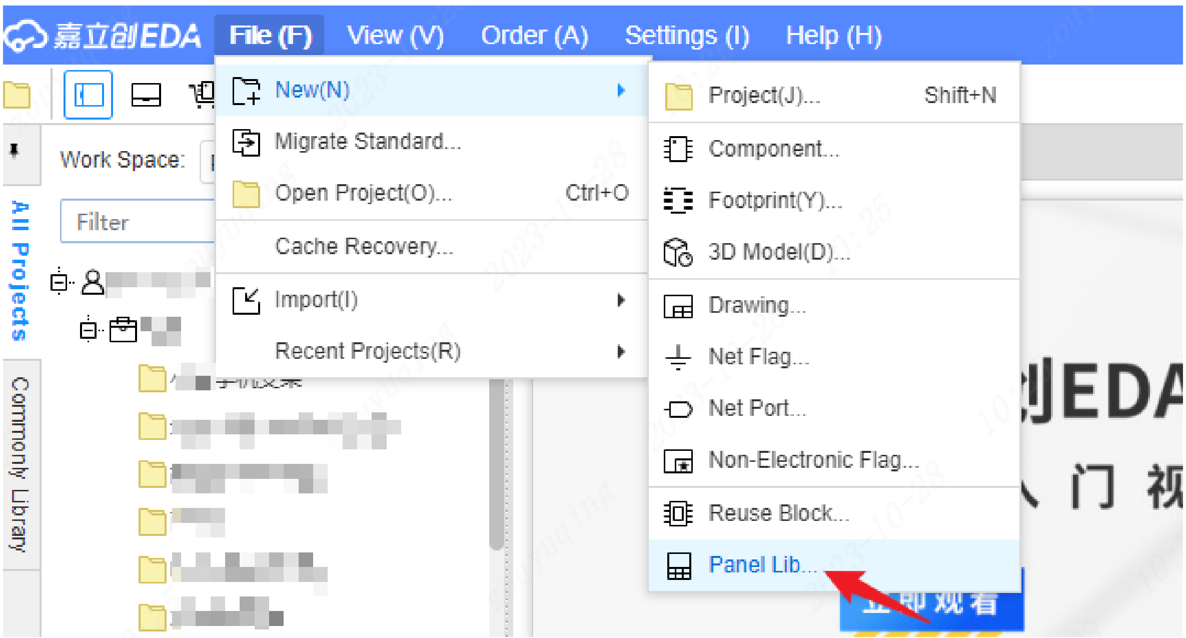
Panel Library

The drawing of the panel library is relatively free, without specific design rules constraints. You can draw any size of any layer element at any position, providing the same drawing tools as the panel. The panel library cannot be directly exported for production, but it can be placed on the panel canvas. After placement, the panel library is not maintained as a whole, which is convenient for further adjustment.

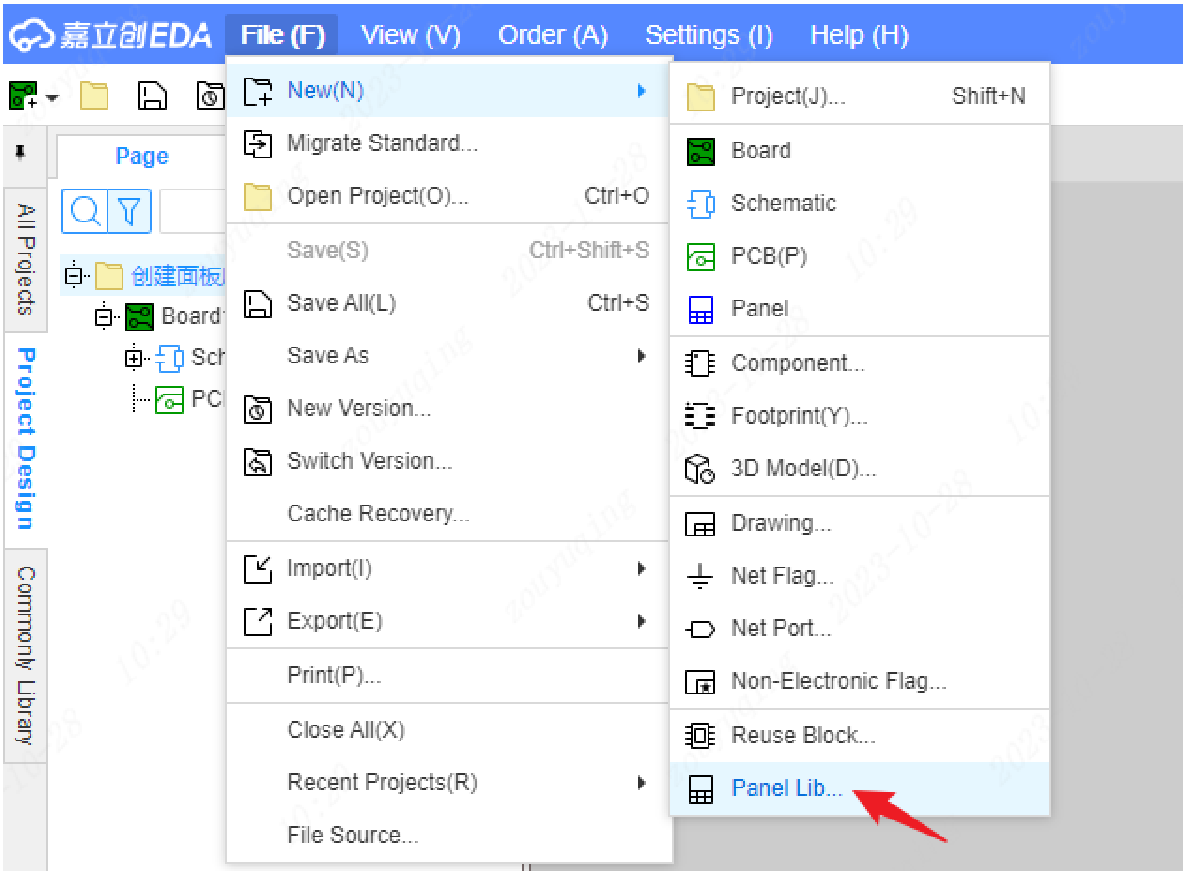
New entrance

Note: Only the professional edition editor supports the creation of panel libraries, which can be created separately.

- **When no project is open**, top menu - File - New - Panel Library



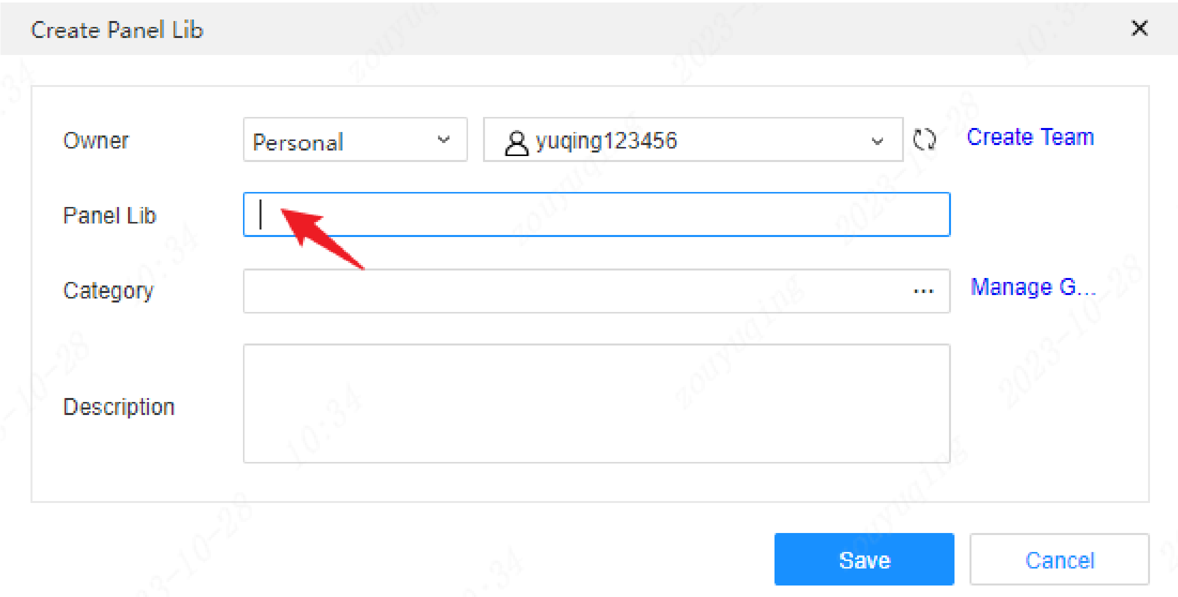
- **When a project has been opened**, the top menu - File - New - Panel Library



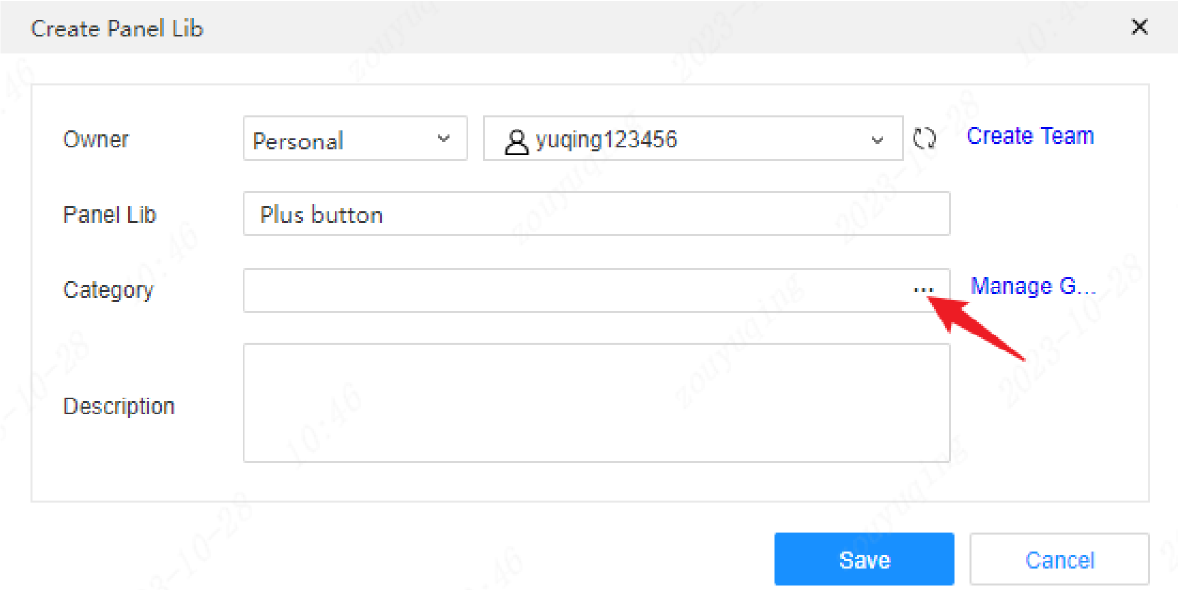
Operation steps

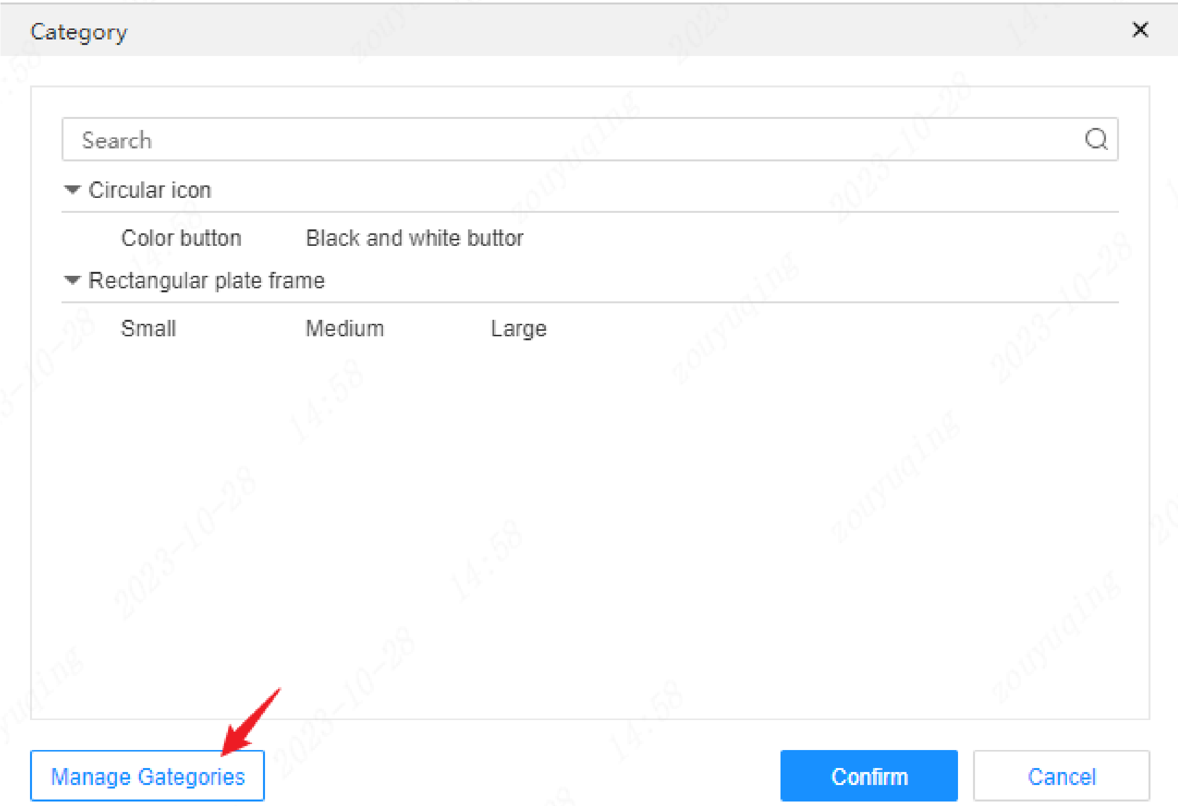
Create a new pop-up window

After operating the New Panel Library menu item, a New pop-up window will pop up.
The name of the panel library needs to be customized, cannot be empty, and the length is limited to 1~ 128 characters. (necessary operation)

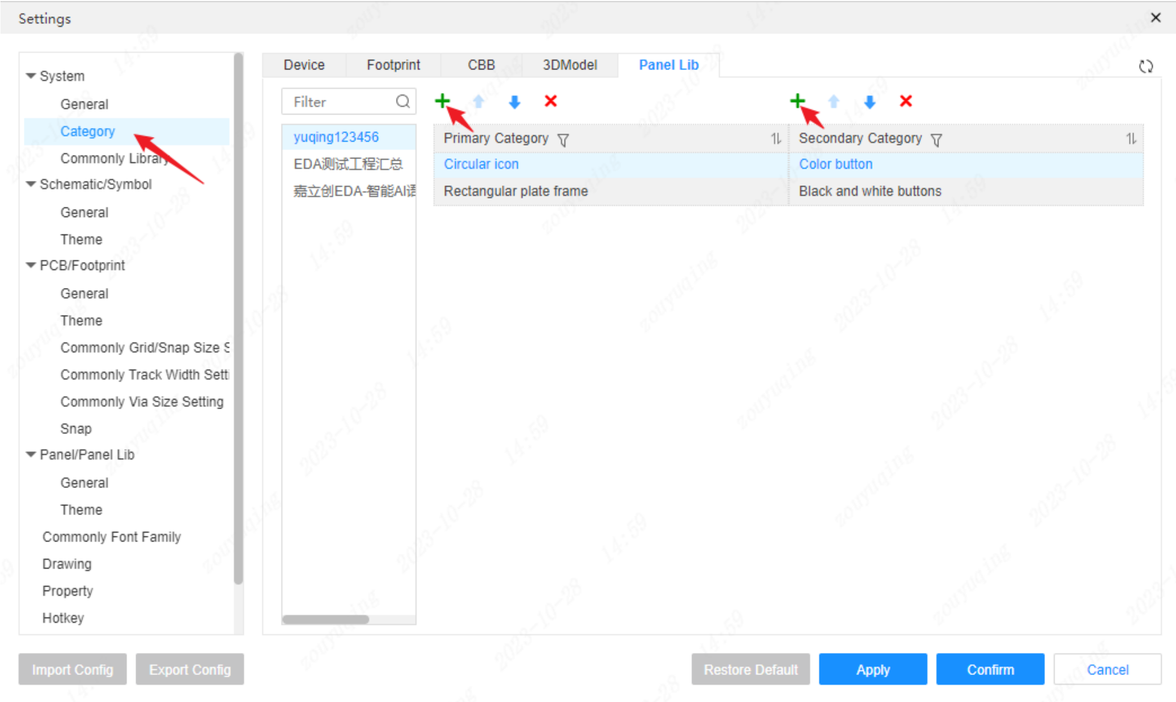


Category management can be performed when creating a panel library. Click "..." on the right side of the category bar to pop up the category selection pop-up window to select a category for the panel library to be created.

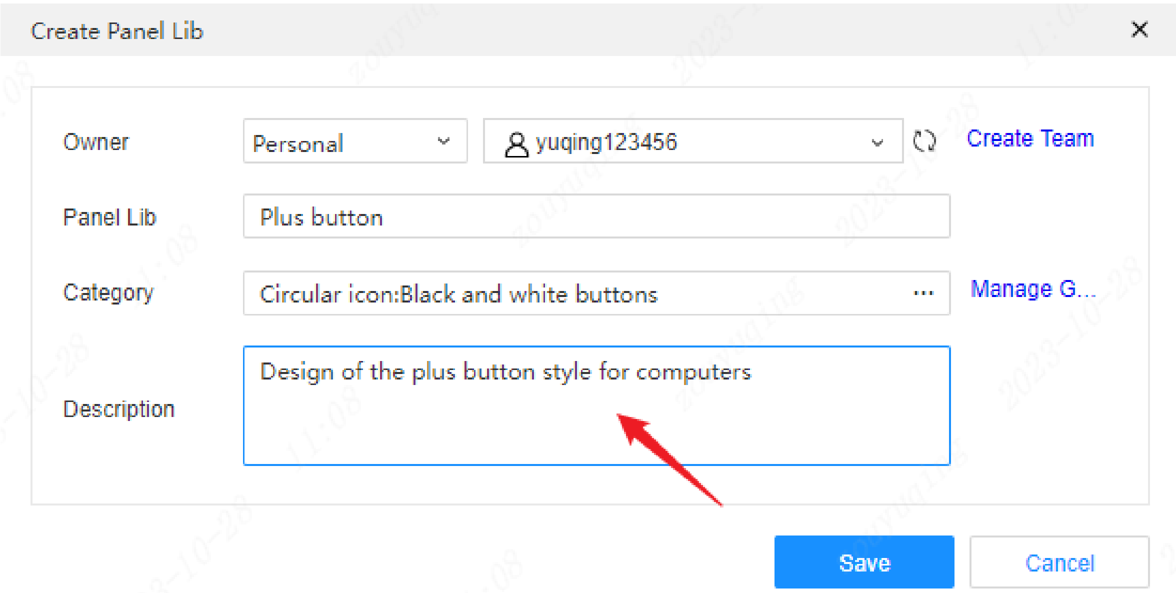




You can also click Manage Category in the New Panel Library pop-up window or the Category Selection pop-up window to bring up the Category Settings pop-up window. Click the green plus sign to add a new category.



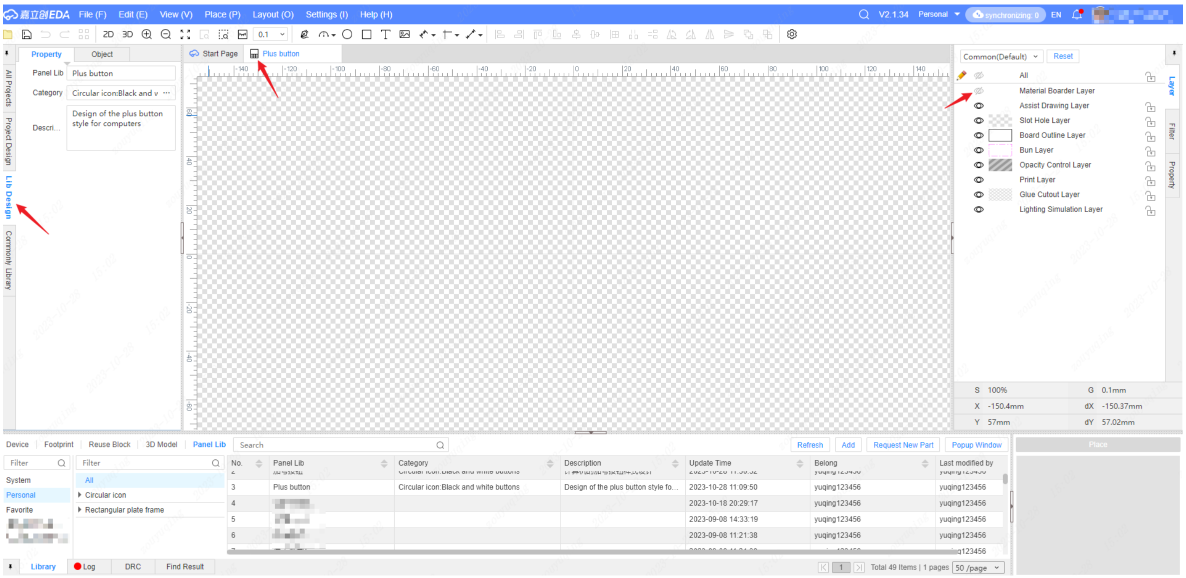
You can also add some descriptions to the panel library.



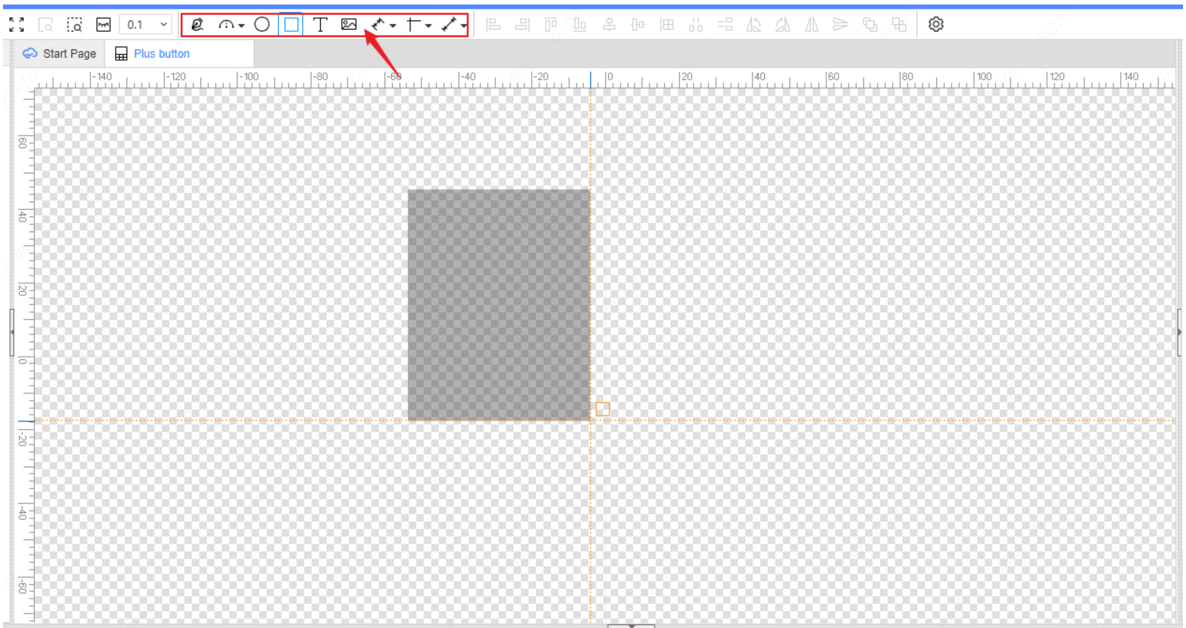
Finally, click the Save button to complete the creation of the panel library, which will automatically open the editing page of the newly created panel library.

Panel library edit page

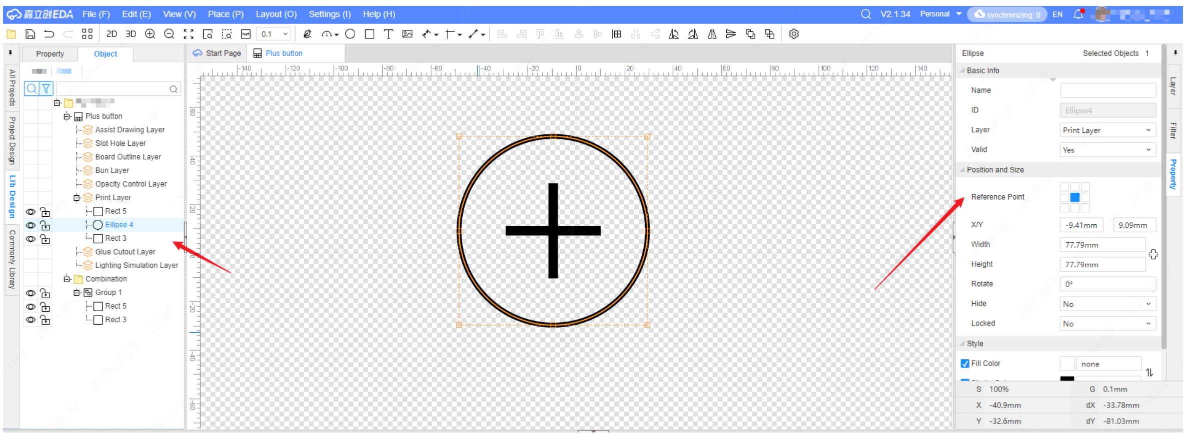
The material boundary layer is hidden by default in the panel library editing page. The left panel opens the library design by default, displaying the custom information when creating the panel library, and supports readjustment.



The toolbar of the panel library is basically the same as that of the panel, and the drawing of elements in the panel library is the same as that in the panel.

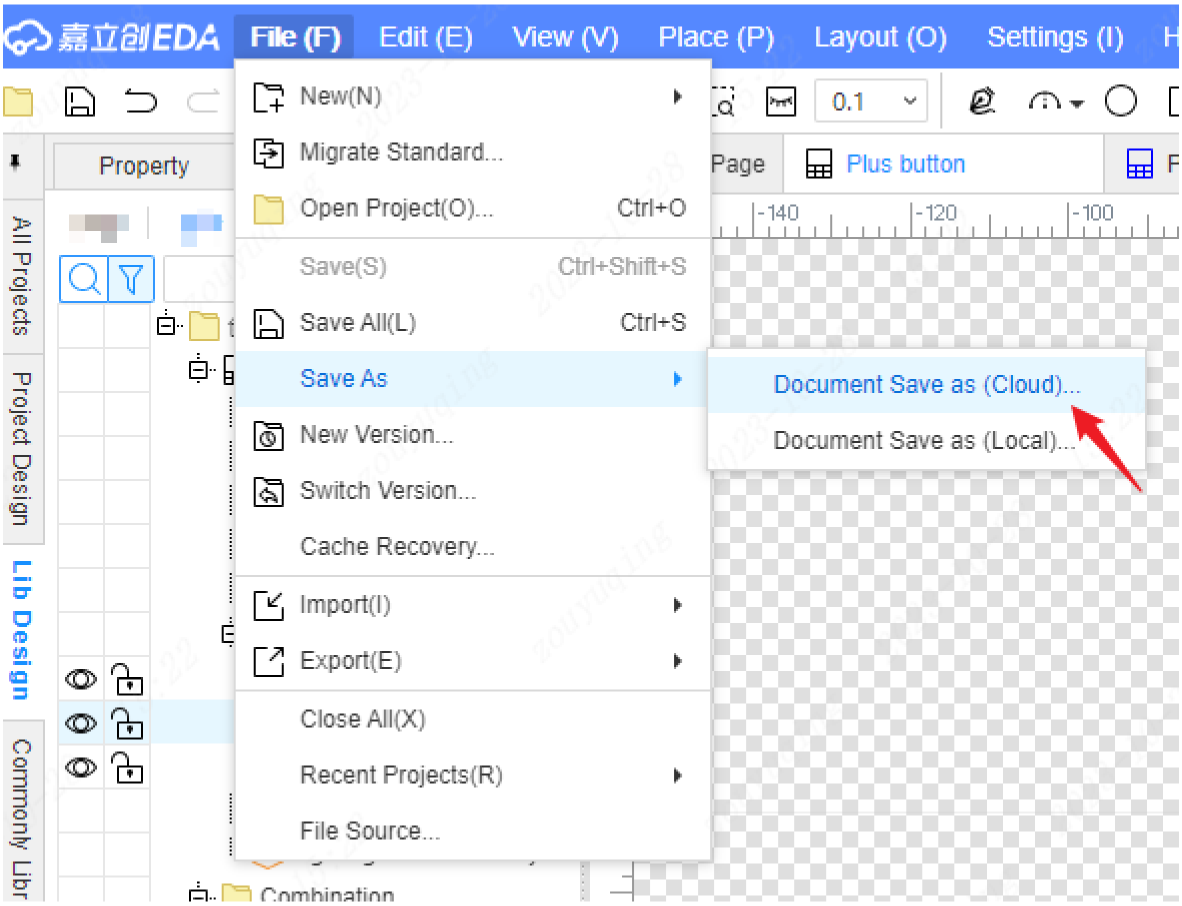


The left object tree, right property panel, etc. remain consistent with the panel editor.

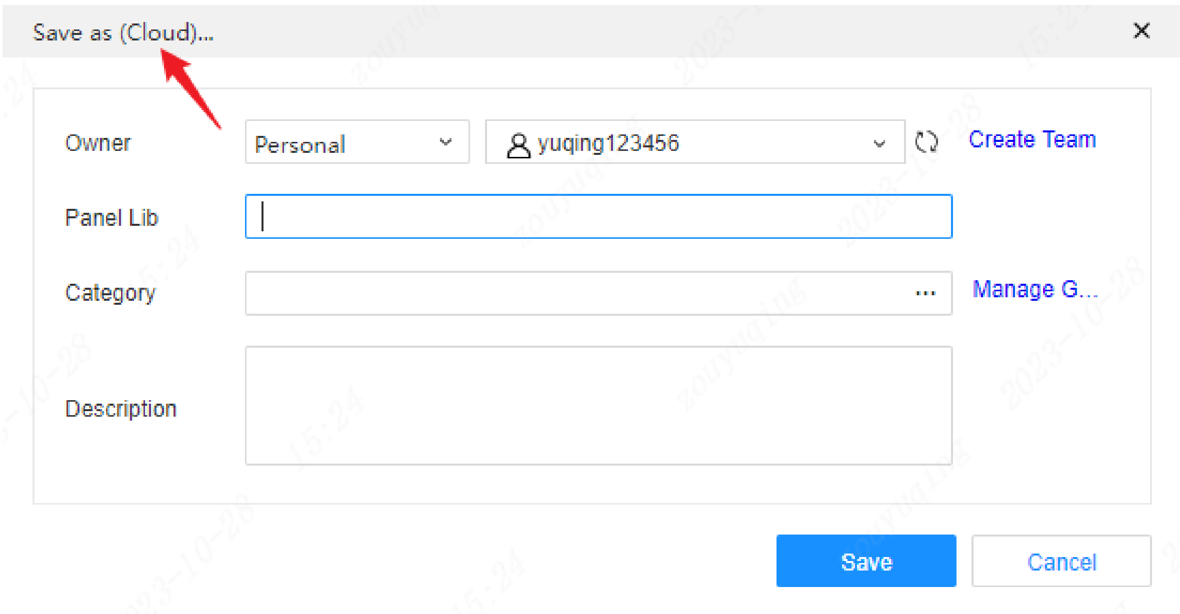


Export panel library

- Top menu - File - Save As - Document Save As (Cloud)...

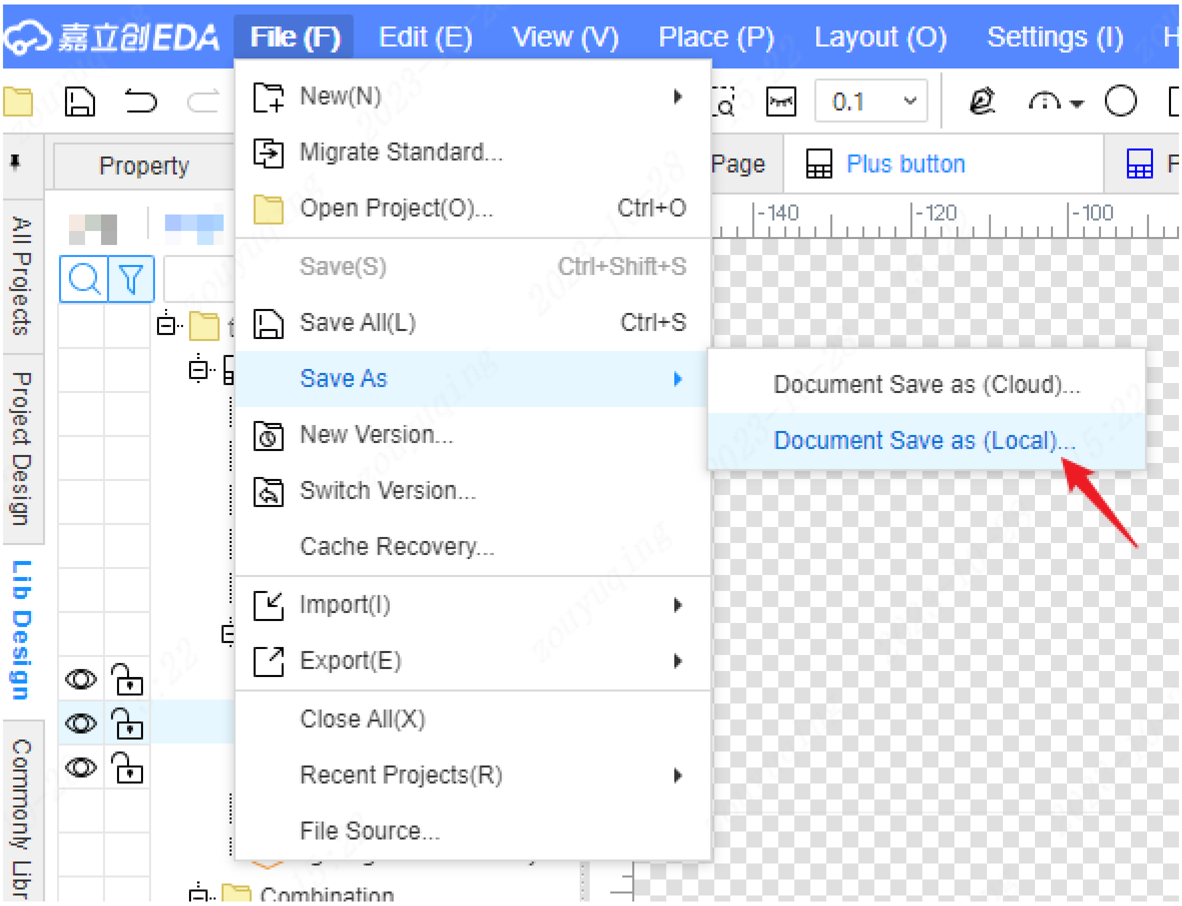


Saving as cloud is equivalent to creating a new panel library. You need to customize the name, category, description and other information of the panel library, and the content in the panel library will be directly copied to the newly created panel library.



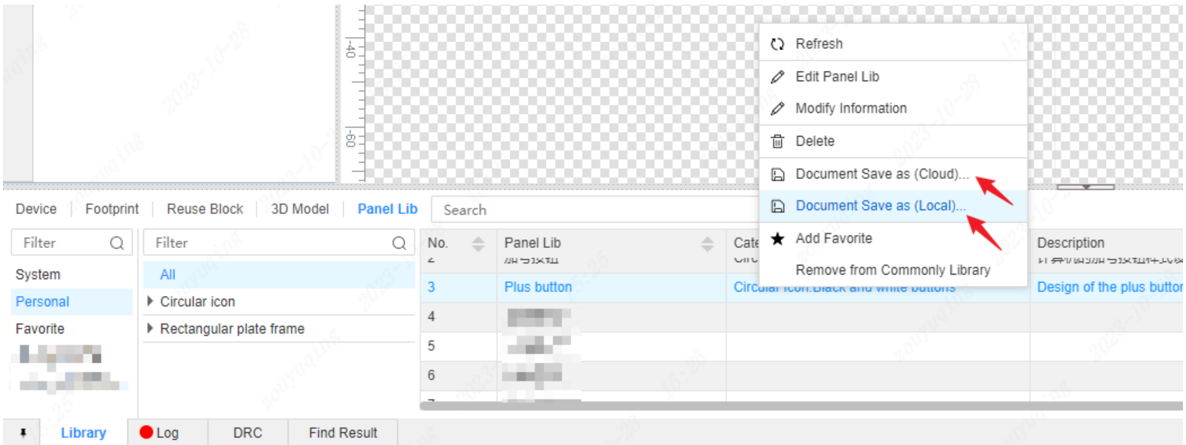
- Top menu - File - Save As - Document Save As (Local)...

The file name extension of the panel library exported to the local is .epanl



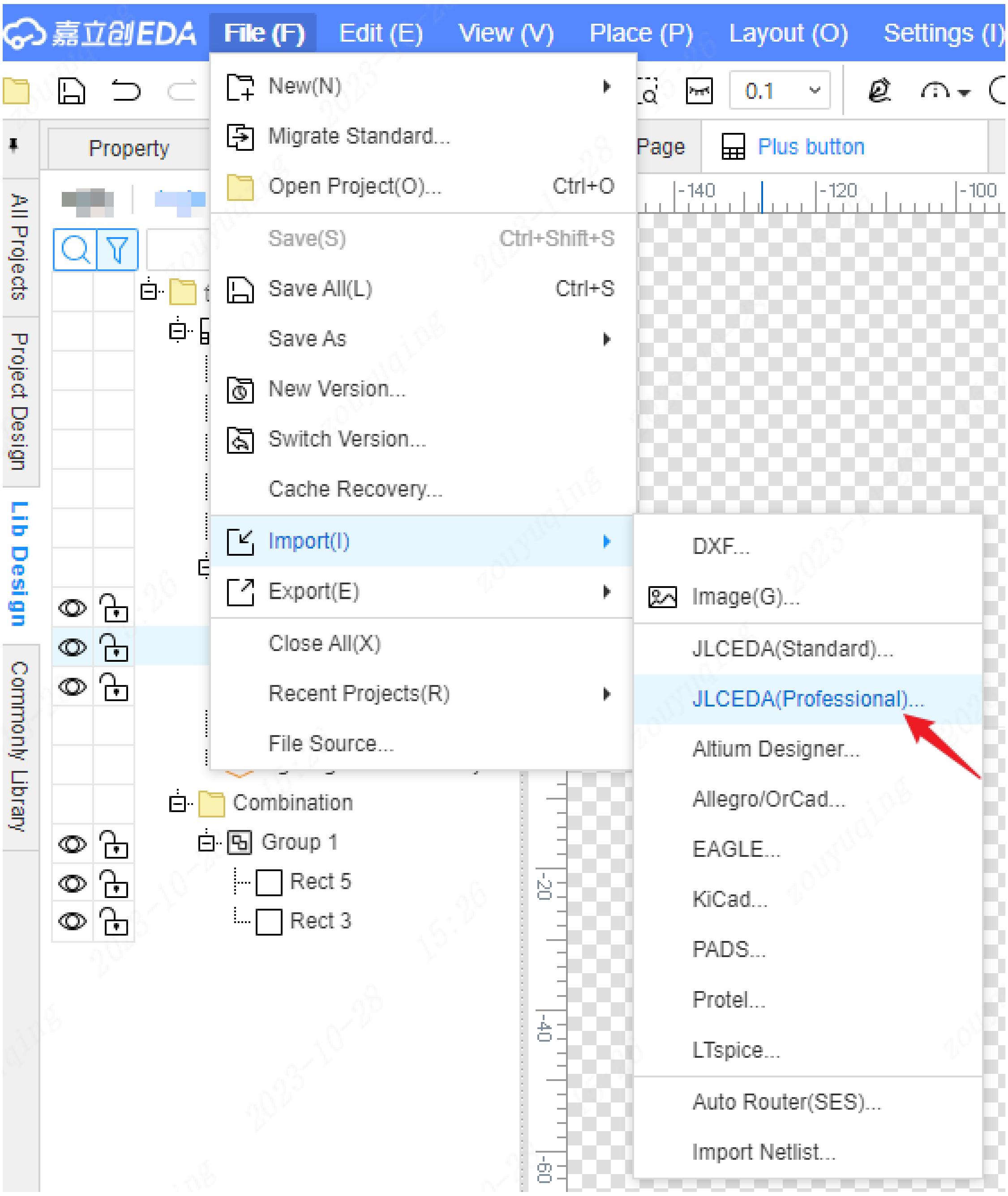
- Bottom Library - Panel Library - Right-click after selecting the specific panel library

In the right-click menu, you can also select the "Save Document As (Cloud)..." or "Save Document As (Local)..." menu items to export the panel library.

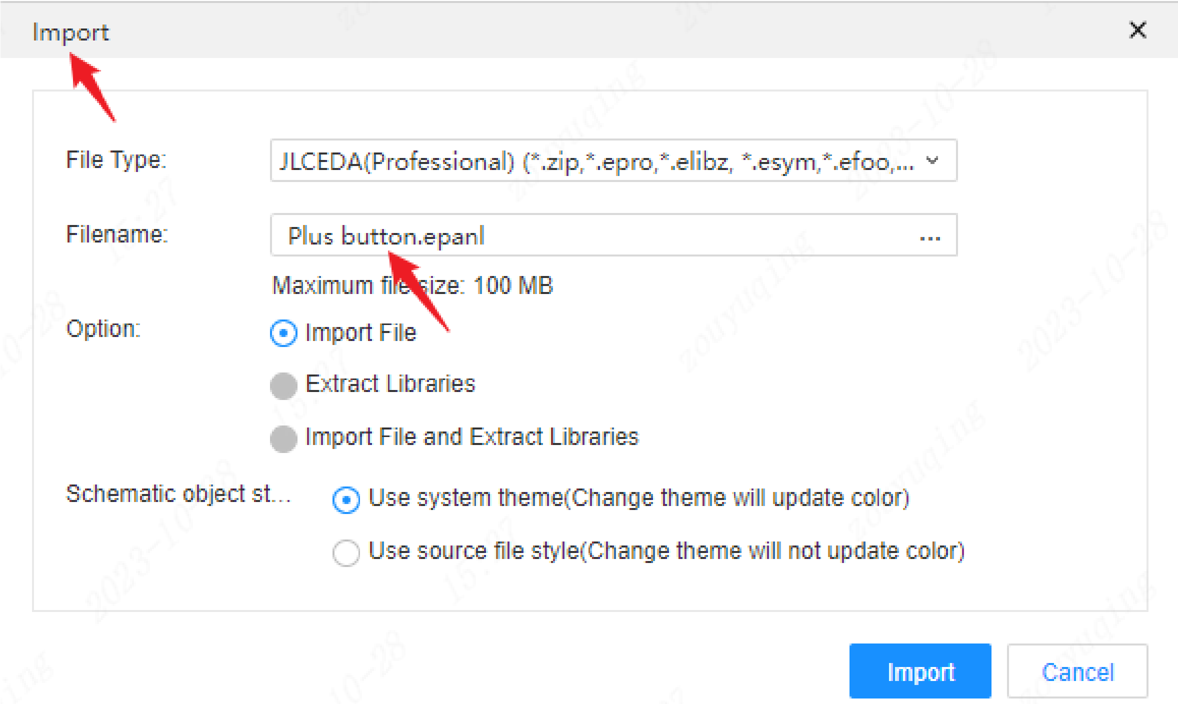


Import panel library

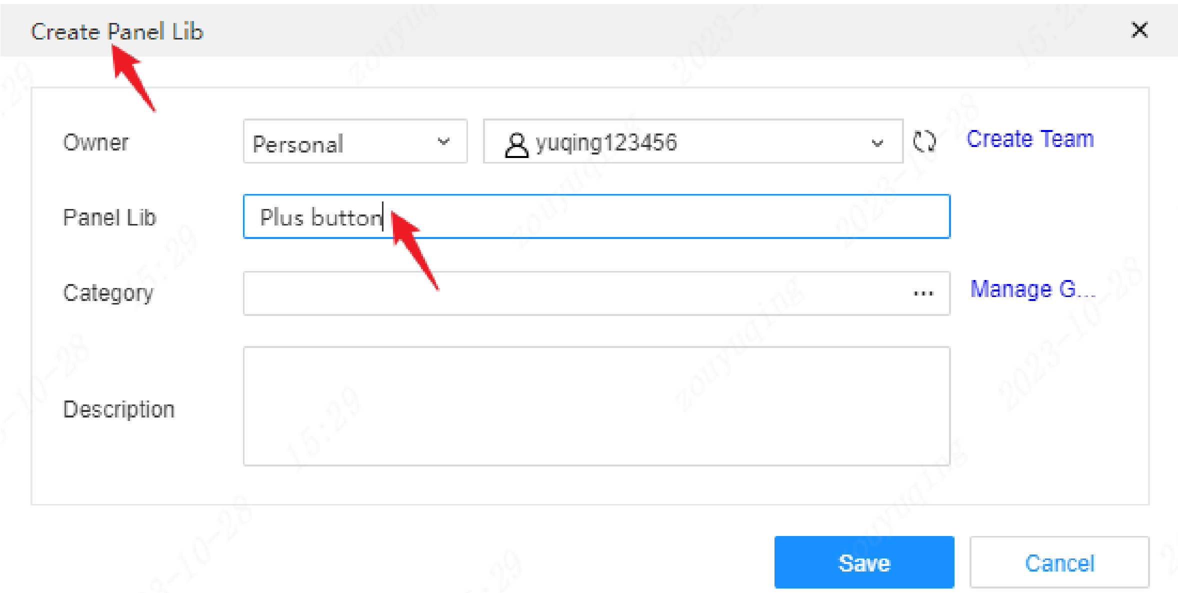
- Top Menu - File - Import - Jia Li Chuang EDA (Professional Edition)...



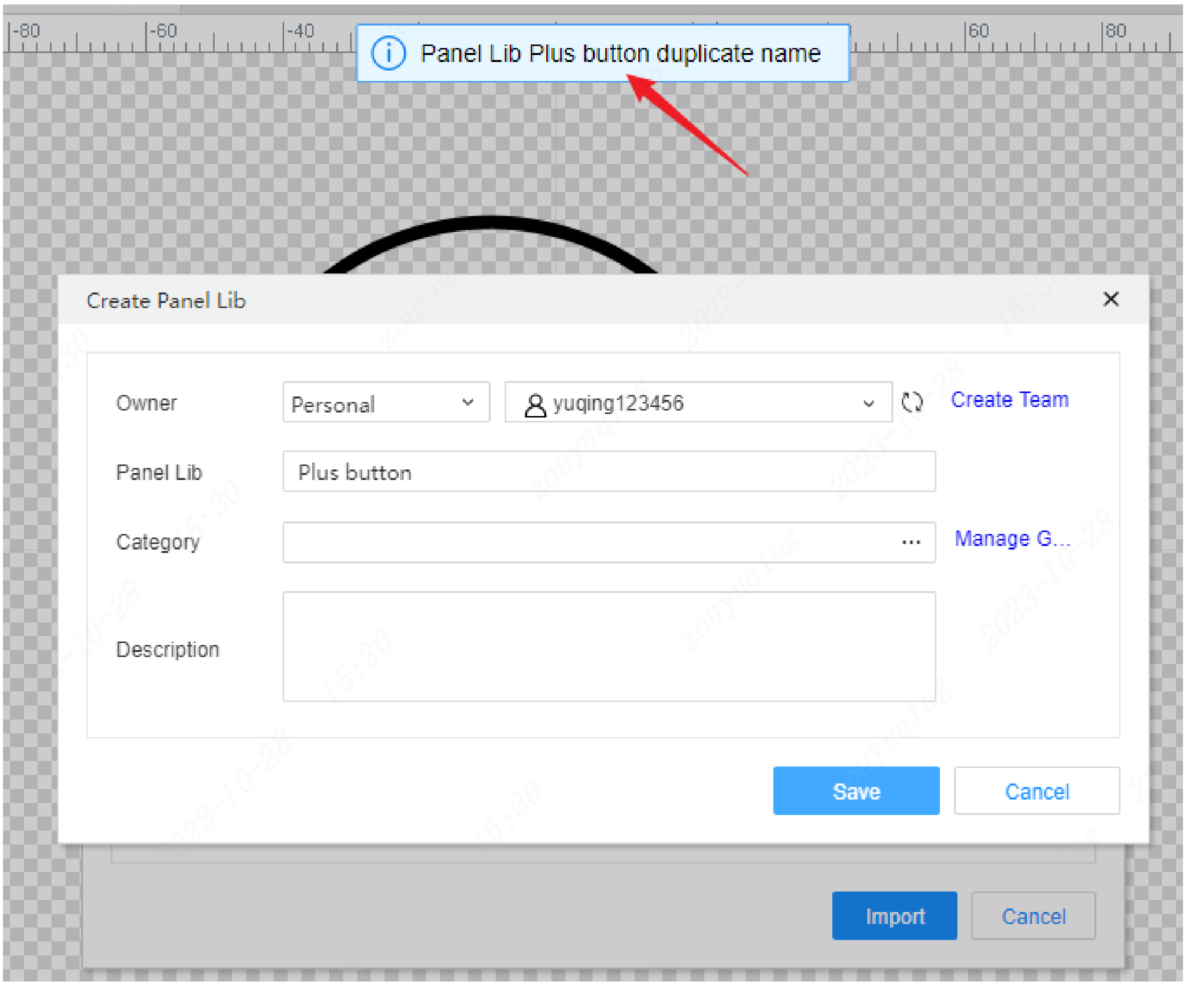
After selecting the panel library file with the suffix .epanl in the pop-up selection file pop-up window, the import confirmation information pop-up window will pop up.



After clicking the Import button in the Import pop-up window, the New Panel Library pop-up window will pop up. The name of the panel library will be automatically filled in with the original file name. After completing the custom editing of the new panel library information, click the Save button to complete the import of the panel library.



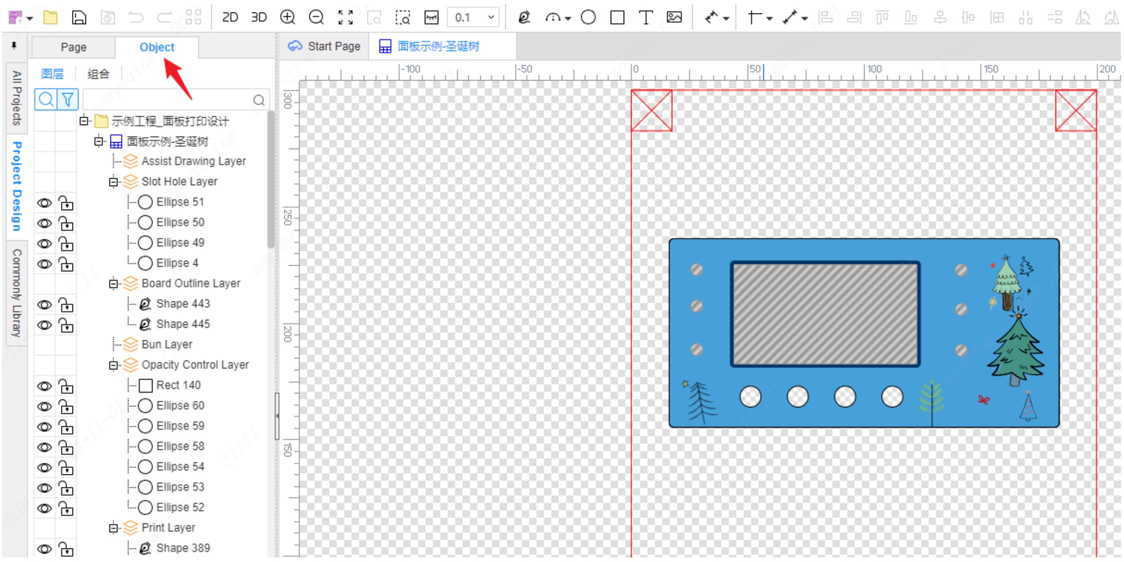
Note: Duplicate panel library names are not allowed. Renaming can be avoided by customizing methods such as adding serial numbers.



Left Panel - Object

Left Panel - Object

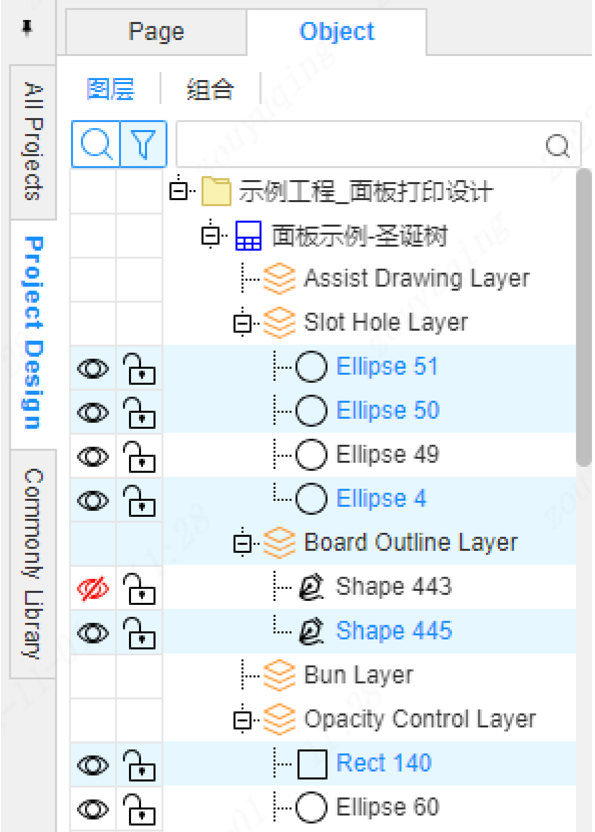
- The Object tab on the left panel can view all the objects in the current panel file, and click to select them.



Select

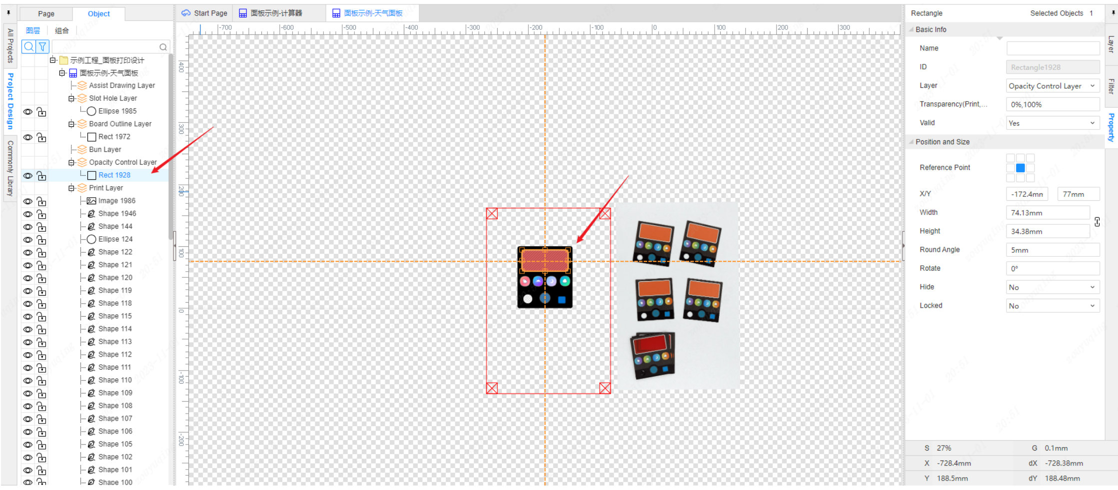
Ctrl, Shift multiple selection

- Support multiple selection of Ctrl and Shift in the left object tree.

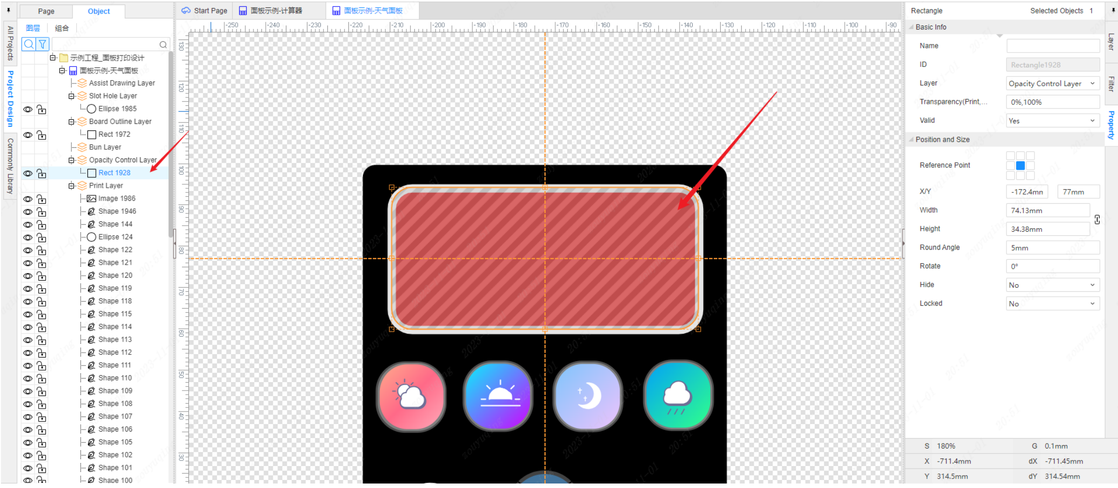


Click, Double Click

- Click on the left to select an object, the object is highlighted on the canvas, and the canvas is moved to center the object on the canvas.
  (Canvas will not be automatically scaled, only positioned and moved)

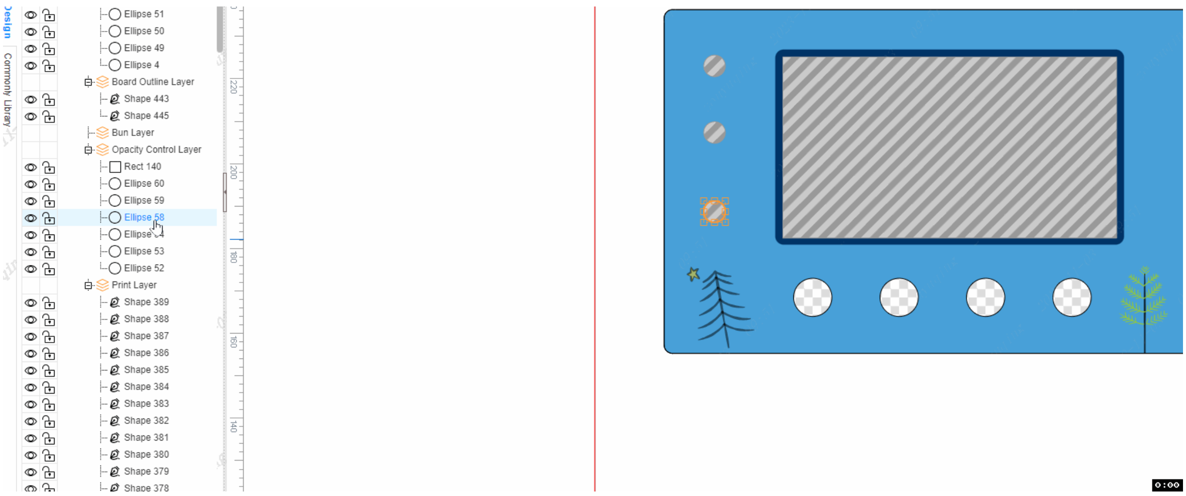


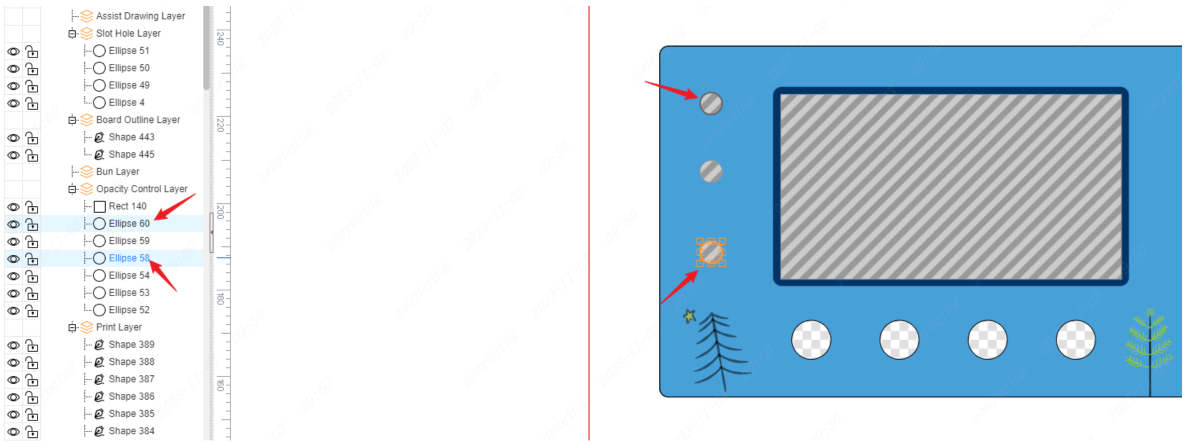
- Double-clicking on an object on the left will automatically zoom the canvas (to make the object appear the right size), and move the canvas to center the object on the canvas.



Arrow keys change the suspension term

- Support up, down, left, right arrow keys in the left object tree to change the suspended item, and press Enter to change the selected item.
  The suspended highlight item in the left object tree corresponds to the element suspended highlight on the canvas.

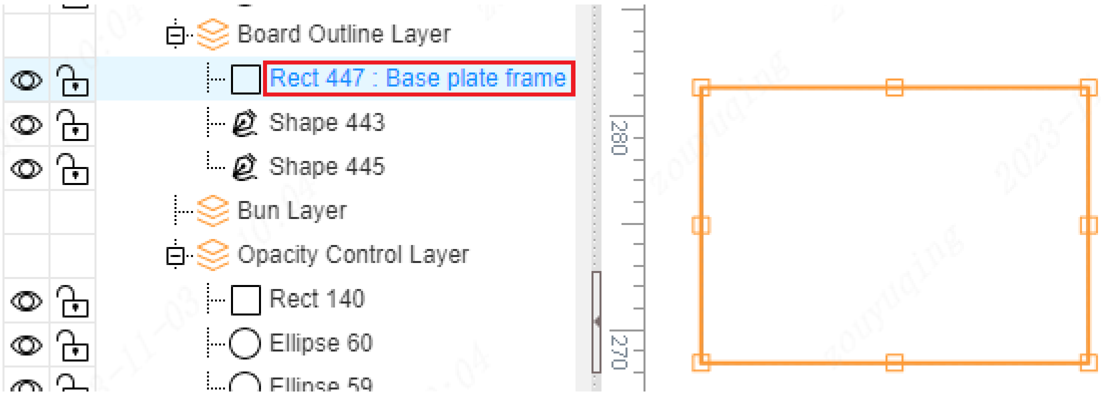




Display

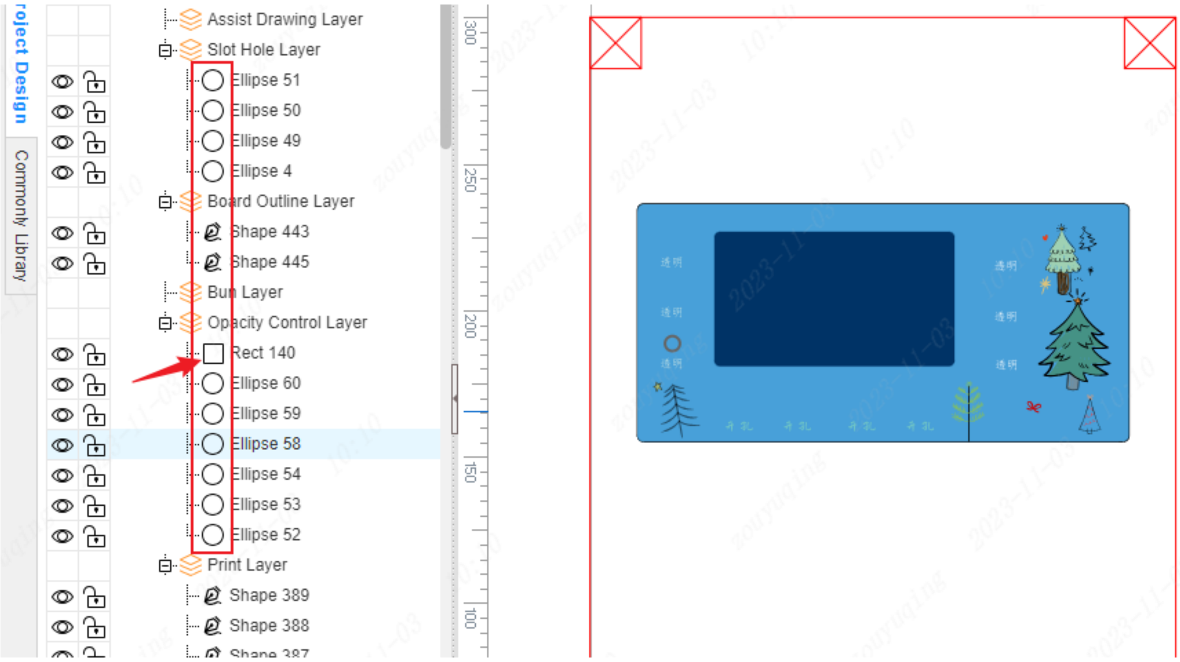
Object ID Name

- The composition of the object ID name in the left object tree: primitive attribute type name + sequence number: primitive name

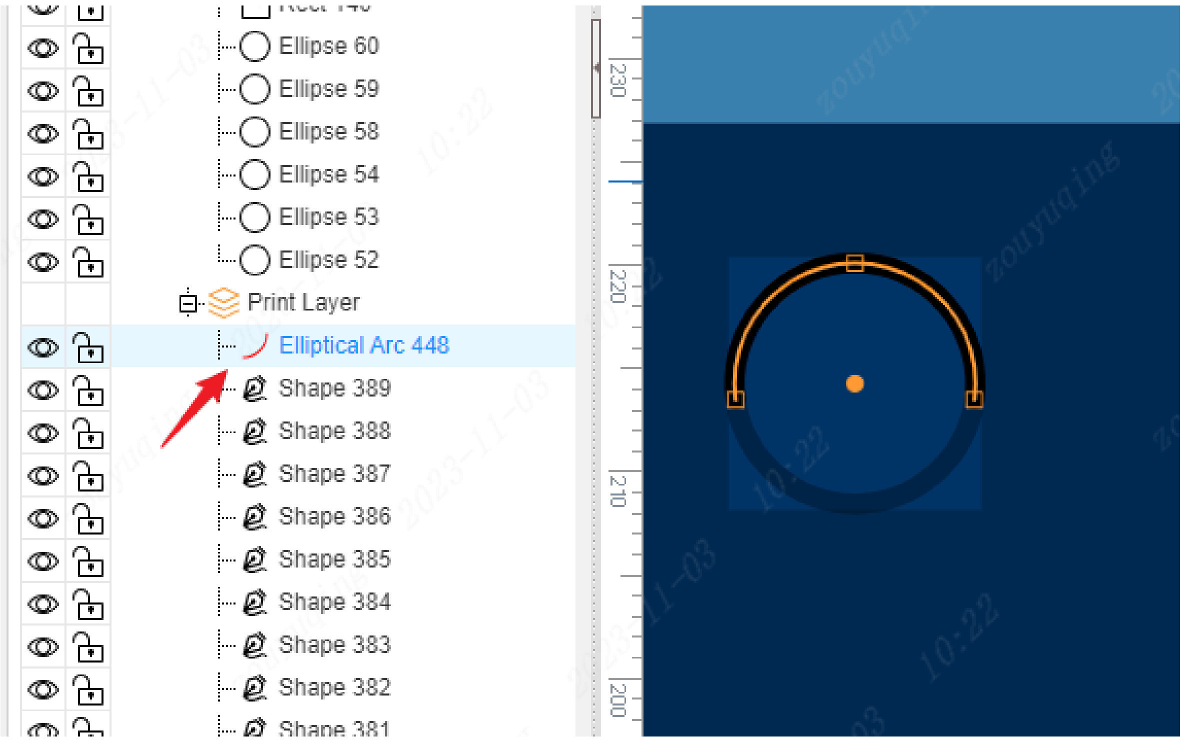


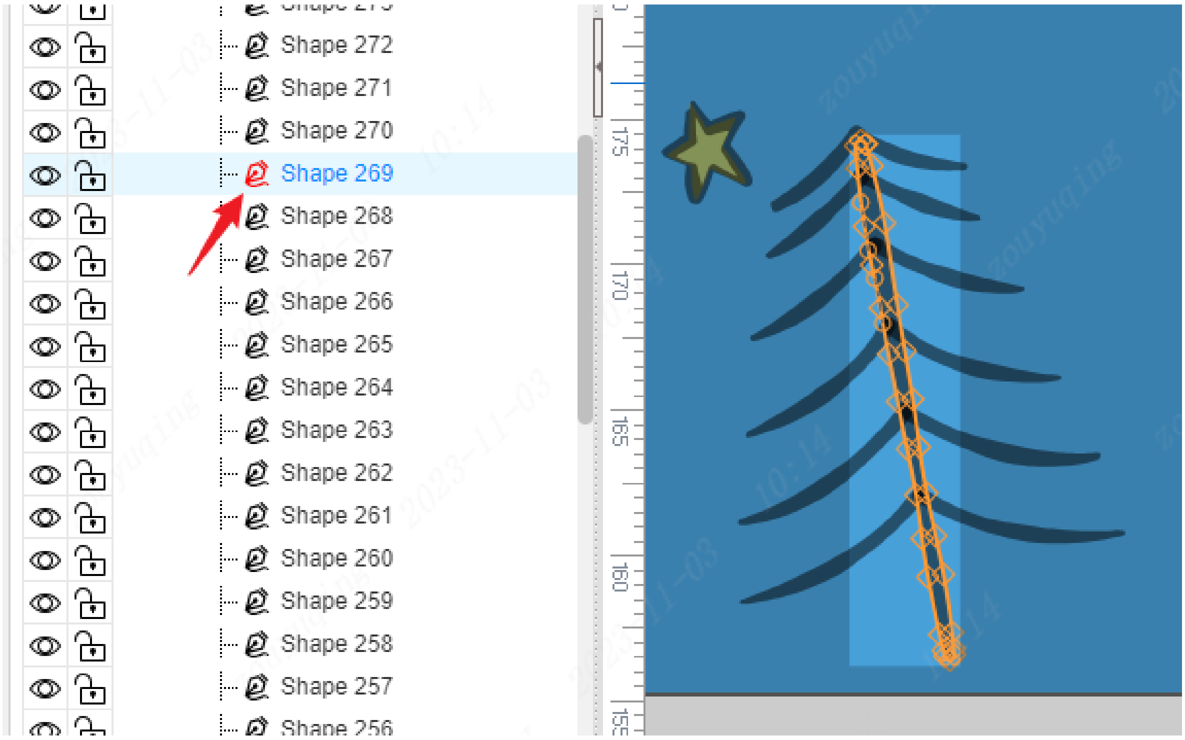
Type Icon

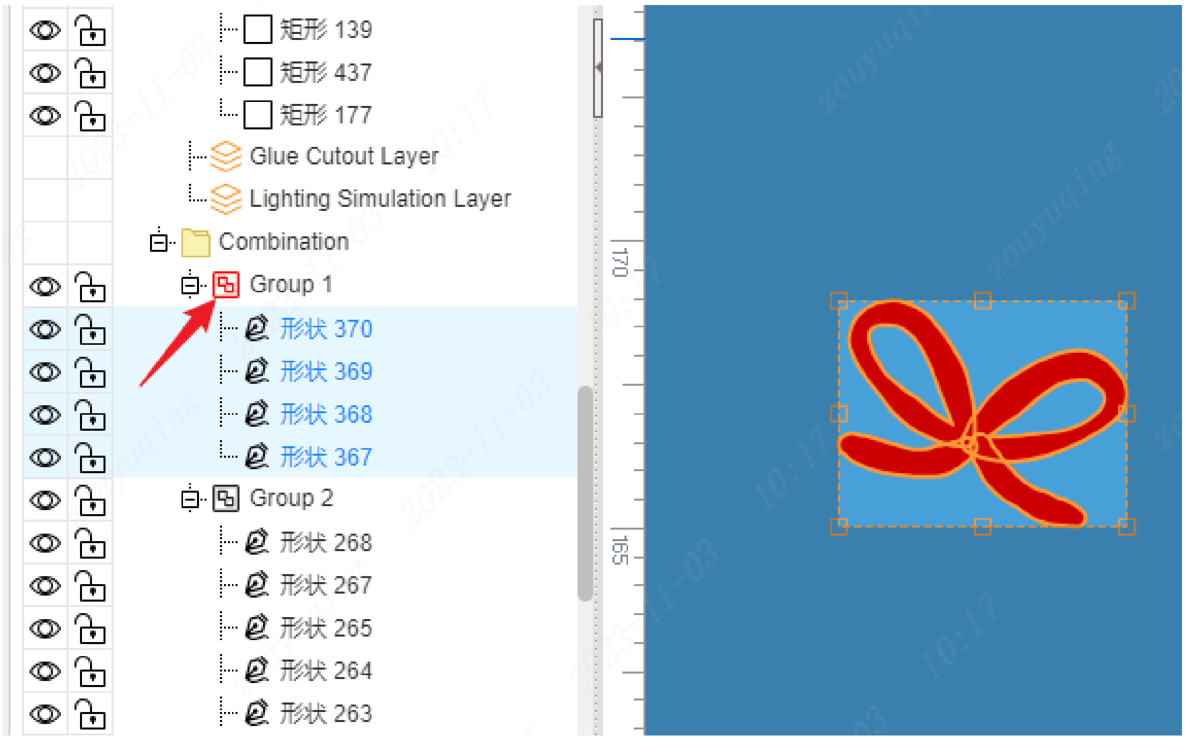
- The element ID name is usually black before displaying the corresponding type of icon



- When the arc/shape/combination type icon is displayed in red, it means that the current canvas is under the corresponding arc/shape/combination mask.

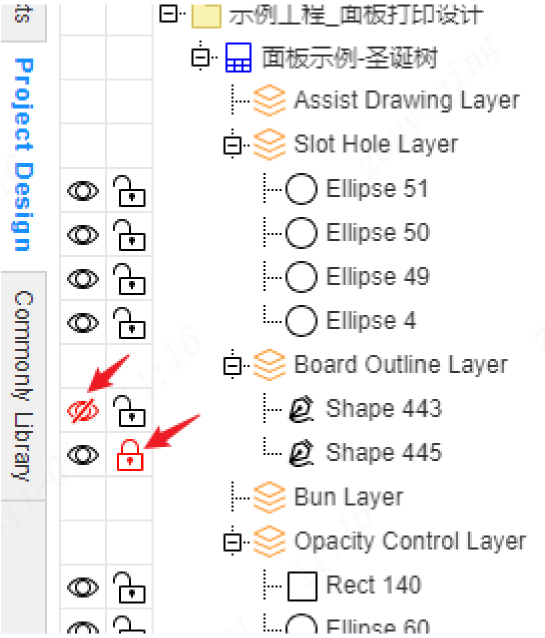






Hide and lock

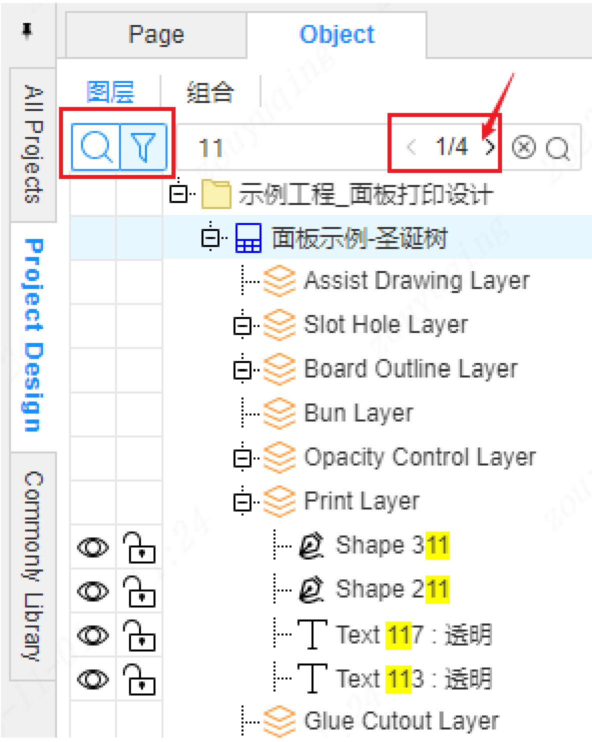
- Click the eye icon to set the hidden properties of the primitive, and click the lock icon to set the lock properties of the primitive.



The red closed-eye icon represents that the corresponding element is currently hidden and invisible on the canvas, and the black eye icon represents the visible state displayed on the canvas.
The red lock icon represents the locked state of the corresponding element on the current canvas, and the black unlock icon represents the unlocked state.

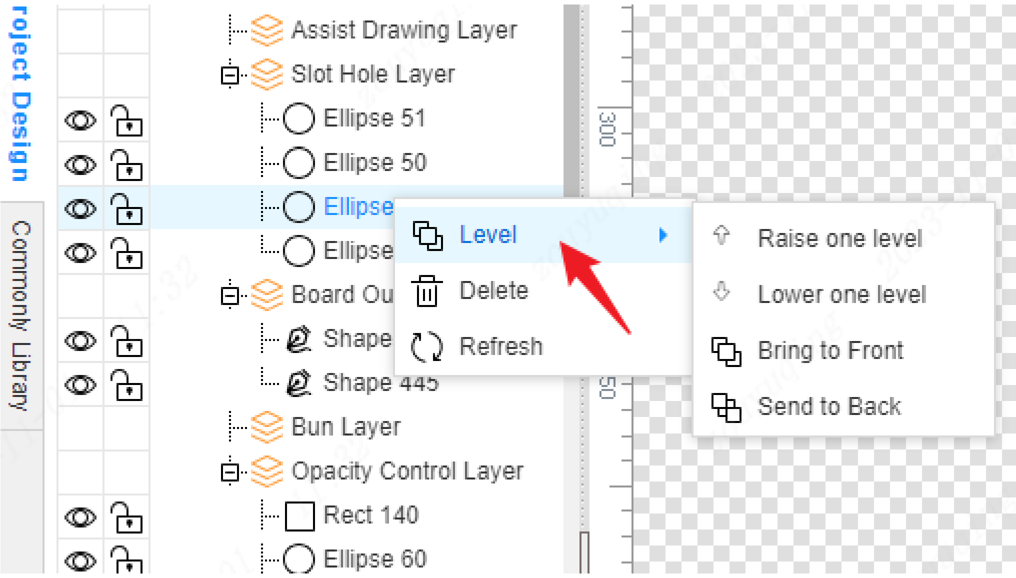
Search and filter

- Finding and filtering are supported. Switching between results is supported.



Right-click menu

The right-click menu in the object list can facilitate level switching operations.
Note: It is to switch the 4-level order of objects within the same layer.

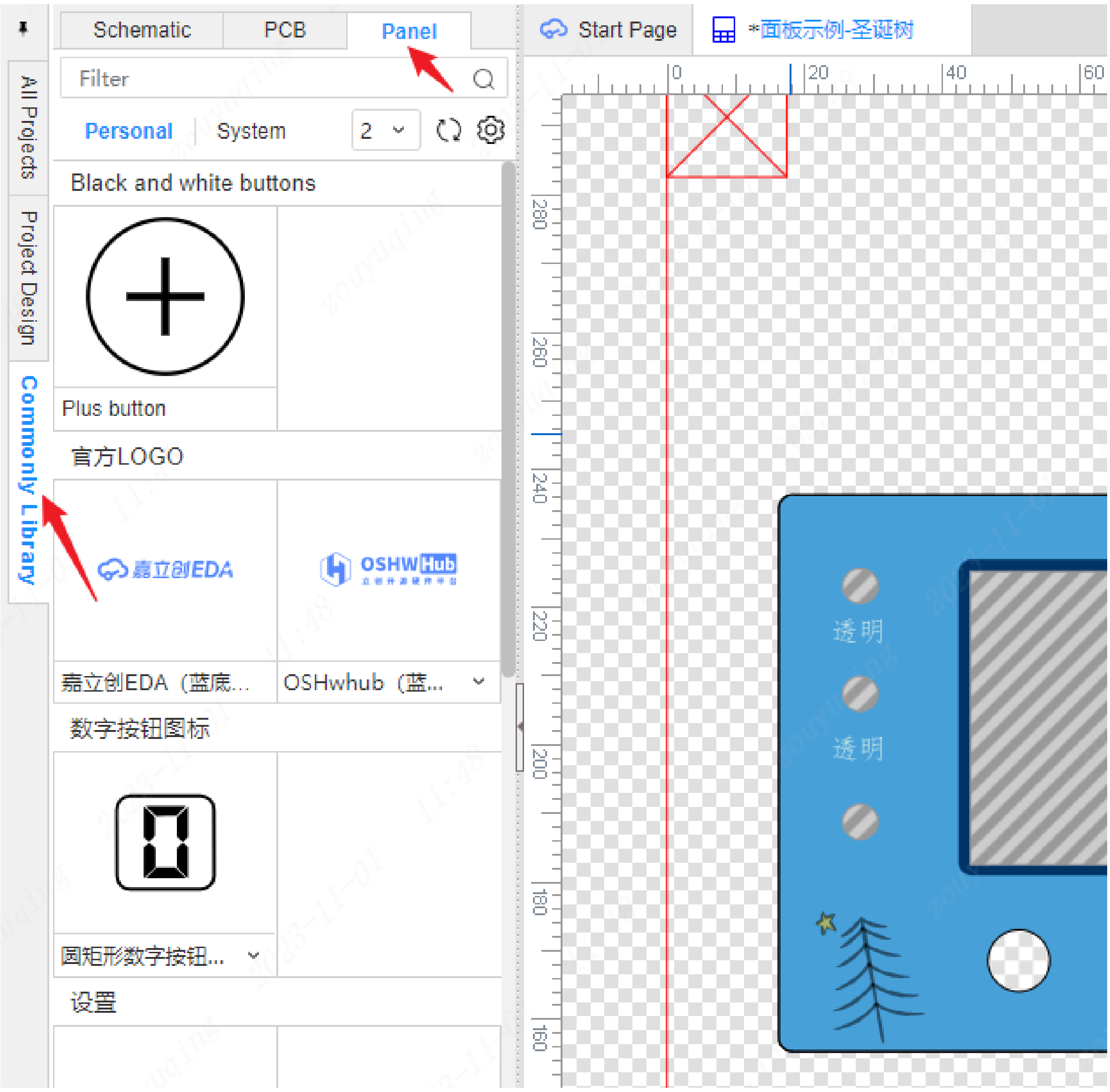


You can also right-click to delete elements and right-click to refresh the left object tree list.

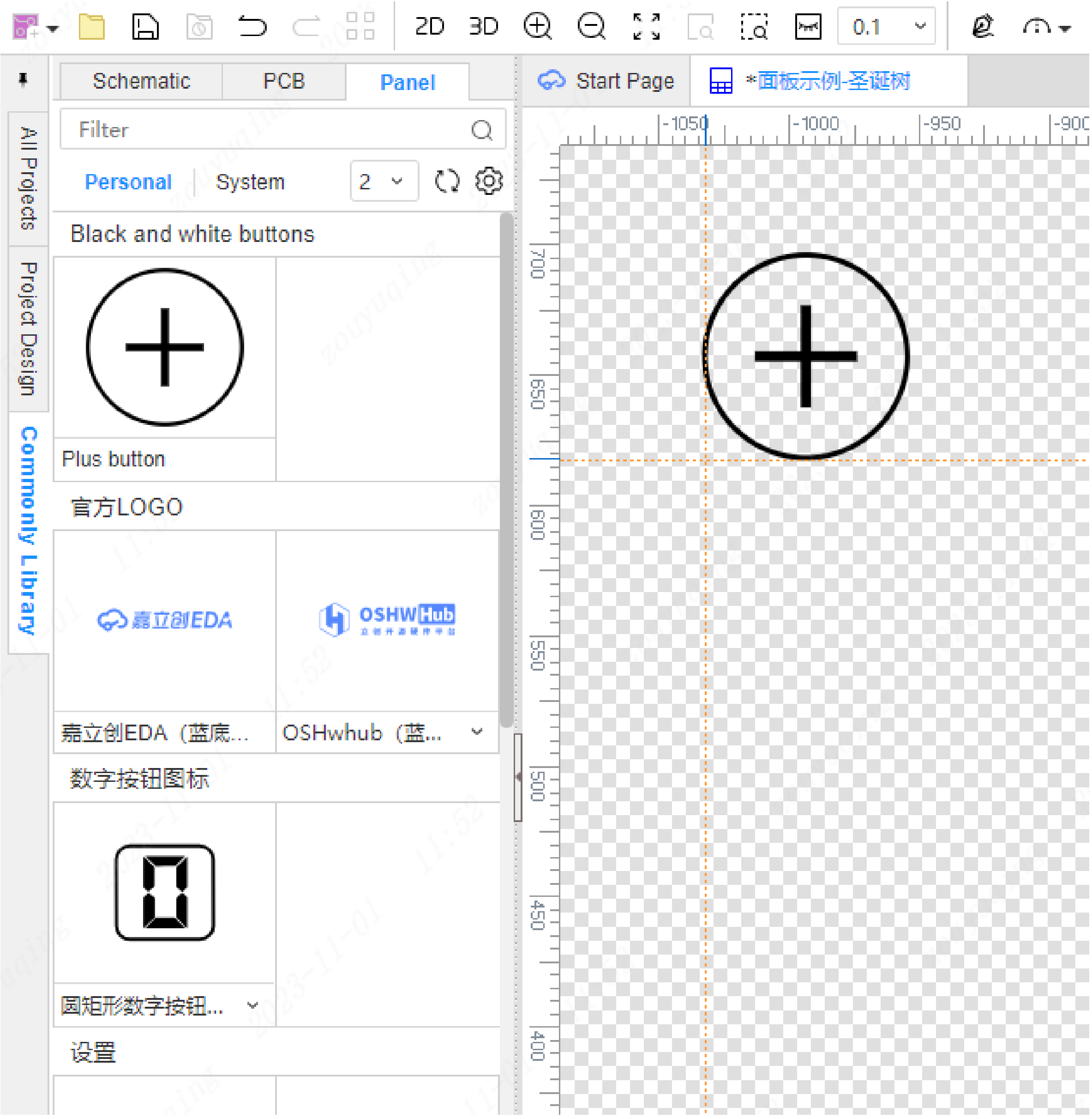
Left Panel - Commonly Library

Left Panel - Commonly Library

- Left Panel - Commonly Library - Panel



When the current edit page is a panel, you can select a panel library through the Common Libraries panel on the left to quickly place the panel library.



Filter lookup

Support input filter for specific search.

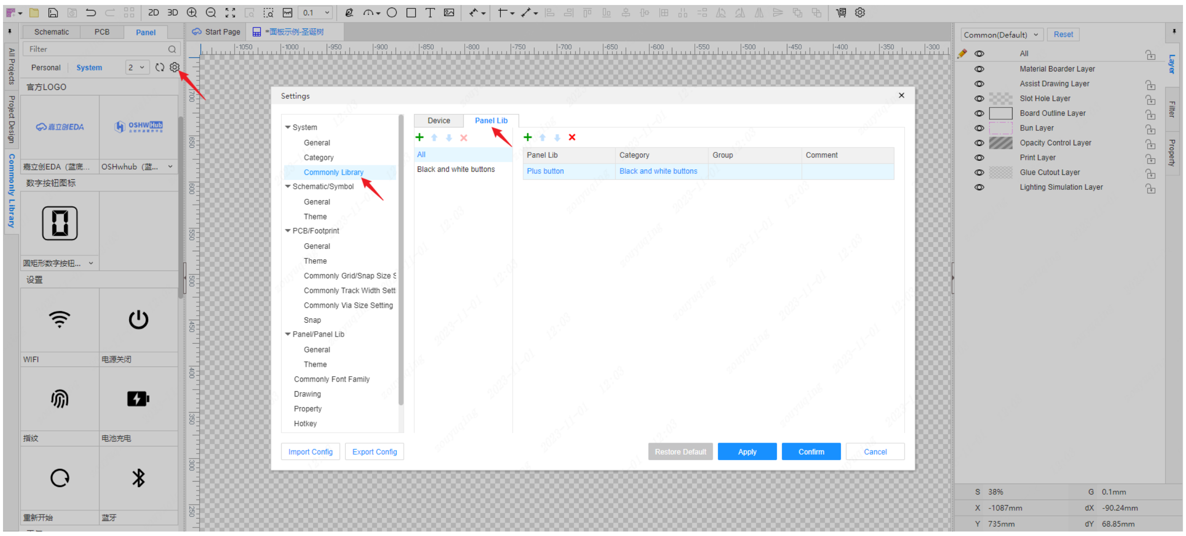
Personal Library & System Library

Personal libraries need to choose panel libraries to add to commonly used libraries.

The system library is a commonly used example panel library provided by the official.

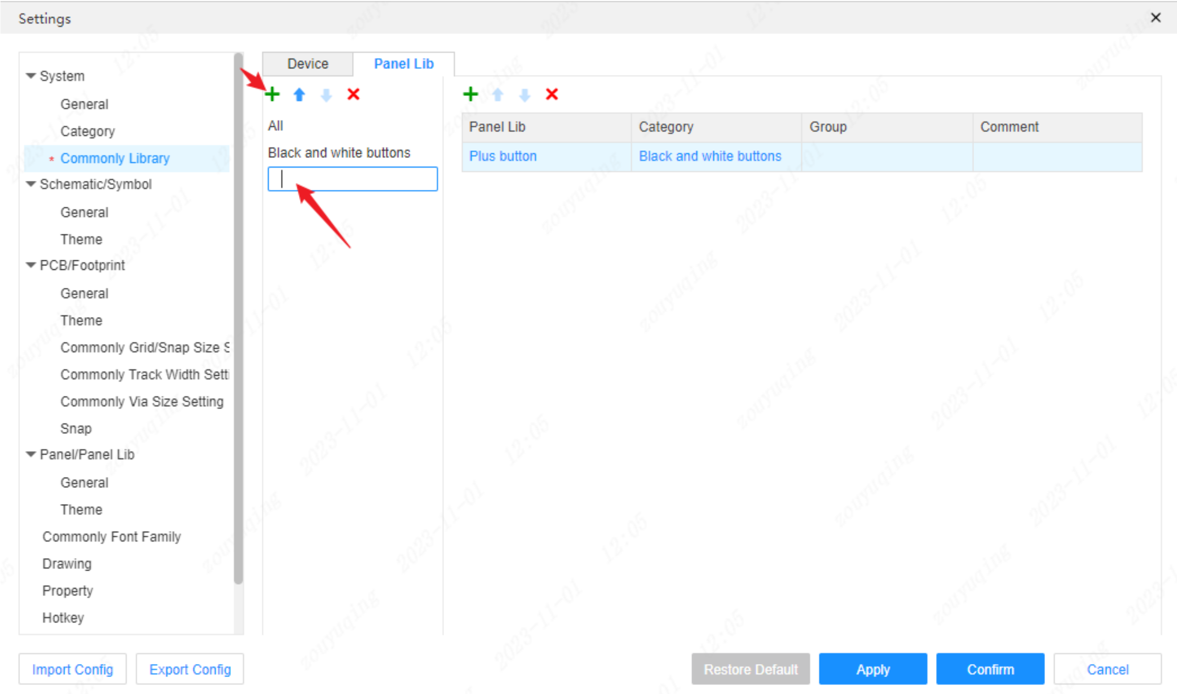
Add commonly library

Click the Settings icon in the upper left corner of the Common Libraries panel to pop up the Settings pop-up window to customize common libraries.



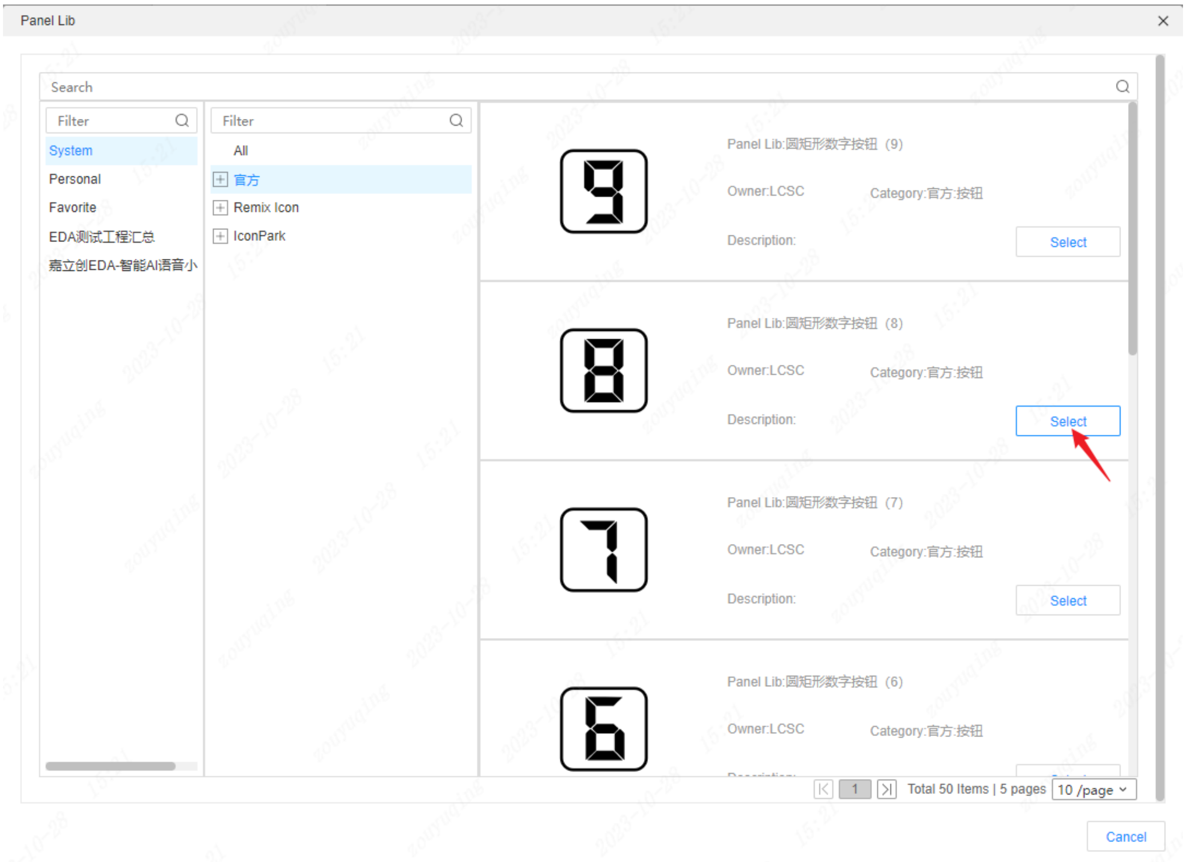
Commonly library classification

Click the green plus icon in the category list to add common library categories, and custom category names are supported.

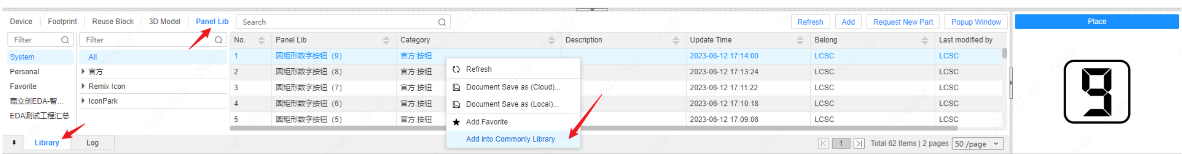


Commonly library added

Click the green plus icon above the library information list on the right panel, and a pop-up window to select panel libraries will pop up, which can be customized to add commonly used libraries.



You can also select the panel library in the bottom panel library, and right-click the menu operation to add to the frequently used library.



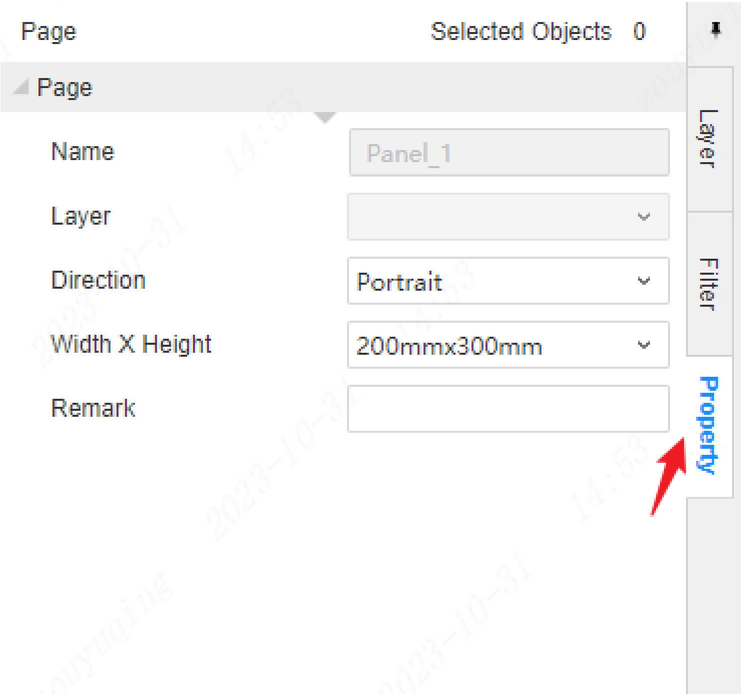
Right Panel - Properties

Right Panel - Properties

Image page properties

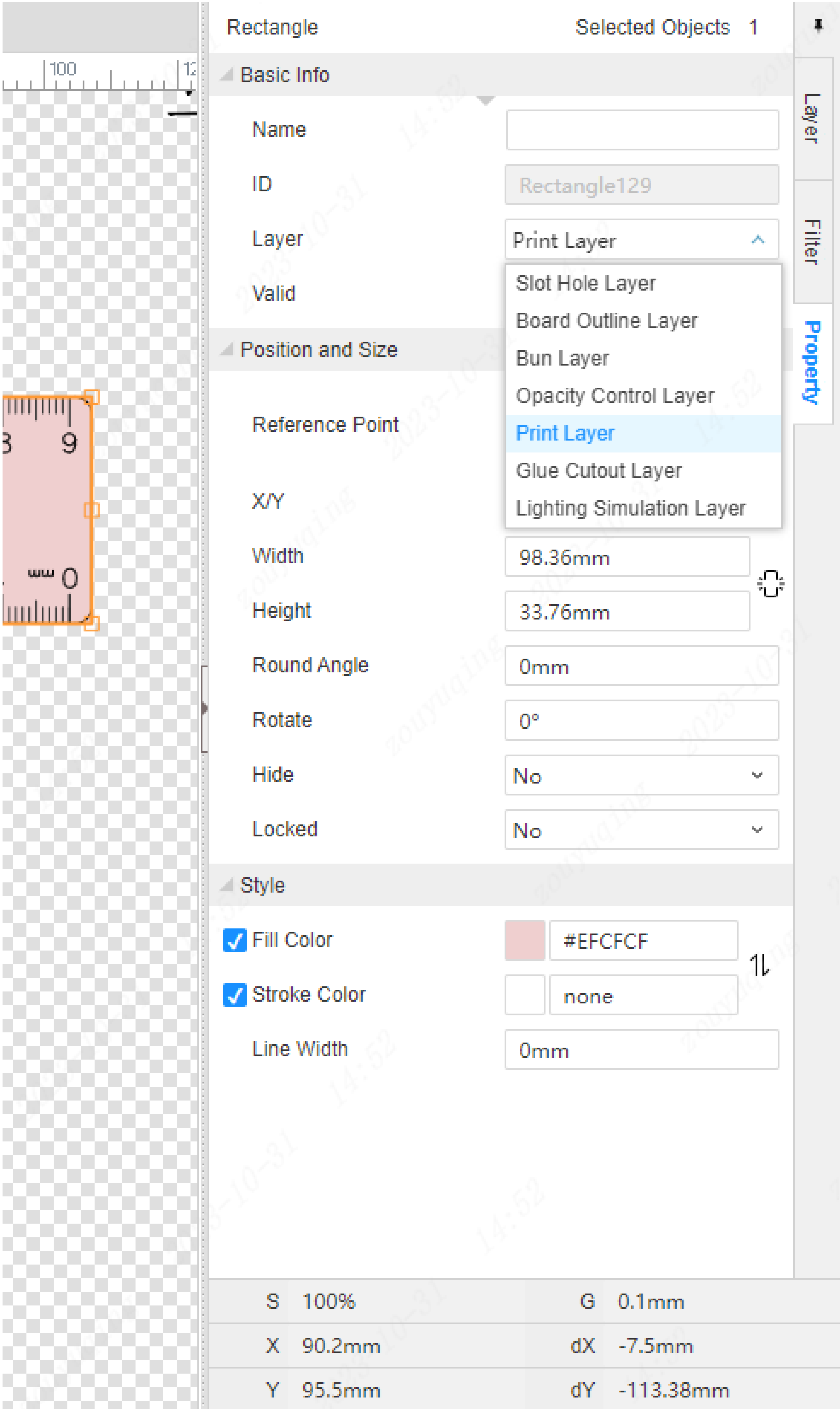
Name

The name of the panel design drawing, which is only displayed on the right and does not support modification. However, you can modify the panel name in the project on the left to sync to the right panel.



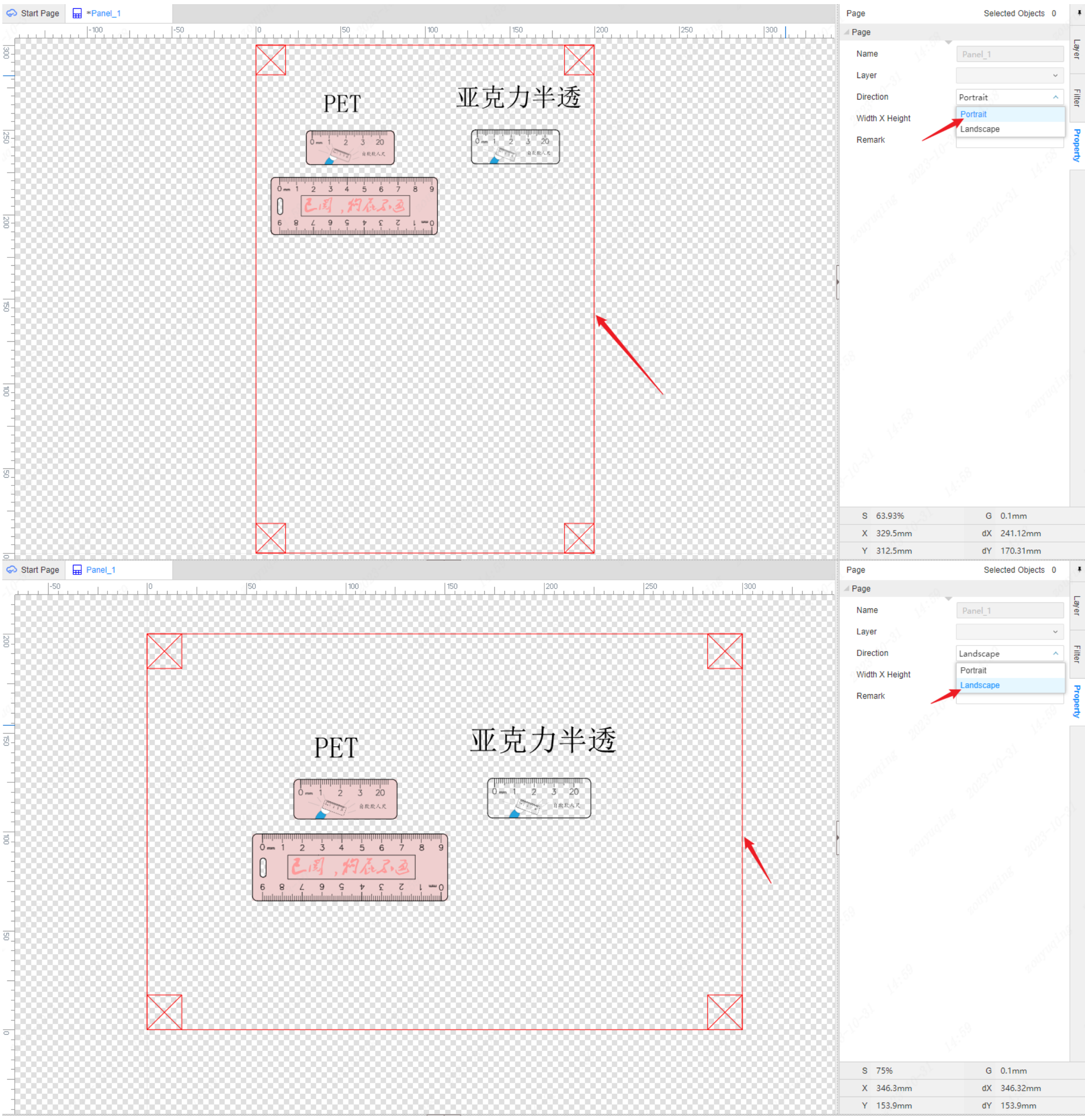
Layer

The layer display of the design drawing is not modifiable by default and can only be modified if the drawing is selected.



Direction

The direction of the design frame can be set vertically or horizontally on the design canvas.

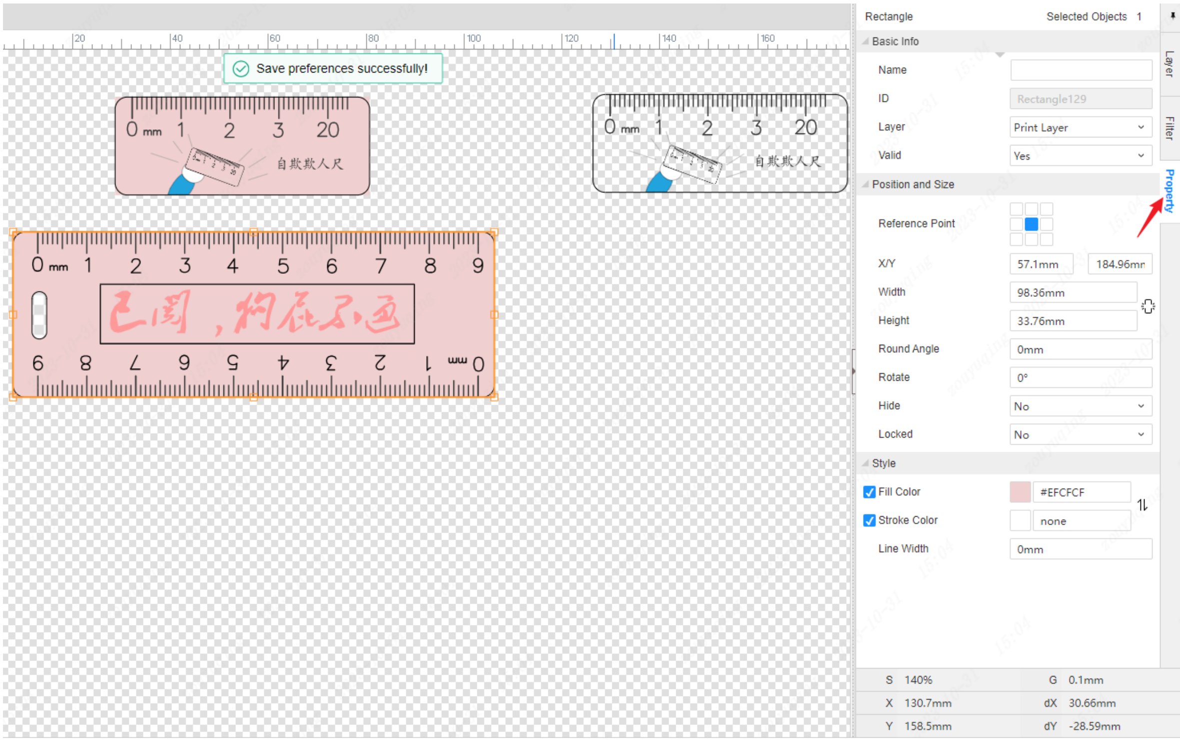


Note

The explanatory information used to annotate design drawings is convenient for explaining to assistants or other engineers.
Note: It is recommended to fill in the remarks column related to production on the next single page.

Object properties

When a primitive is selected on the canvas, its properties can be set in the property panel on the right.



Name

The name of the object supports customization and is usually empty by default.

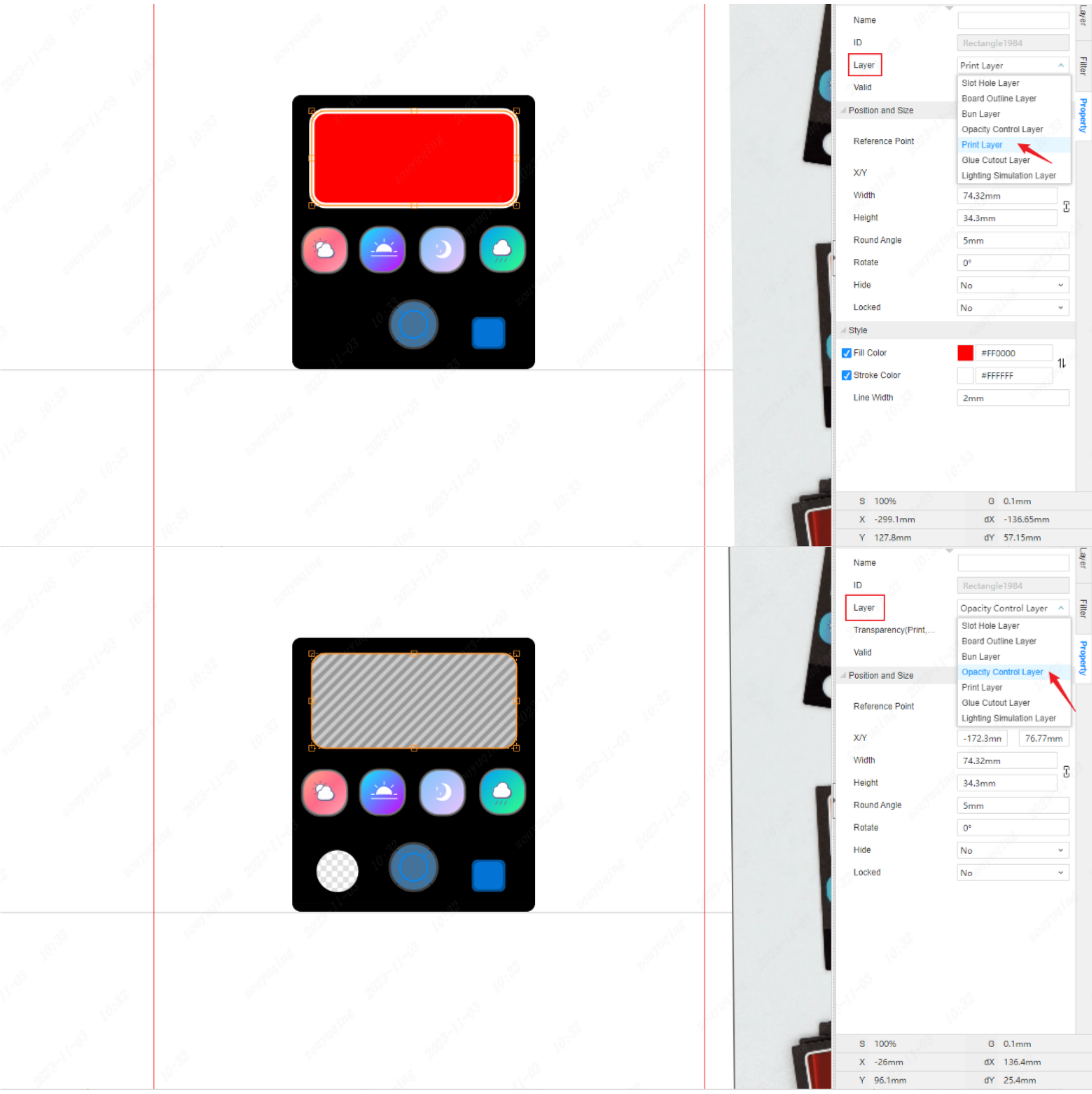
ID

The ID of the object is assigned by default, consisting of the primitive property type name + sequence number. It only displays and does not support customization.

Layer

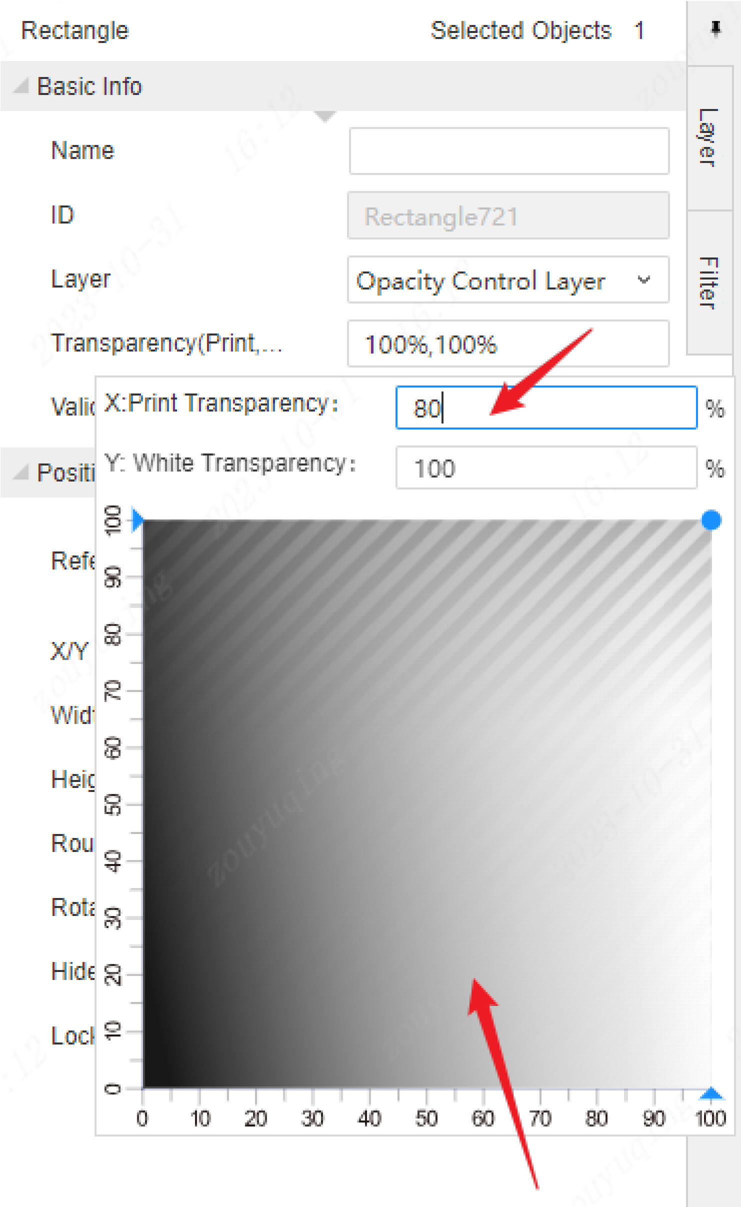
The layers that support switching selection in the layer drop-down box are all the layers that the object allows to draw. Text only supports transparent control layers and print layers, images only support print layers, and auxiliary elements only support auxiliary drawing layers.

After editing the layer properties on the right side of the property, the current editing layer of the canvas will automatically switch to the edited layer. (Consistent with the effect of switching the current editing layer in the layer panel on the right)



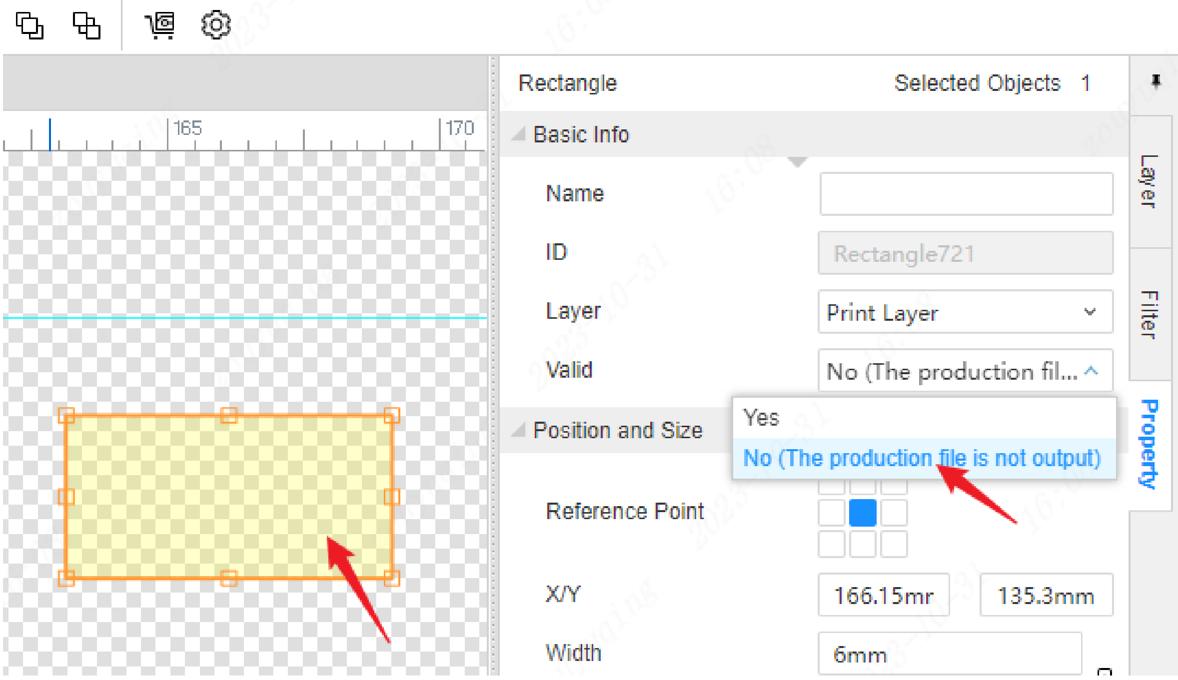
Transparency(Print，White)

To set the transparency of the printing layer and white layer, you can directly enter specific values or quickly set relevant transparency parameters through the coordinate selector below.



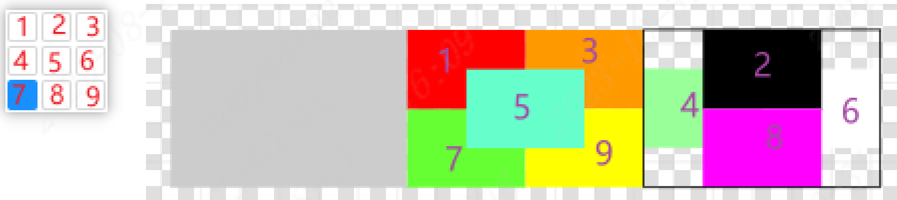
Valid

The panel elements drawn by default are all valid and can be set to not be valid in the properties on the right. The information of the object will be kicked out of the actual production file, and a special style highlighted in yellow will be visible on the canvas.

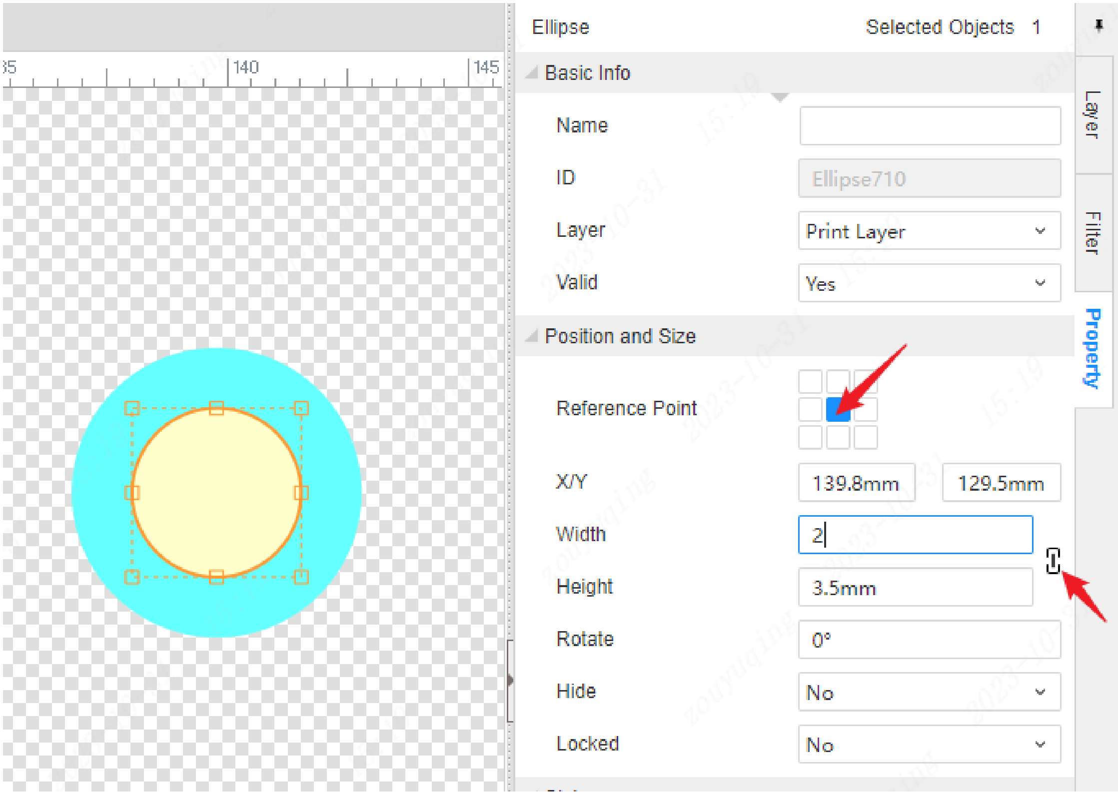


Reference point

- Modify the reference point of any object, and the reference points of all objects on the canvas change synchronously, that is, the global object reference point remains consistent.
- The same primitive can be deformed differently on the canvas by setting different reference points.
  For example, when 9 different reference points are set separately, and the ratio of rectangle width and height is reduced to half of the original, the changes shown in the figure below will have different effects



- Set the middle point as the reference point, and modify the width and height of the perfect circle on the right side to keep the center of the circle fixed, which is convenient for modifying the size of concentric circles



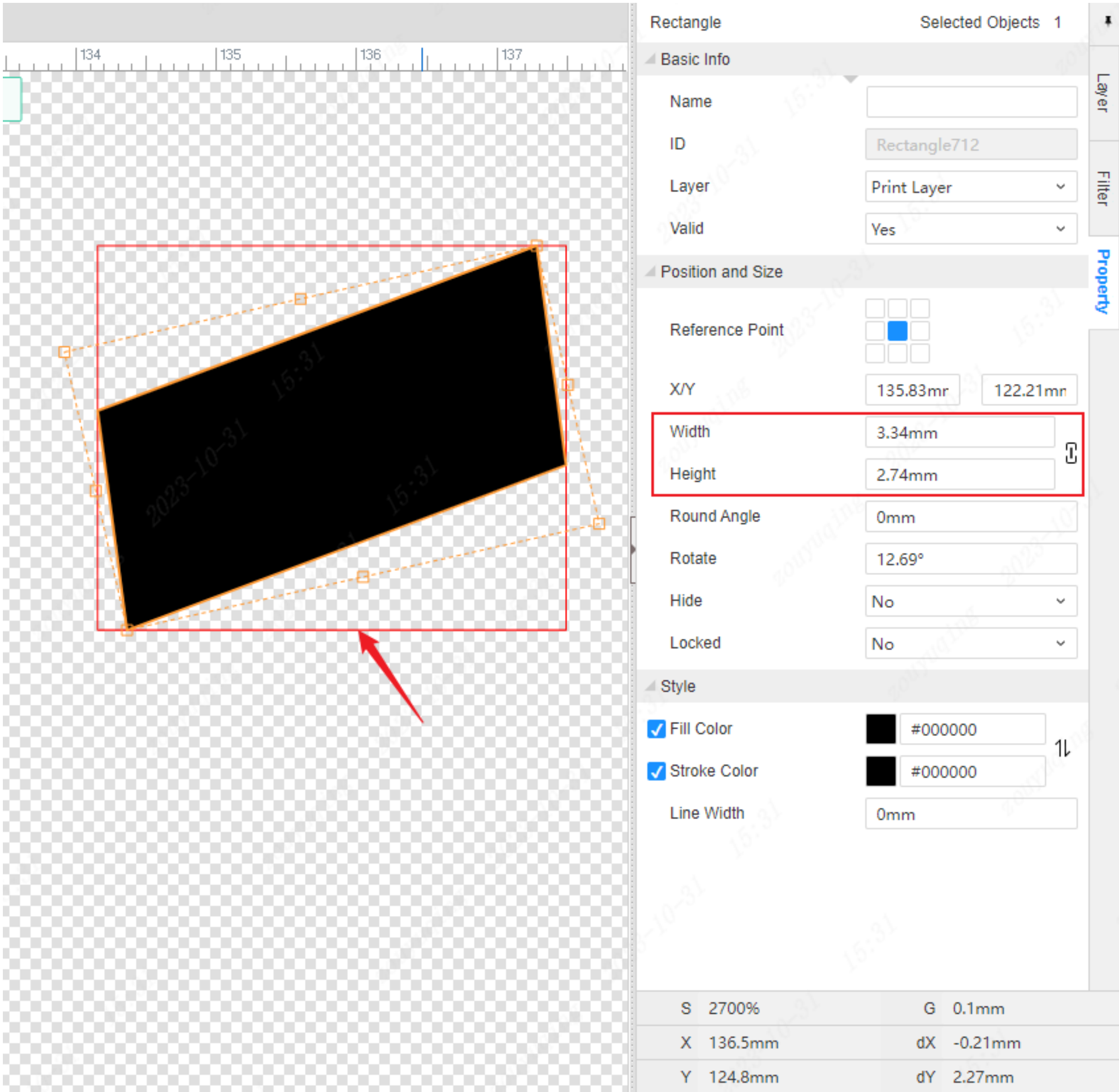
- The adjustment results of the rotation angle, width and height dimensions, position coordinates and other attributes of the object are all related to the setting of the reference point.

X/Y coordinates

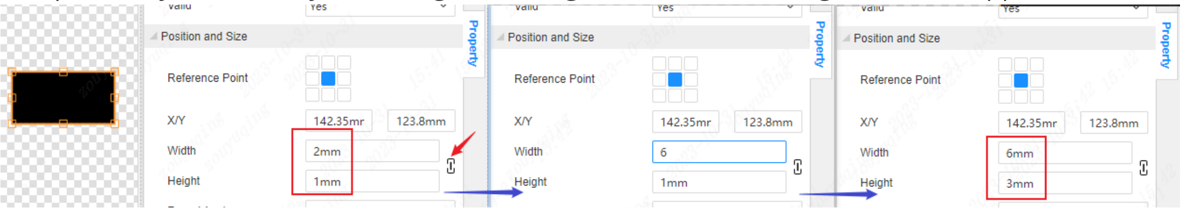
- The XY coordinate value at the reference point of the object, in mm, supports adjustment.

Width and height

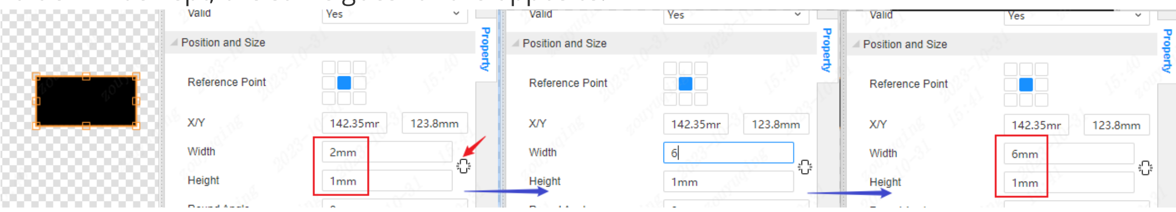
The provided here is the width and height value of the object's axis alignment surrounding the rectangle, that is, a rectangle parallel to the coordinate axis, which completely surrounds the primitive, and each edge is parallel to the coordinate axis.



When setting "Keep aspect ratio", only the width value is modified, and the height value will also be updated synchronously according to the original ratio; the same goes for the opposite.



After clicking to switch to not keeping the aspect ratio, modify the width value, and the height value will be kept; the same goes for the opposite.



Round Angle

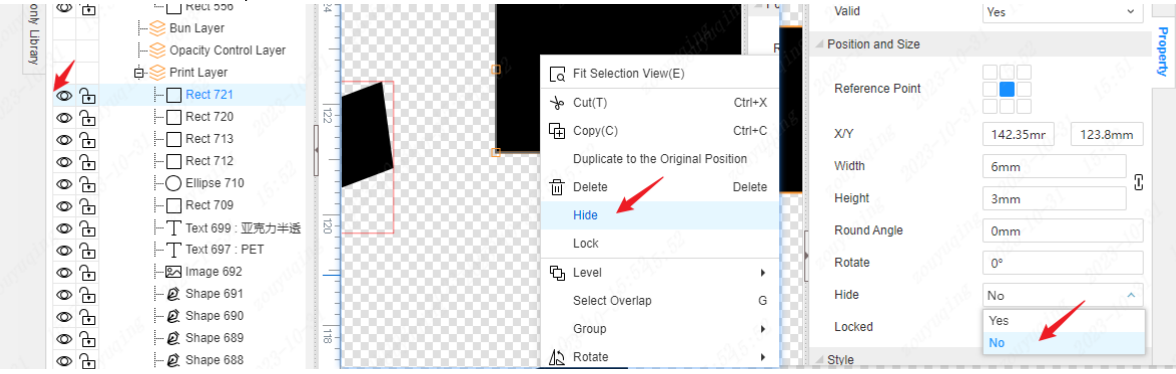
Currently, only rounded corners of rectangles are supported, and only 4 rounded corners can be set at the same time.

Rotate

The rotation angle value generated by directly manipulating the object on the canvas will be synchronized to the right property. Similarly, the rotation value on the right side is adjusted, and the rotation angle of the object on the canvas is updated in real time.

Hide

The drop-down box can be selected as "Yes" or "No", corresponding to the hidden display settings of the object tree on the left and the right-click menu of the selected object on the canvas. The three settings are linked in real time.
Note: Hidden is only temporarily invisible on the canvas and does not determine whether it will take effect in actual production.



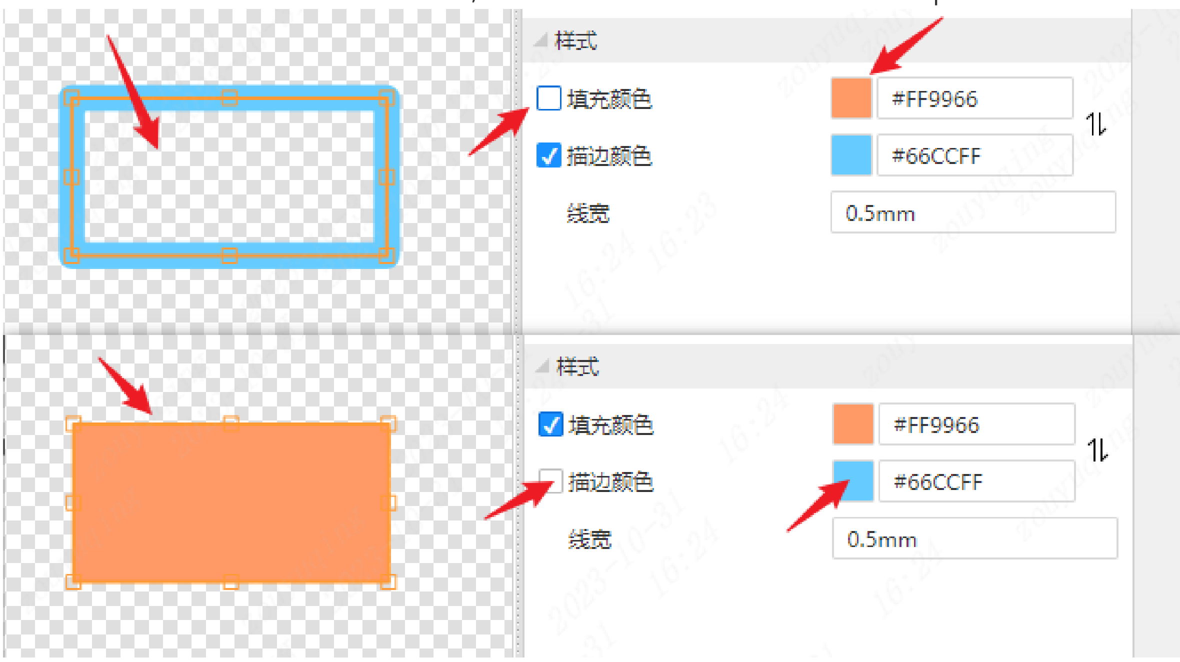
Locked

The drop-down box can be selected as "Yes" or "No", corresponding to the lock unlock setting of the object tree on the left and the right-click menu of the selected object on the canvas. The three settings are linked in real time.

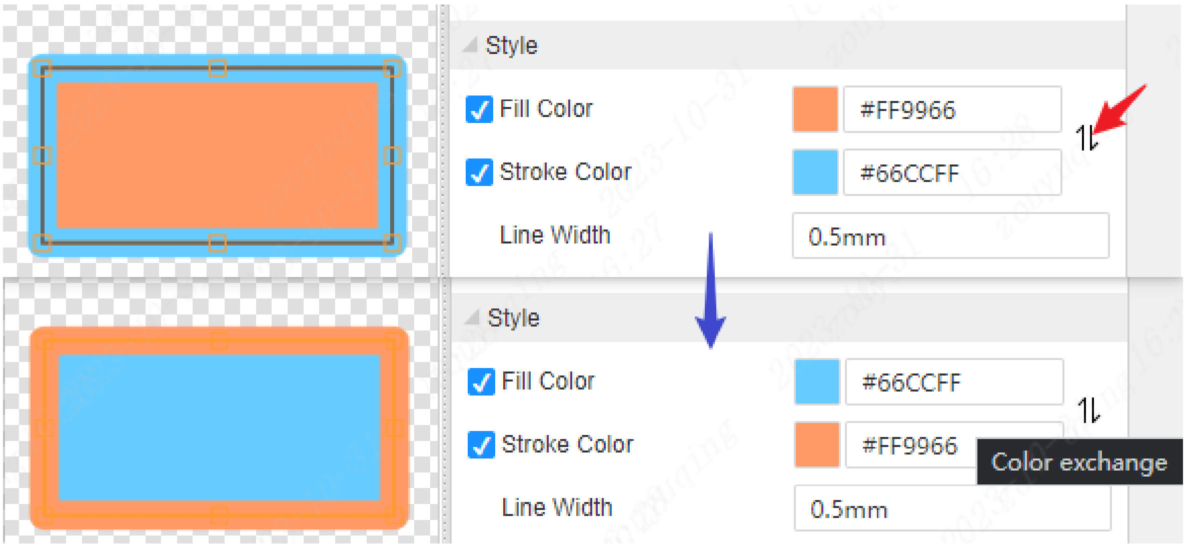
Fill Color, Stroke Color

Only objects in the print layer and light simulation layer can be set to fill and stroke styles.

The fill color and stroke color are selected by default, that is, the set color is applied to the current object.
Uncheck, the object's fill and stroke color settings remain, but do not apply to the current object. No fill or stroke is visible on the canvas, and there is no fill or stroke in actual production.



Support for quick swapping of fill colors and stroke colors. Click the "Color Swap" setting icon behind the color value box.



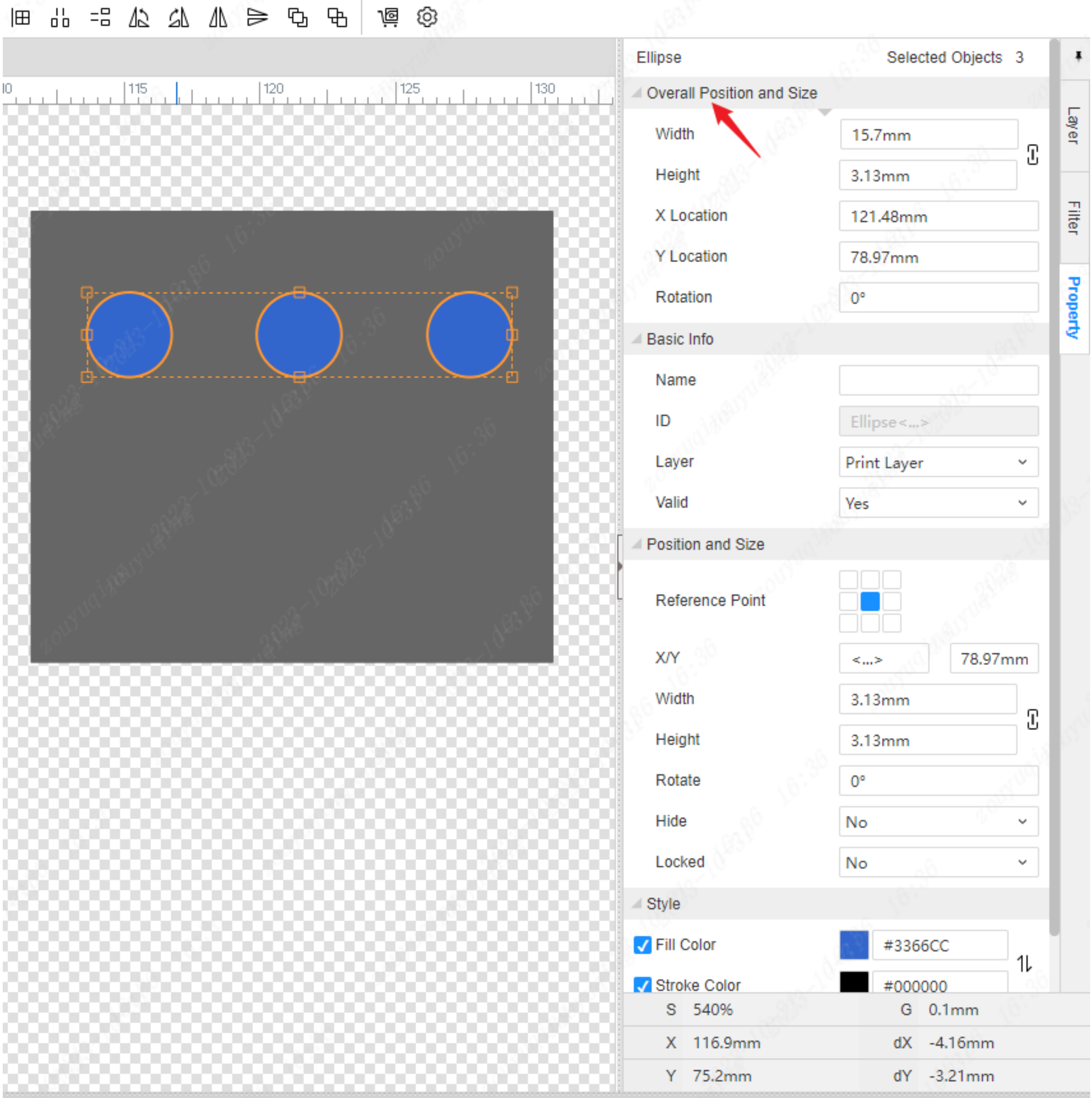
Line Width

The line width of the object drawn by default is 1mm, which supports customization.

For the numerical text box in the property panel, it supports simple four arithmetic input, which will automatically calculate the result. For example, if the width is input: 5 + 3, press enter will automatically calculate 8 and apply it. If the input does not meet the rules, it will be reset to the old value.

Multi-object properties

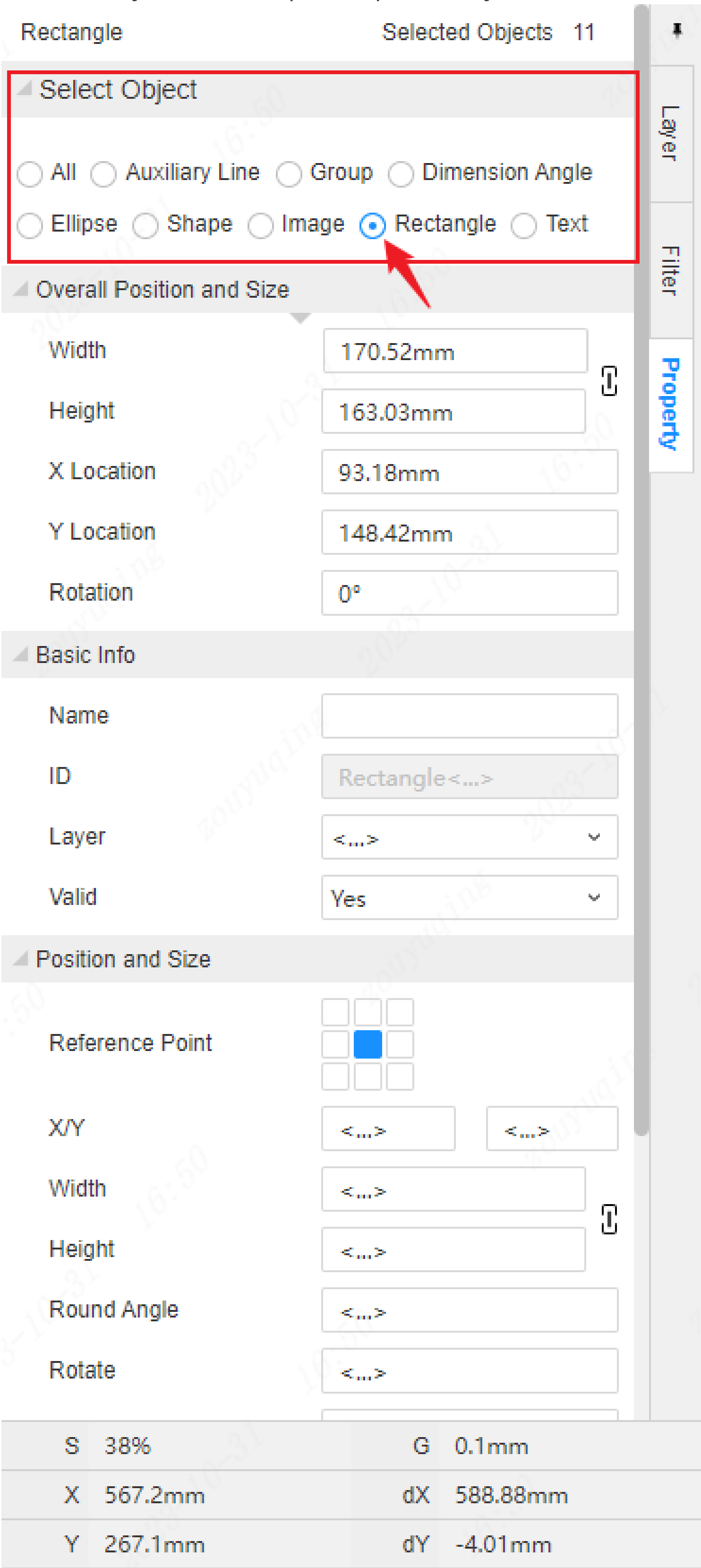
When multiple objects are selected, the overall position and size information and other properties can be modified in the property panel on the right.



Angle of rotation

The **rotation angle** here is the angle at which all selected objects are rotated as a whole, which is different from the **rotation** angle of a single object.
Setting the overall rotation angle will increase the rotation angle of each individual object by the same degree of rotation

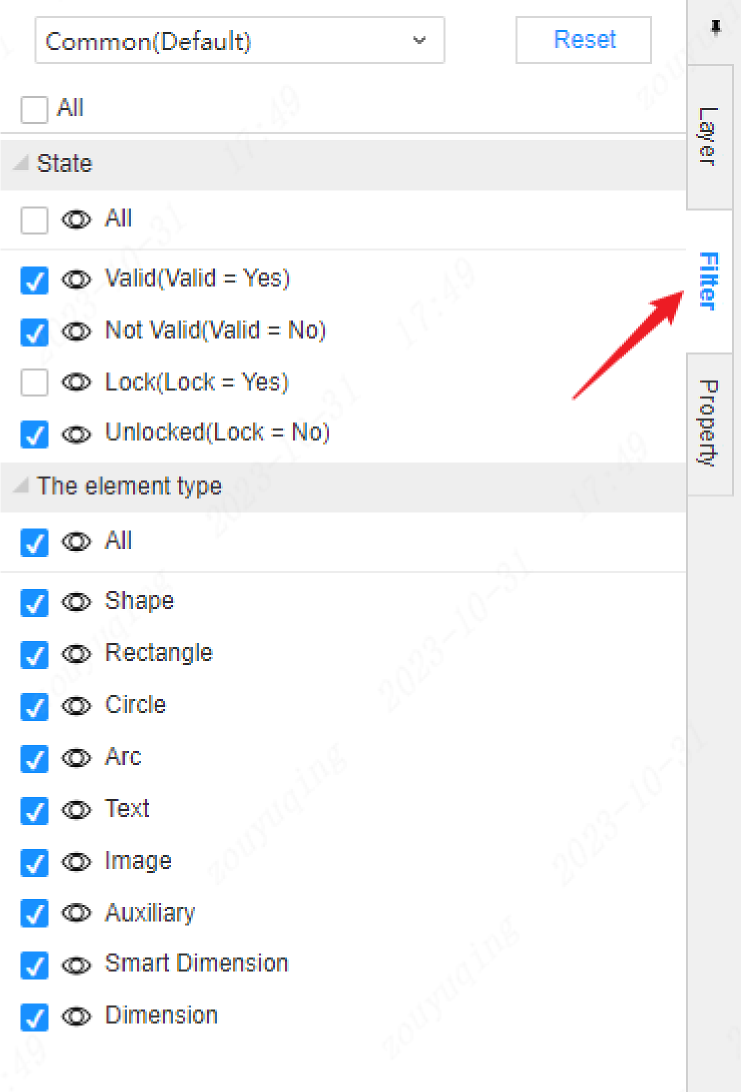
When selecting multi-object operation properties, be cautious and make batch changes. It is best to select objects at the top and operate objects of the same class in batches each time.



Right Panel - Filter

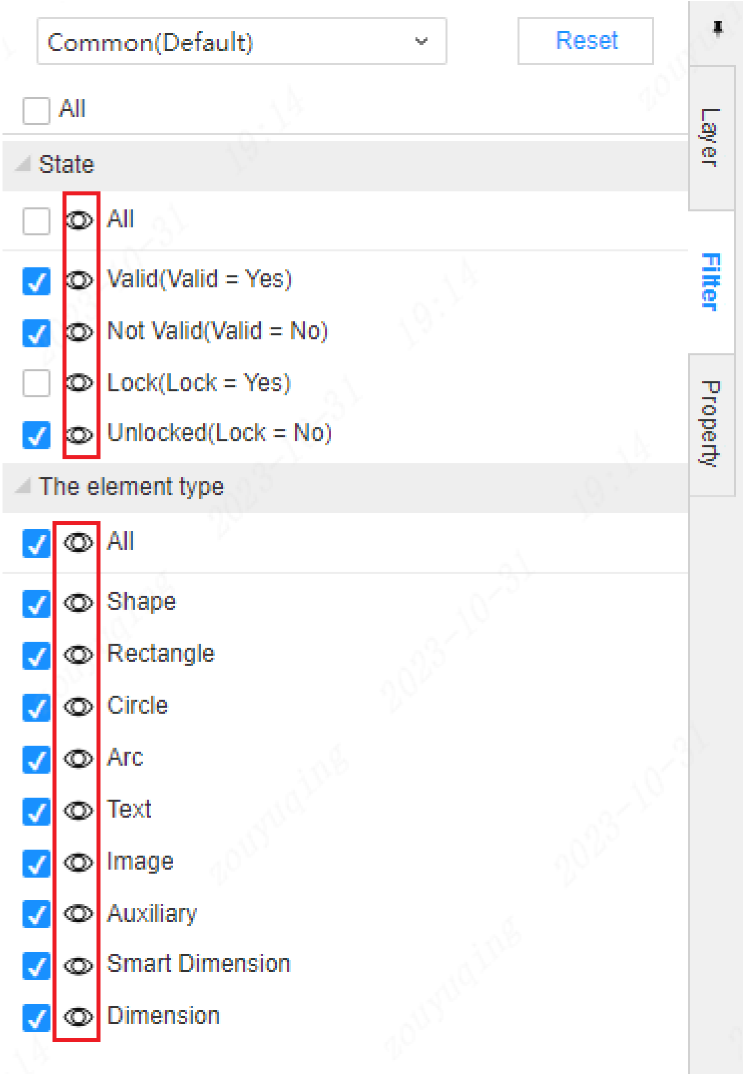
Right Panel - Filter

The required objects can be easily filtered in the right filter panel.



Show/hide

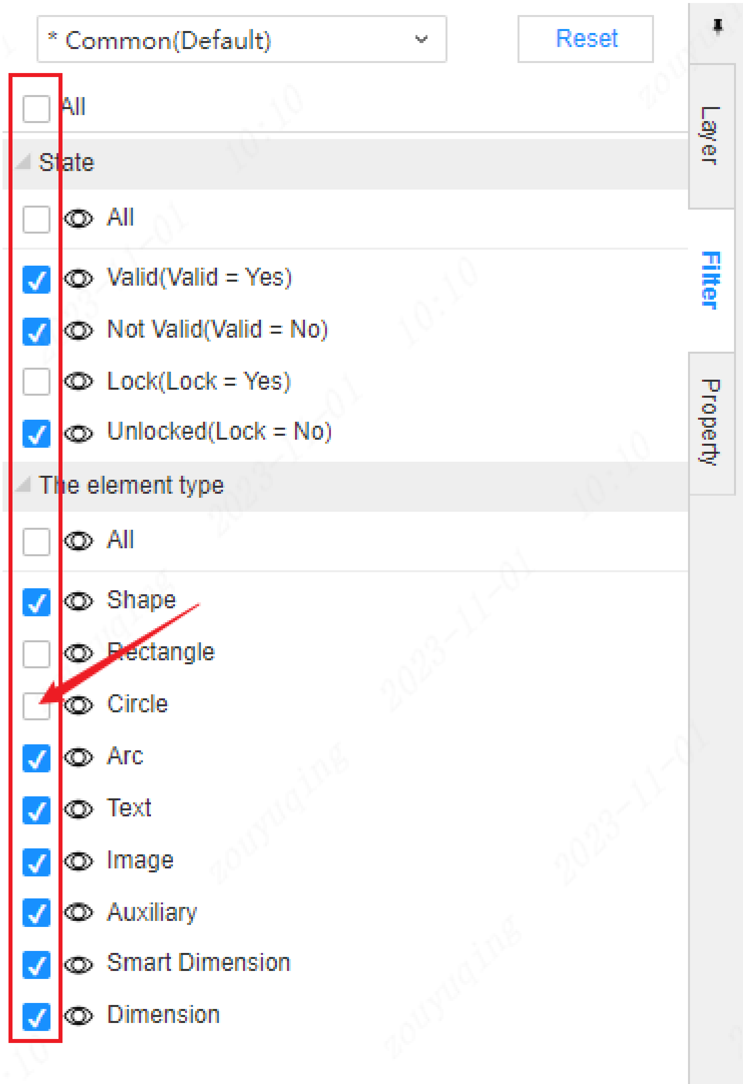
You can set to show or hide certain types of elements in the panel canvas.



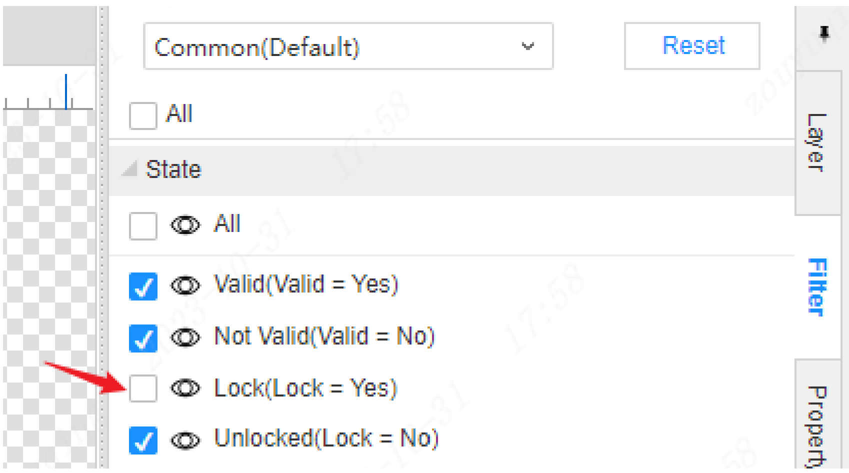
Click the small eye next to it to batch hide elements of the corresponding type on the canvas.

Check option

You can set filter conditions by clicking the option tab to select or cancel the corresponding option. If the filter type name is unchecked, it cannot be clicked or selected on the canvas, but can be selected on the canvas when checked.



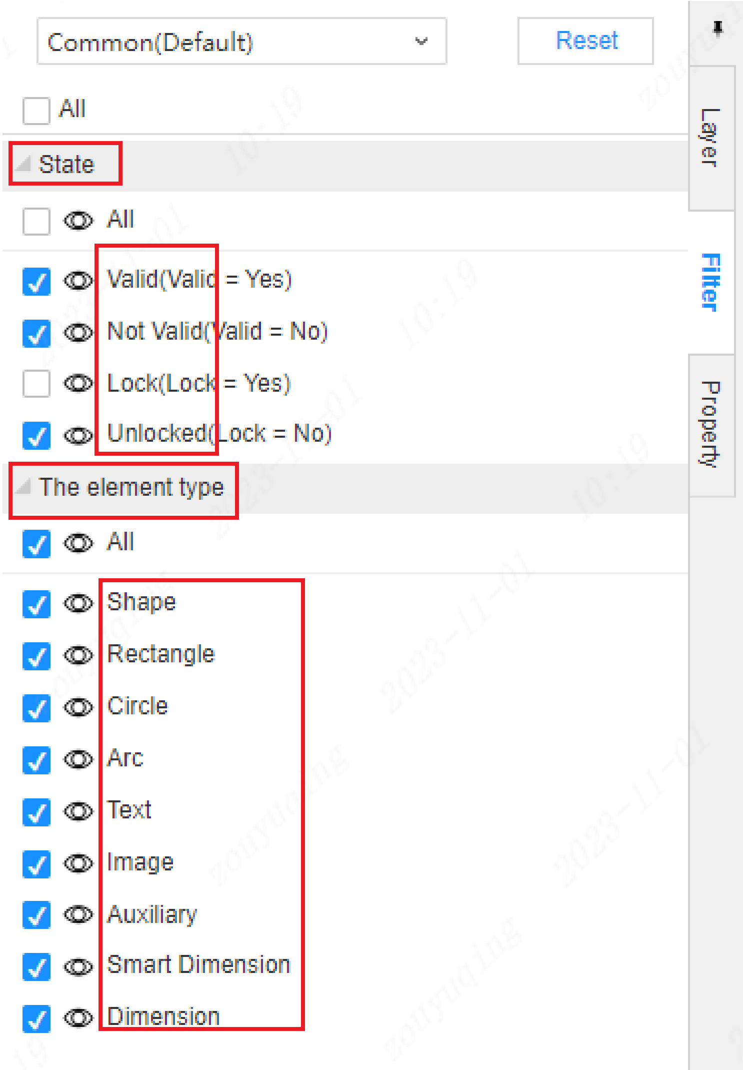
Objects of the default lock type are filtered.



Filter type

The relevant properties of whether the primitive is effective or locked are set in the right property panel of each primitive

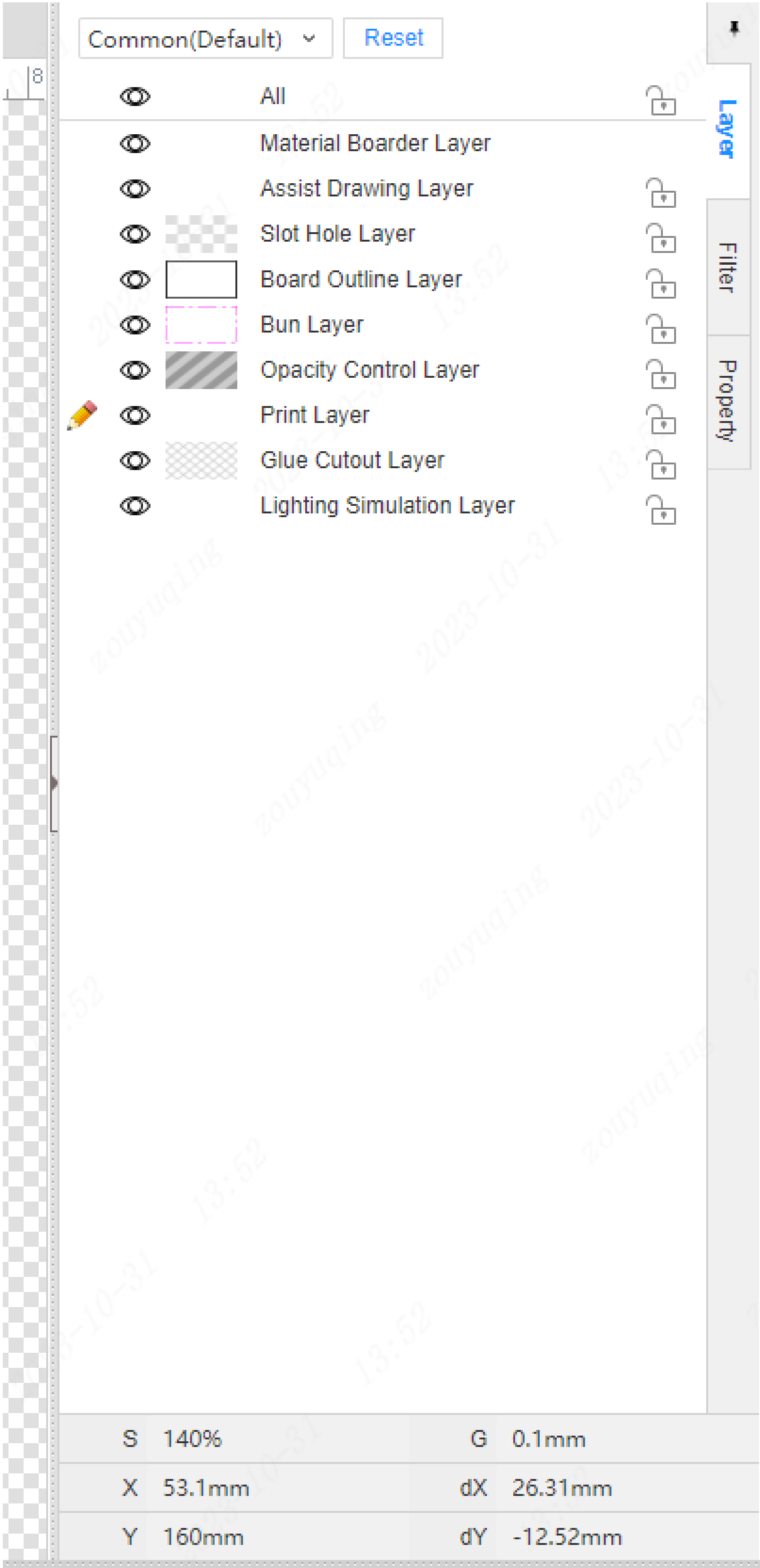
- **Status**: Filter according to the state attribute type of the element.
- **Element type**: Filter by panel element type



Right Panel - Layer

Right Panel - Layer

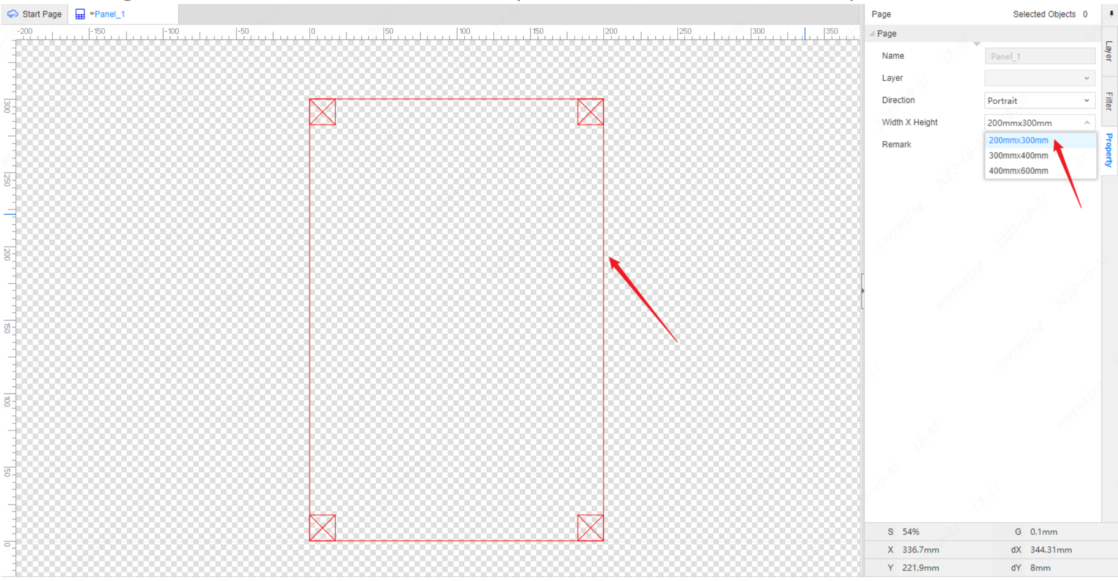
The panel design needs to know the concept of several layers. The settings of these layers are in the properties of the primitives. Currently, there is no separate layer display interface.



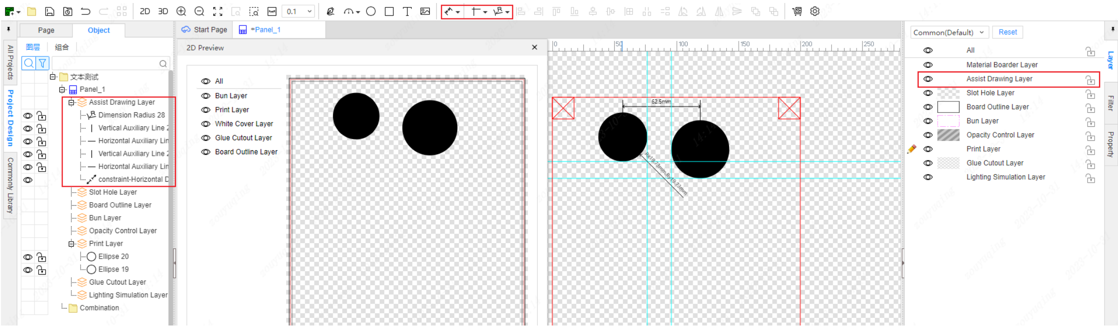
Layer introduction:

- **Material Boarder layer**: Represents the selected material area, which cannot exceed this area, otherwise it will not be possible to print and cut.
  This layer is only used to display the red material bounding box, and the orientation and width dimensions can be customized in the property panel on the right side of the canvas.

The settings here determine the material specifications used in actual production.

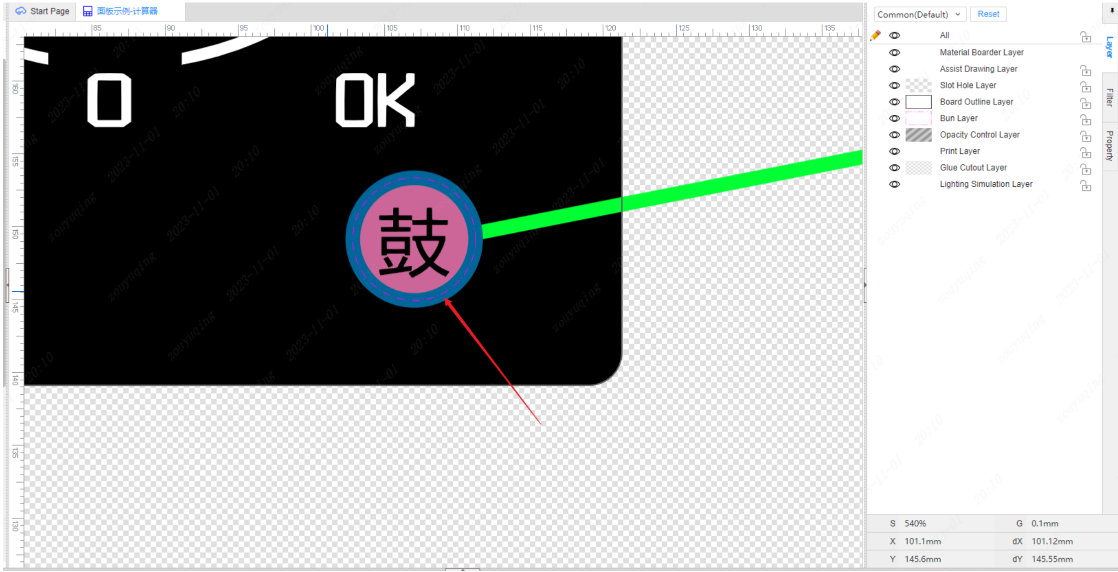


- **Assist Drawing Layer**: intelligent size, auxiliary line, auxiliary point, size are auxiliary elements that can only be located in the auxiliary drawing layer, and the layer elements are not effective in actual production.

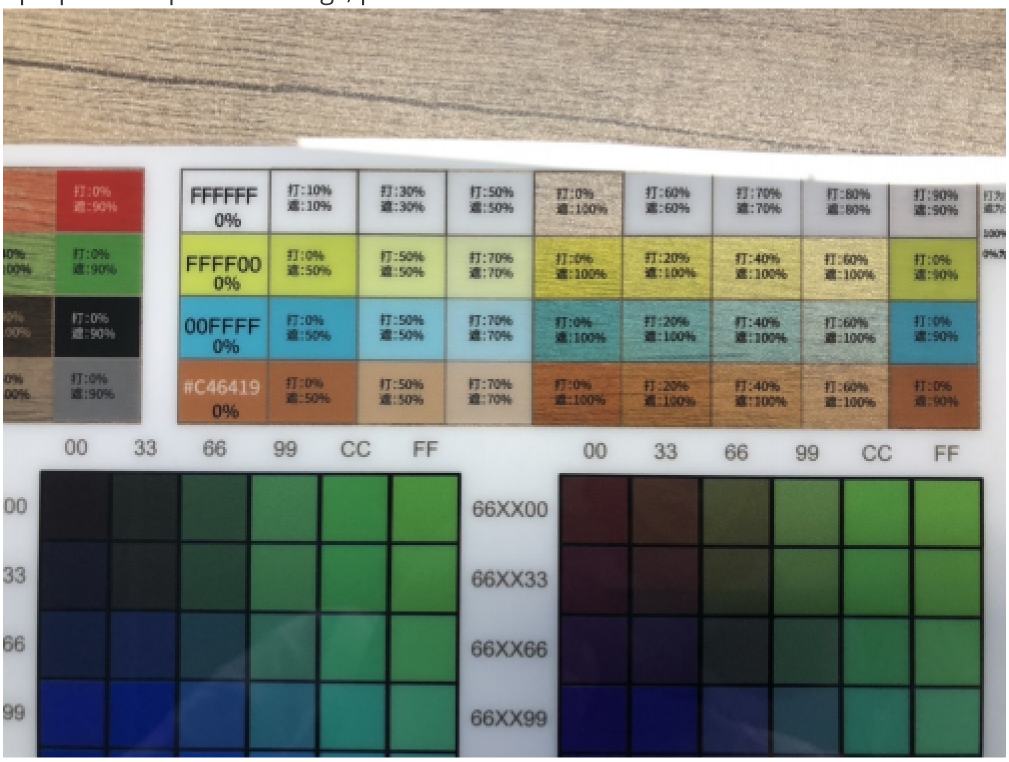


- **Slot Hole Layer**: The graphic placed on the digging layer means that the hole is opened here, and the entire panel is opened.
- **Board Outline layer**: It is used to draw the outline drawing that needs to be cut. It can cut special shapes, all wave shapes are supported, and the path needs to be closed. Content located outside the board frame layer cannot be exported.
- **Bun Layer**: used to draw the raised drum package graphics, this layer does not support the placement of pictures, text, and the raised height and range of the drum package are also default, it is recommended that the drum package size diameter is greater than 6mm, and the drum package diagram element spacing is greater than 2mm.
The drum package diagram on the canvas has a fixed dotted line stroke style by default, the

default color is #FF00FF, and the line width is 1.

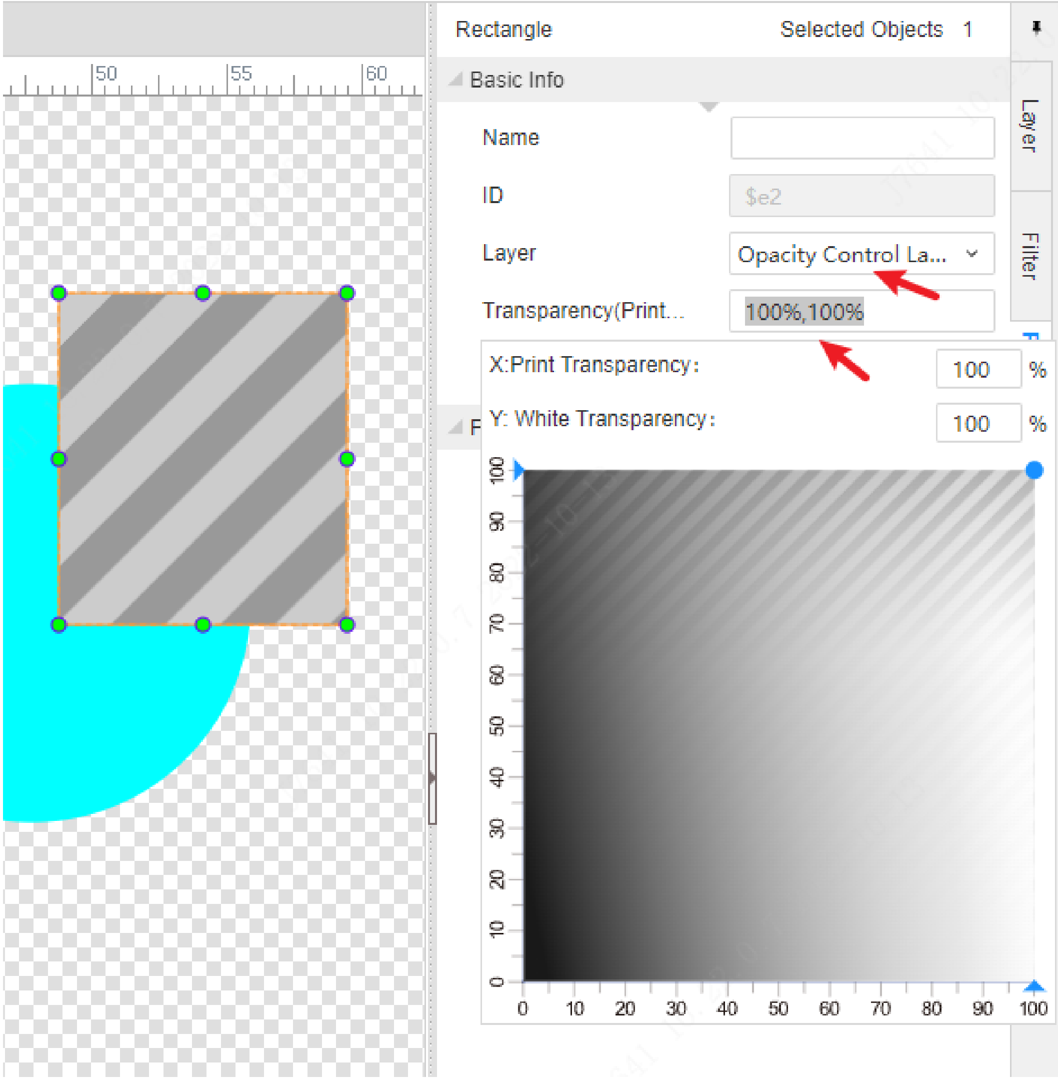


- **Transparent Control layer**: The graphics of the transparent control layer will affect the transparency of the print layer and white bottom mask layer in the area.
    - The higher the value, the more transparent. (100,100) is fully transparent, (0,0) is opaque. For specific settings, please refer to the color card.



    - **The translucency of white color is different from the translucency setting of other colors**: The translucency of white color is printed through the white color cover layer, so you need to set the cover layer to 80% for the best effect. Transparency is optional. Such as: white translucent (80%, 80%).
    - For the translucent color of other colors, you need to remove the white bottom cover layer, the best effect is to print the full layer, and the color is translucent (0%, 100%). If it is too transparent, you can increase the white background a little bit, you can set it to

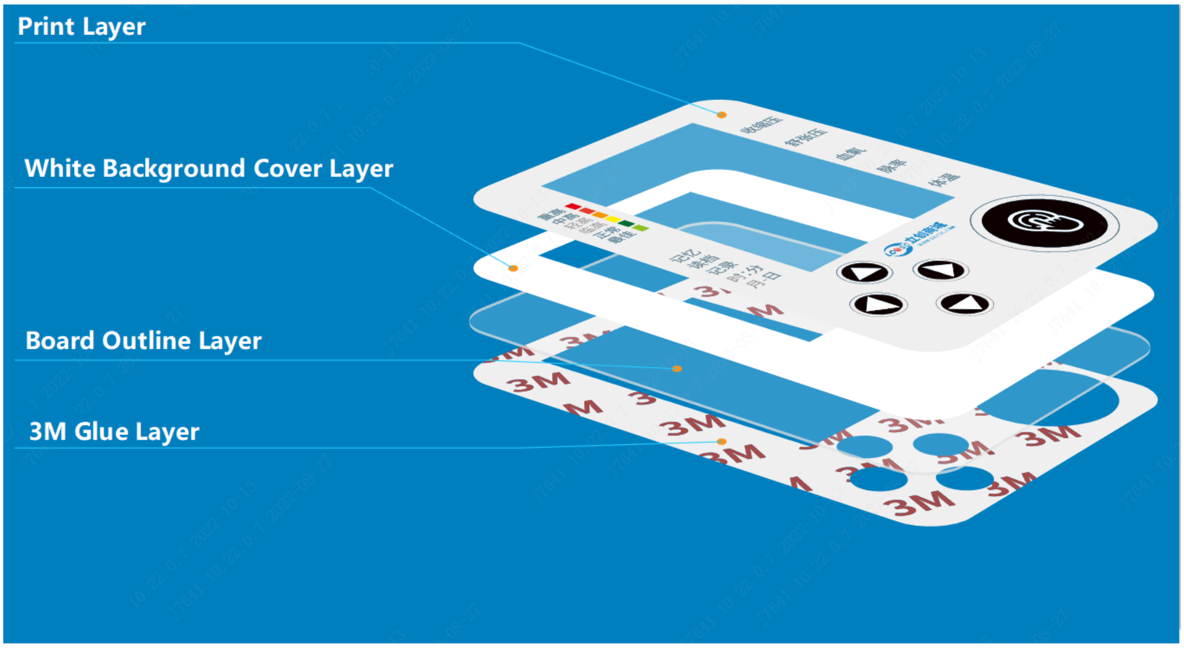
(0%, 90%), different colors have different effects.



- **Print Layer**: The layer used to print ink, it will not be printed by default without drawing, just the raw material itself. The graphics drawn on the print layer have a white background mask by default, and the general shading is used. If you want to adjust the opacity, you need to use the transparent control layer to achieve.
- **Glue Cutout Layer**: It is used to draw additionally where the adhesive will be removed. By default, the system will cut out the adhesive in the transparent control area and the opening area.
- **Lighting Simulation layer**: It is used to simulate the lighting effect. After drawing, you can view the effect in the 3D preview, but it does not actually affect the drawing.

Layer description

Click the thumbnail below the layer to view the layer description.



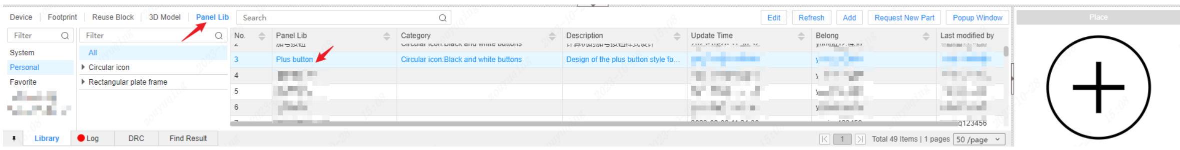
Tip:If the conventional shading cannot meet the needs, and the surrounding area should be more shading, you can choose strong shading when placing an order, and black ink will be printed in the non-transparent and non-translucent areas to block the light. No need to draw your own pictures. By setting different layers and different attribute parameters of primitives, the final required production effect can be obtained.

Bottom Panel - Panel Library

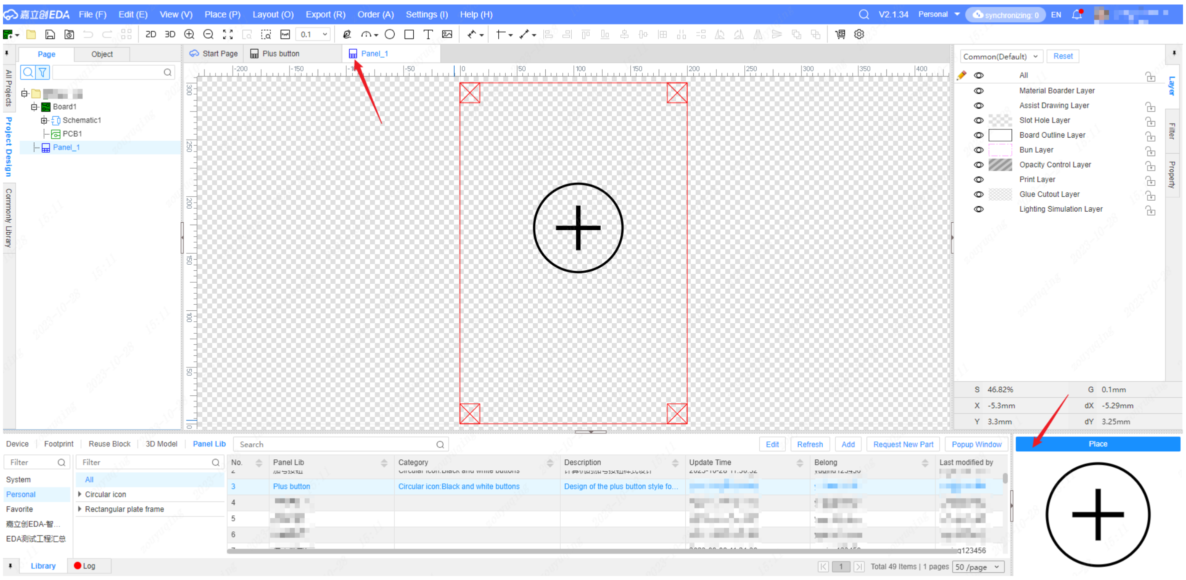
Bottom Panel - Panel Library

Bottom panel library

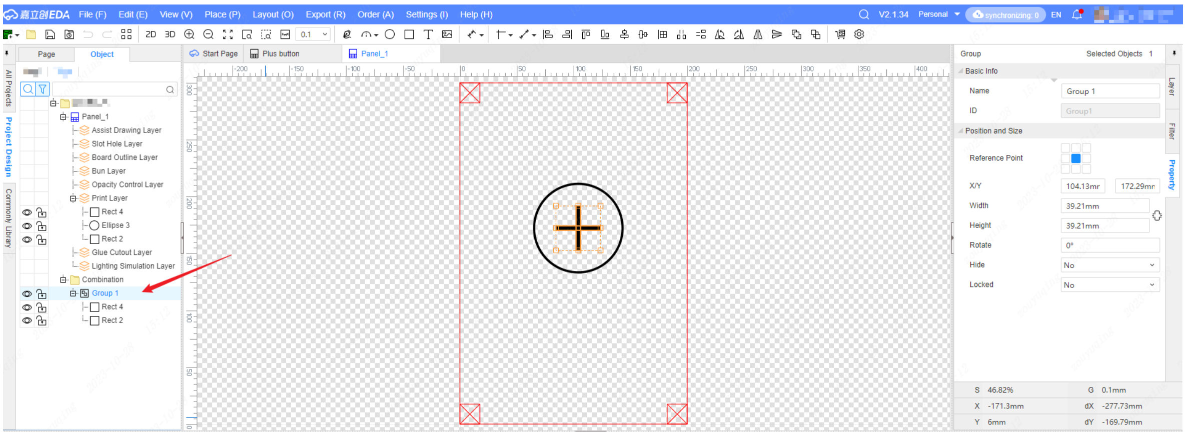
After completing the drawing and saving of the panel library, the panel library can be found through the bottom library.



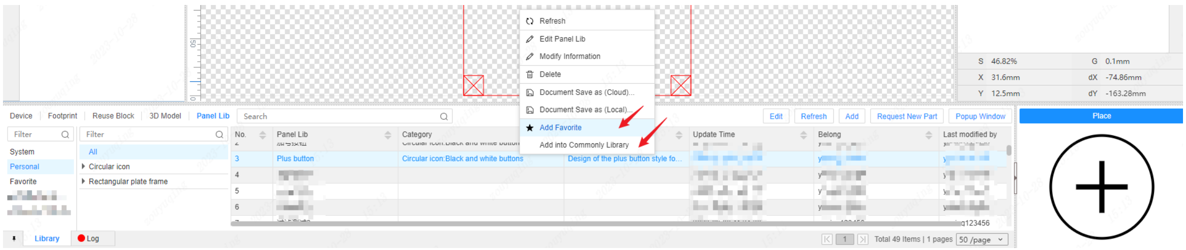
The panel library can only be placed on the panel canvas. When the current editing page is a panel type document, the place button on the preview window at the bottom panel library can only be operated



After being placed on the panel canvas, the primitive will retain all the properties of the panel library when it was drawn. For example, if it was originally combined, it will still retain the combined properties when placed on the panel.



After selecting the specific panel library in the bottom library, right-click to operate the collection of the panel library, and you can also add it to the frequently used library.

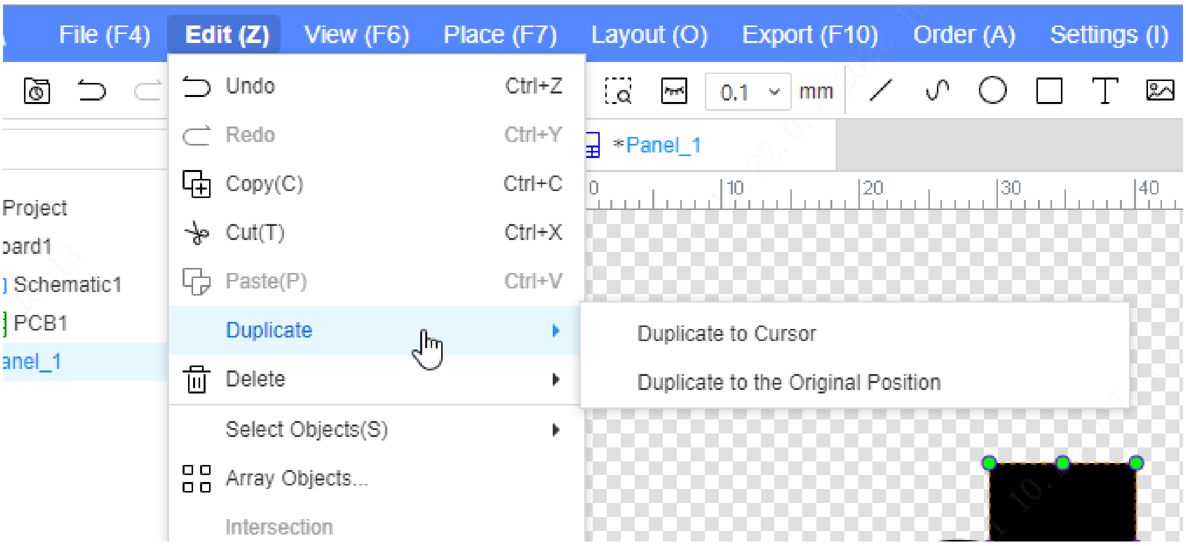


Duplicate

The duplicate function is similar to the commonly used copy and paste, which duplicates the selected object on the canvas of the editor, and supports duplicateing to the cursor and the original position.

Instructions:

- selected element - Top Menu - Edit - Duplicate

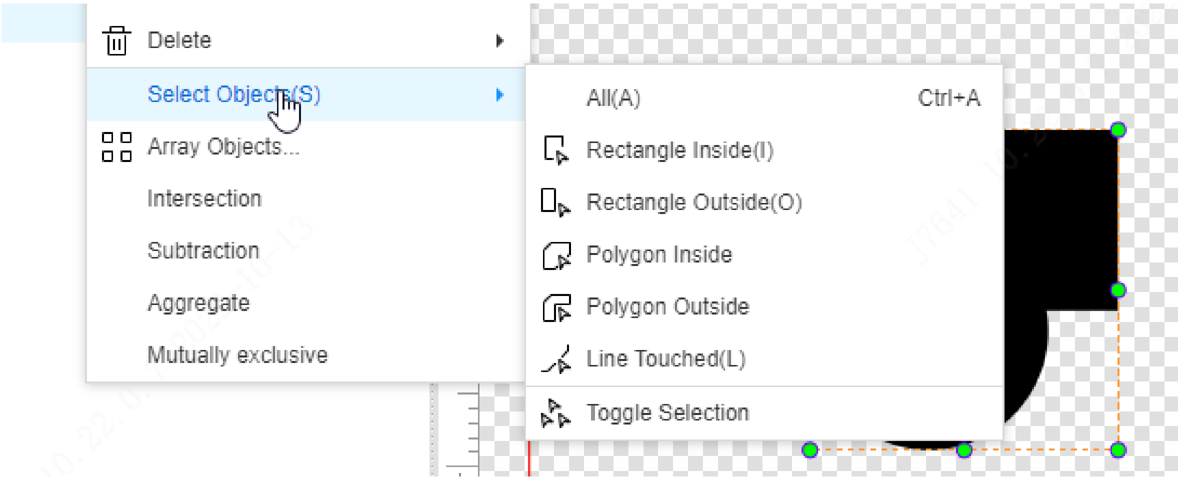


The primitive will generate a copy in the original position, because the shapes overlap, you need to manually drag it away to see it.

Select Objects

select object

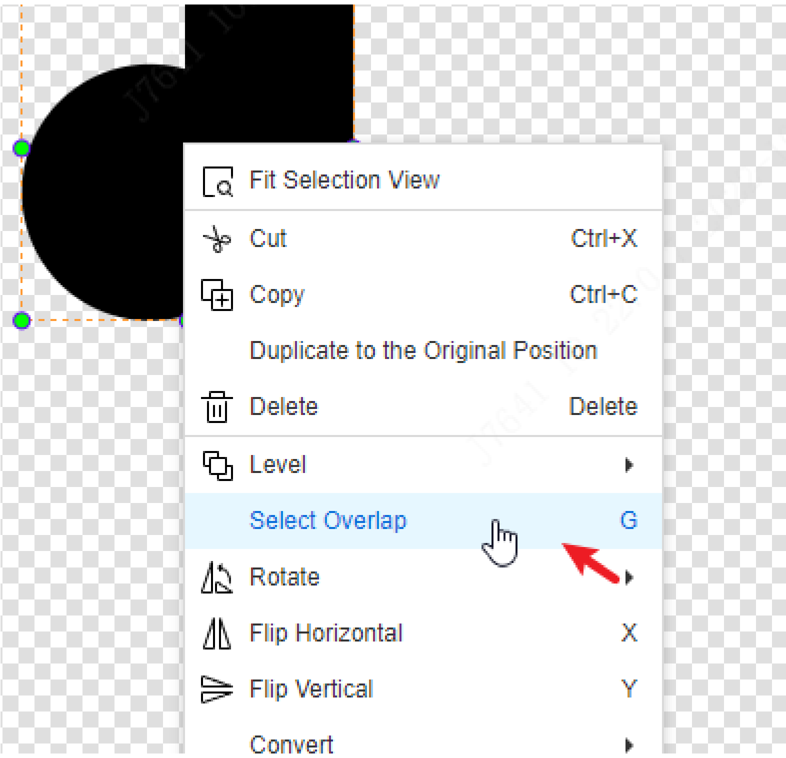
Supports a variety of ways to select objects, supports selection methods inside rectangle, outside rectangle, inside polygon, outside polygon, and touching lines.



Switch selection: After clicking, you can continuously select on the canvas without pressing the CTRL key.

select overlapping objects

When two primitives are overlapped, it may not be easy to select the desired one by clicking and picking with the mouse. You can use the selection overlap function. Shortcut key G



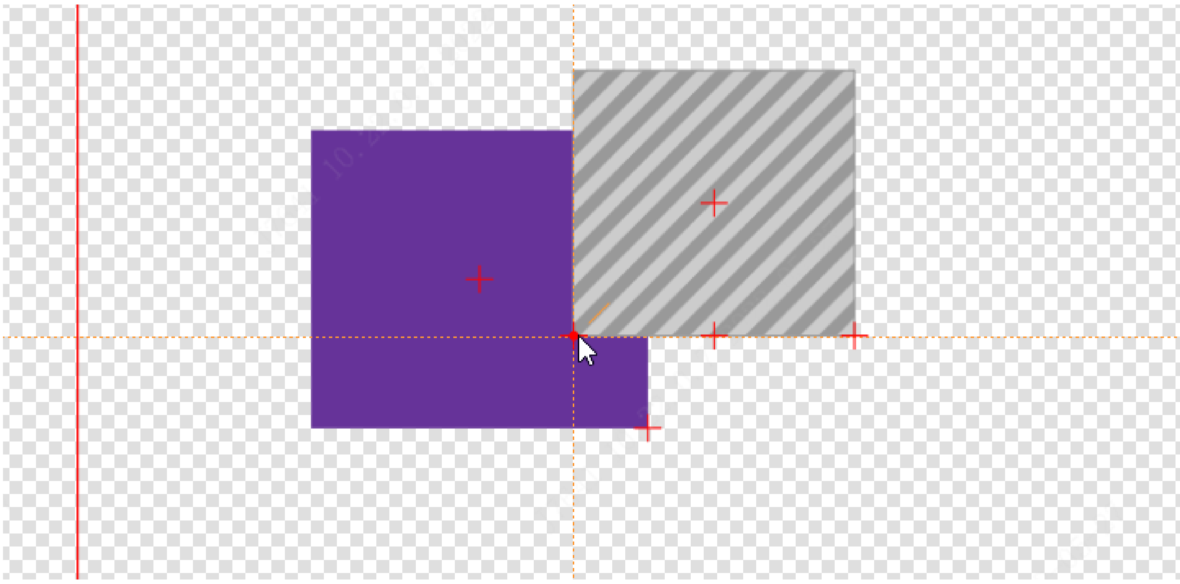
Use shortcut keys to continuously cycle through selected elements around the mouse.

Snap

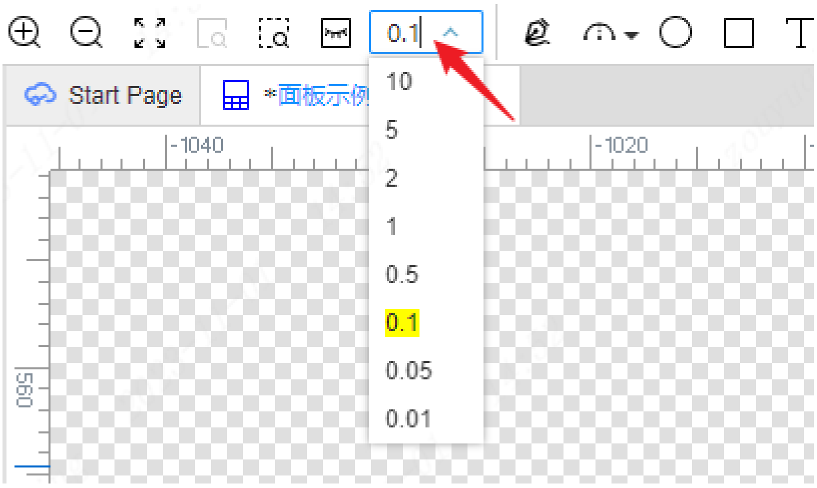
Snap

When drawing primitives, it supports the display of small adsorption cross marks and red alignment guide lines. When moving primitives, the cursor or the adsorption point of primitives can be adsorbed at the positions where the adsorption marks appear.

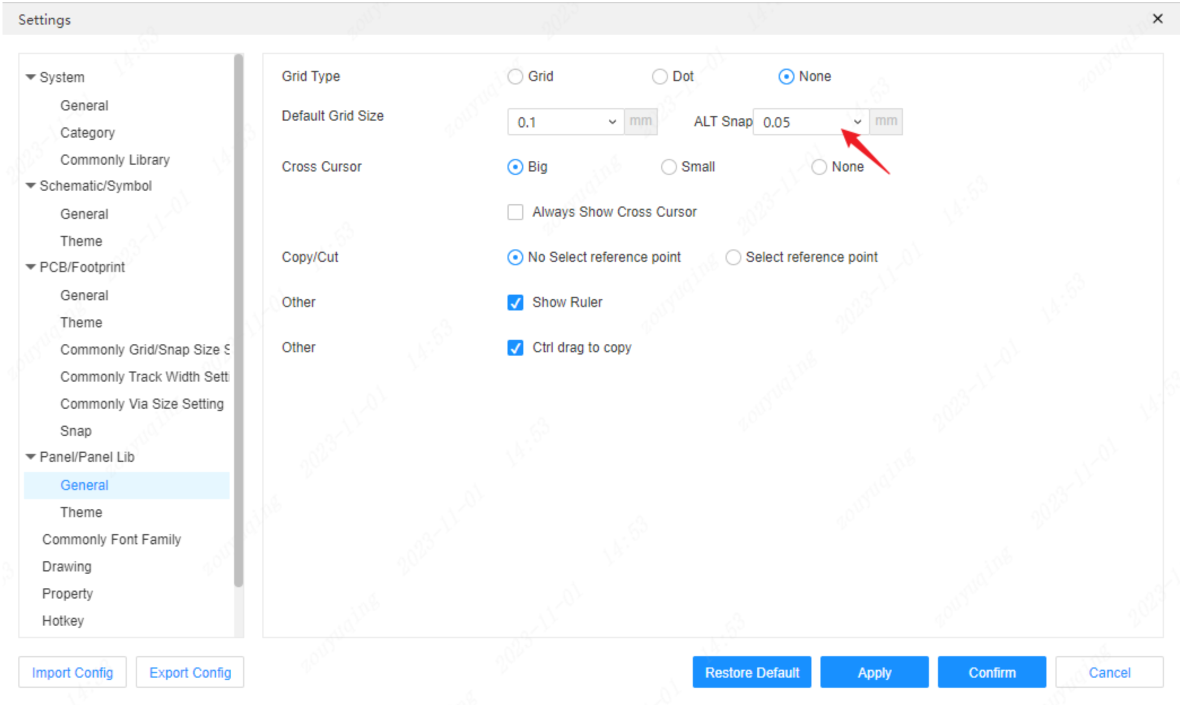
The adsorption is enabled by default, and it is not supported to turn off the adsorption.



The grid adsorption distance can modify the grid size with the top toolbar.



Setting - Panel/Panel Library - General , ALT grid size can also adjust the default grid size.



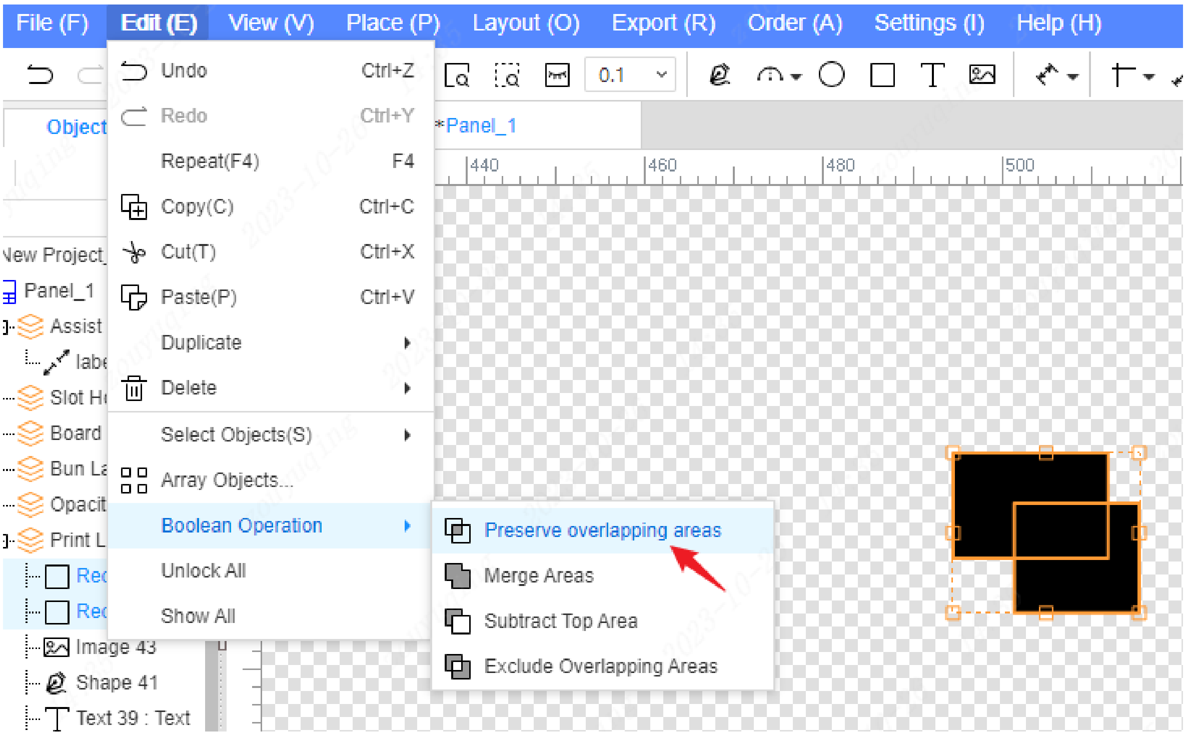
Boolean Operation

Reserve overlapping area

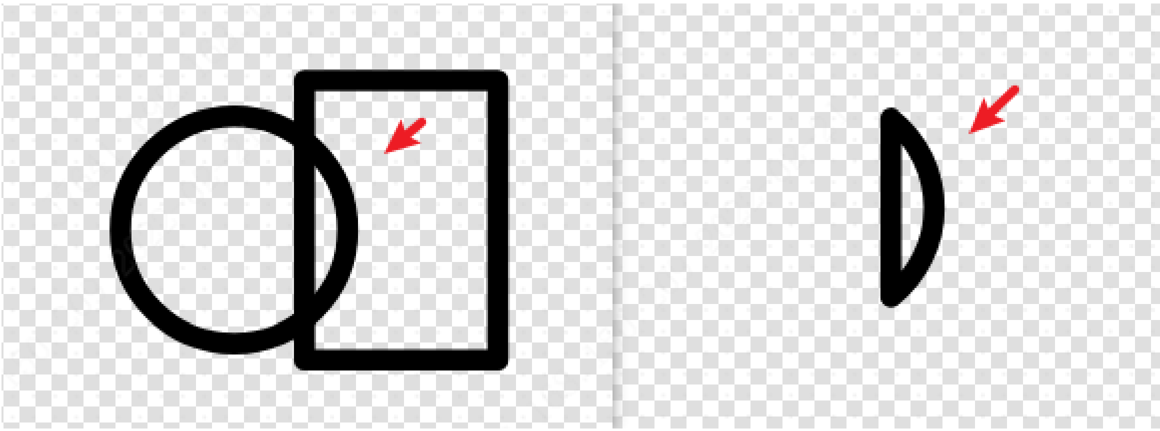
By retaining the overlapping area function, the intersection of two elements can be intercepted, which can generate many difficult-to-draw graphics. Simply put, it is to take the place where A and B intersect.

perating instructions:

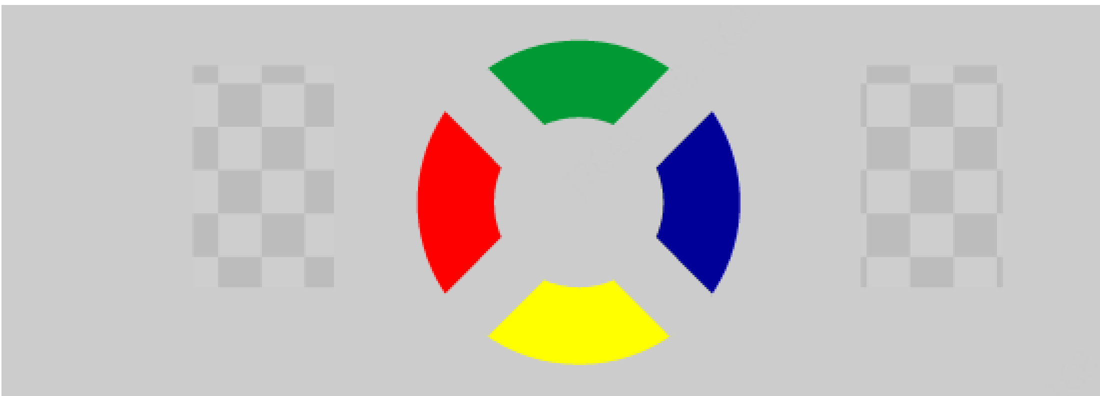
- Select Element - Top Menu - Edit - Boolean Operation - Reserve Overlap Area



Take the overlapping area of two elements in the following figure:



As shown in the figure, the shape of the arrow button element can be generated by taking multiple intersections.

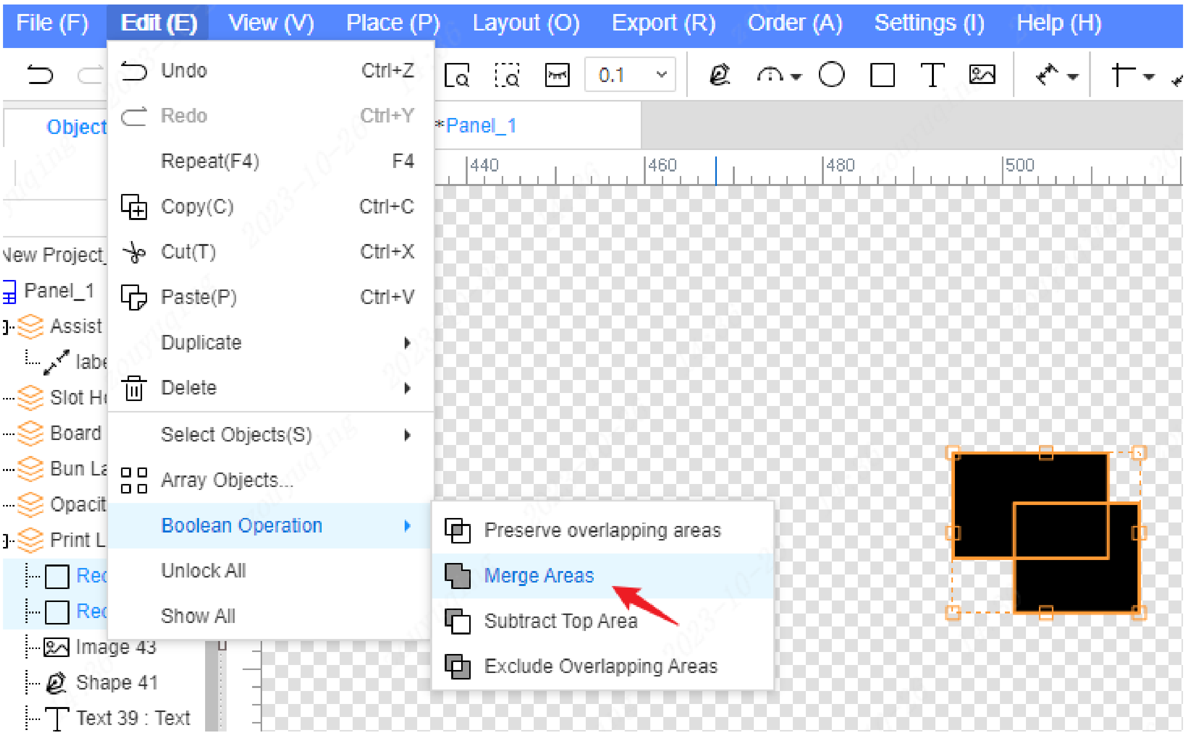


Merge area

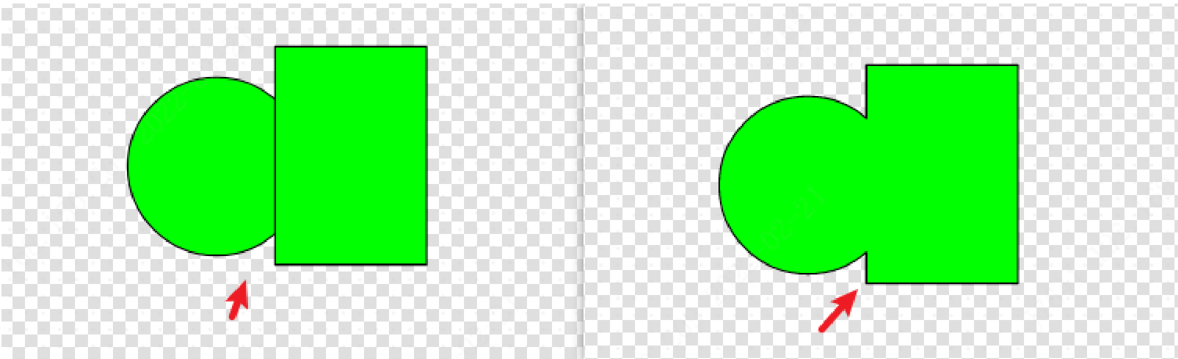
Use the merge area function to merge two different shaped elements into one element, and merge the outer edges of the elements. Simply put, it is to merge A and B together.

Operating instructions:

- Select Element - Top Menu - Edit - Boolean Operation - Merge Area



After selecting multiple intersecting elements, click the Merge Area menu.



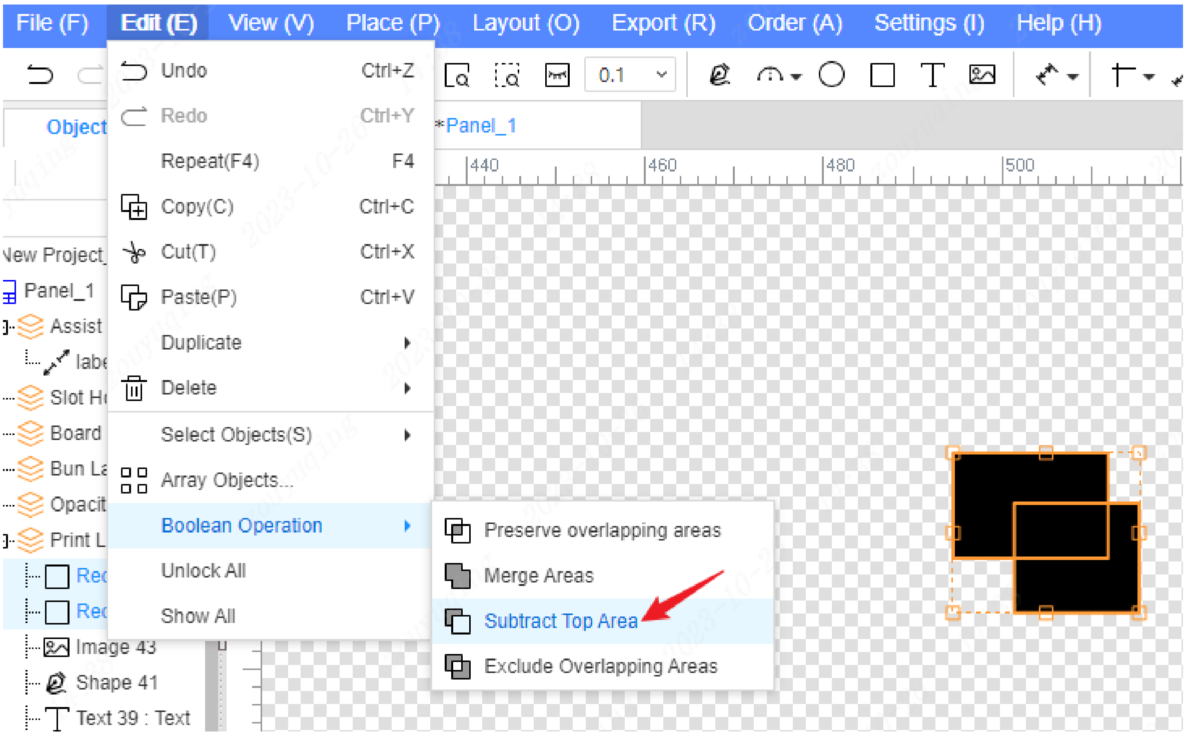
Some difficult-to-draw outlines can be generated by merging regional functions.

Subtract the top area

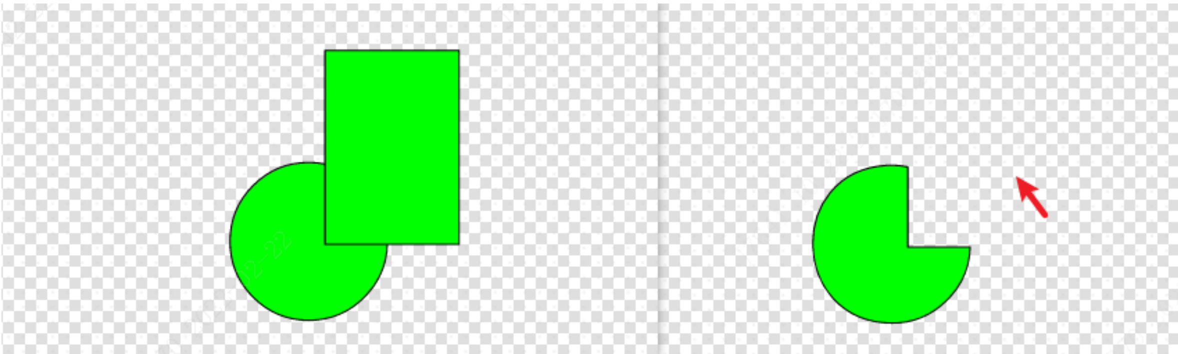
By subtracting the top-level area function, the internal intersection of two different-shaped elements and one of the elements can be removed. If A is at the next level of B, subtracting the top-level area will retain the part of A-B.

Operating instructions:

- Select Element - Top Menu - Edit - Boolean Operation - Subtract Top Area



As shown in the figure below, the rectangular part is removed by subtracting the top area and removing the parts that do not need to be retained.



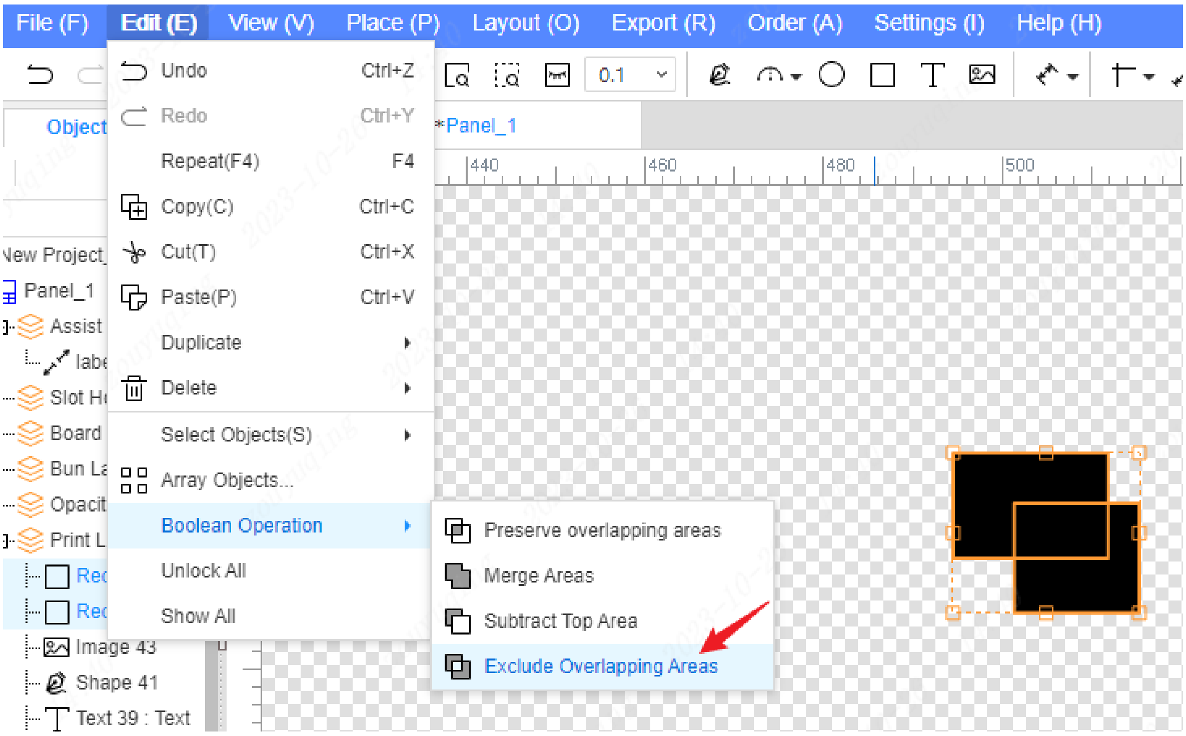
Note: The part of the top-level area that is eliminated is based on the drawing order of the primitive, and the part that is drawn later will be eliminated.

Exclude overlapping area

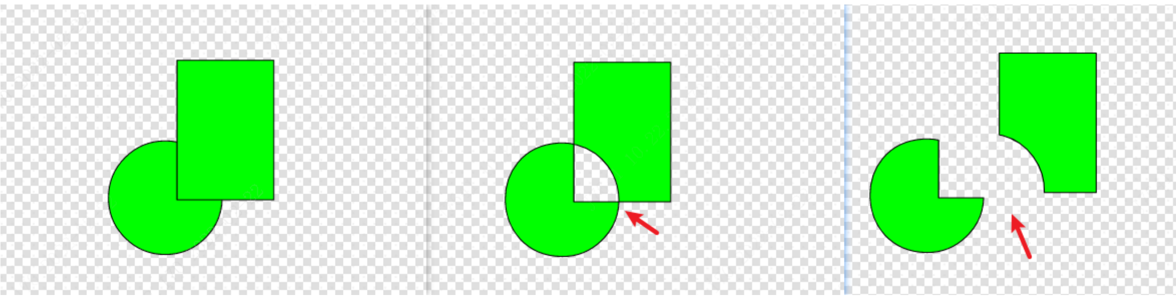
By excluding the overlapping area function, the intersecting parts of A and B can be removed, and the non-intersecting parts can be retained. The intersecting parts of different-shaped elements are removed.

Operating instructions:

- Select element - Top menu - Edit - Boolean operation - Exclude overlapping area

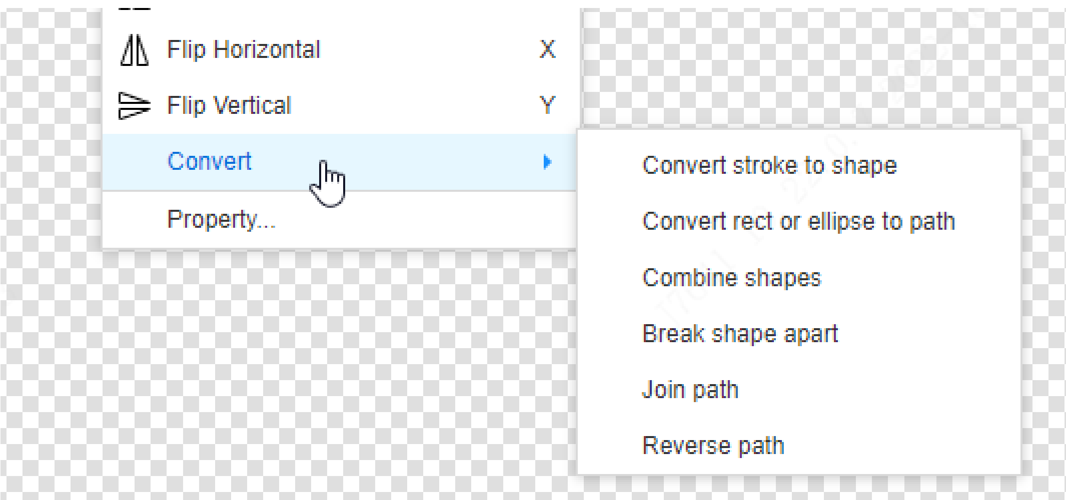


Select multiple elements and click the Exclude Overlap Area menu.



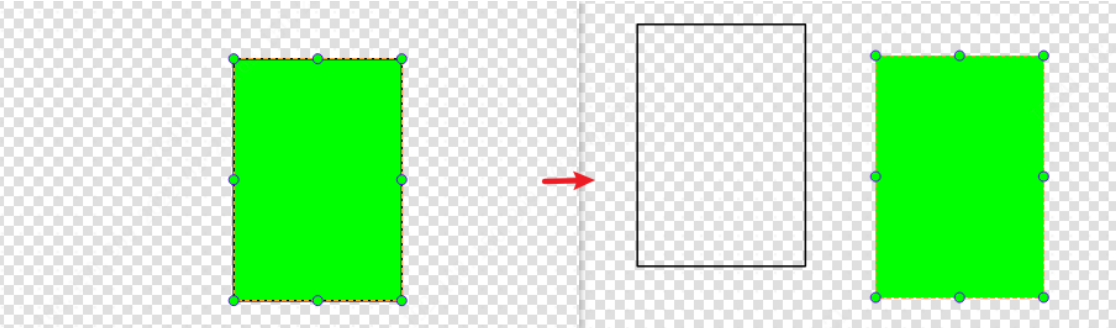
Convert

After selecting multiple primitives, right-click to convert the primitive shape type.

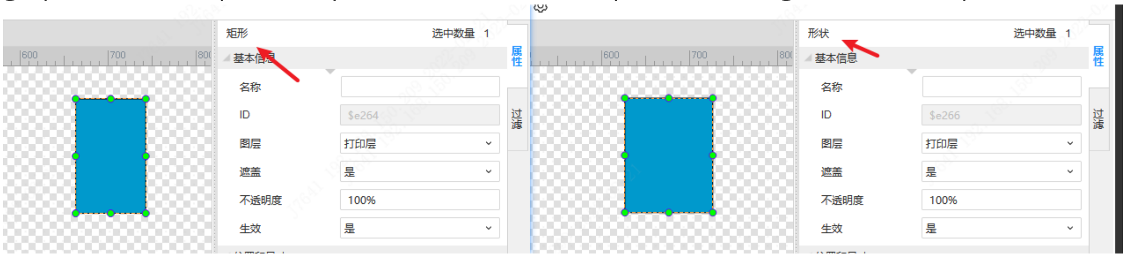


Function Description:

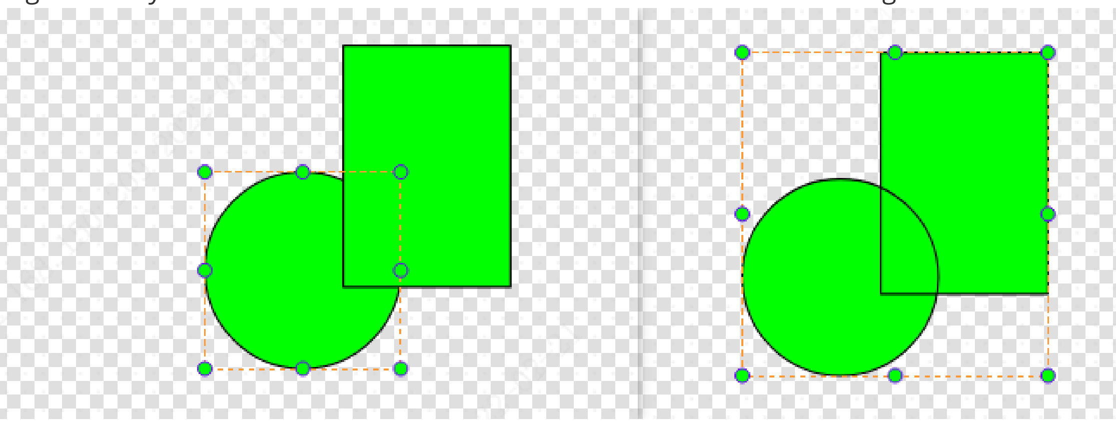
- Convert stroke to filled shape: Generate a filled shape based on the outer outline. As shown in the figure: The rectangle is converted into two primitives, one with no filling in the outer contour, and one with filling and no outer contour.



- Convert rectangle and ellipse to path: The default rectangle and ellipse are independent graphics, not composed of paths, and become shapes after being converted to paths.



- Shape Merge: Similar to the combination function, it combines different primitives or shapes together. Pay attention to the selection effect and outline lines in the image below.



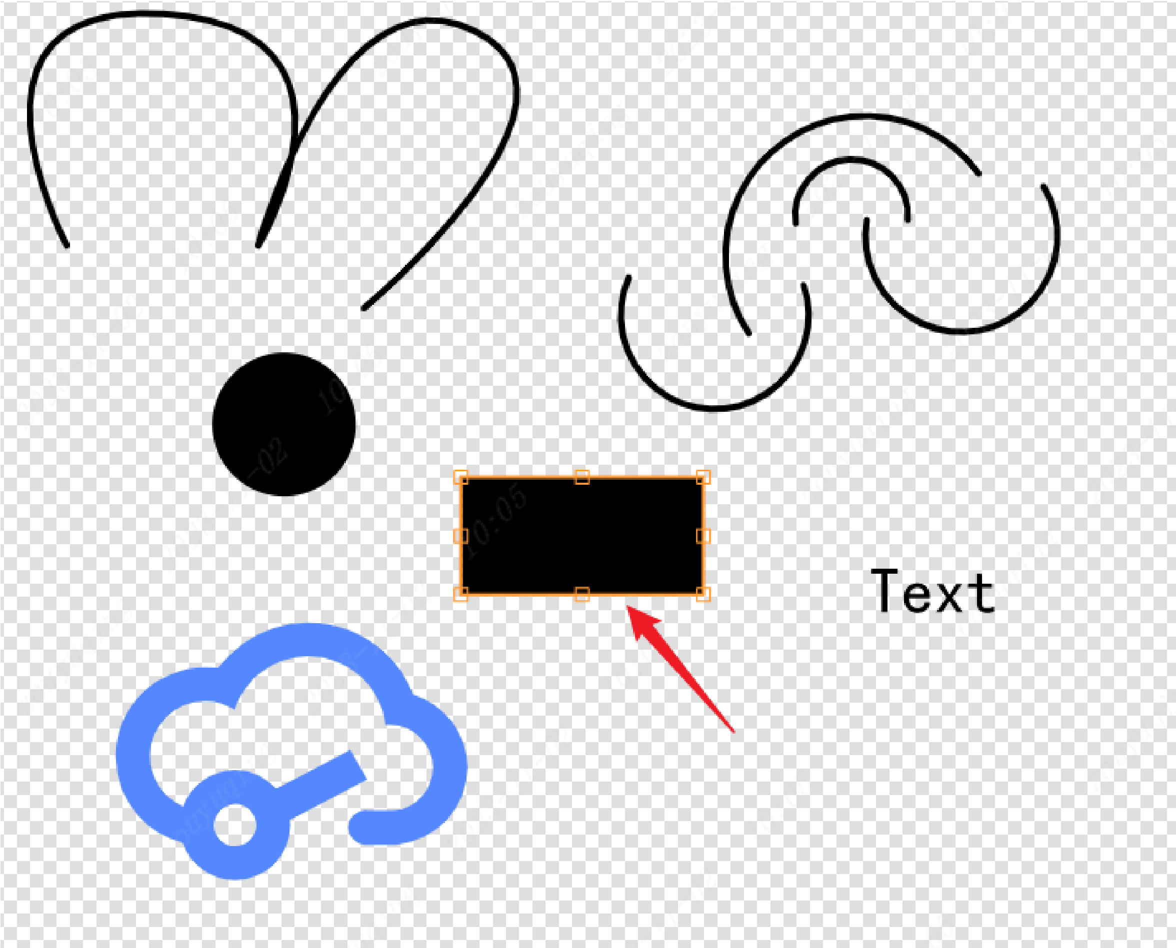
- Shape break up: break up the merged primitives.

canvas

Canvas

Eight-point control

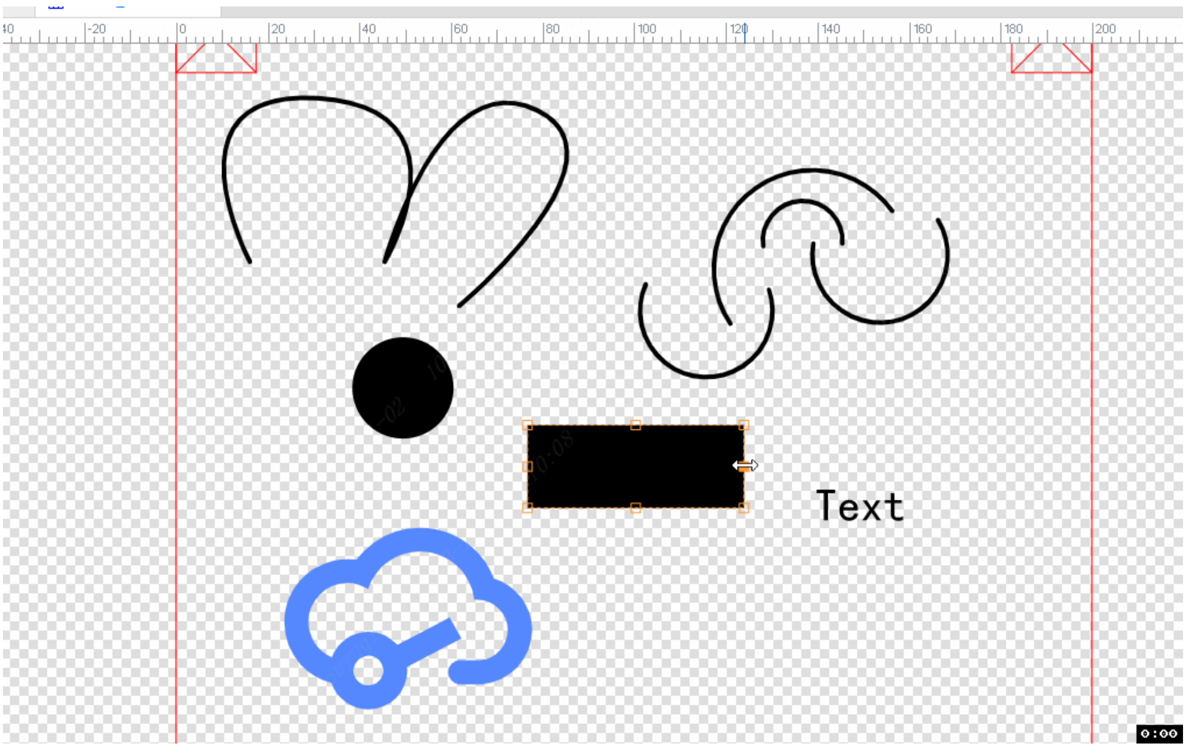
In the panel, the completed**Shape**、**Arc**、**Ellipse**、**Rectangle**、**Text**、**Image**these ordinary elements are selected on the canvas to display the 8-point control box.



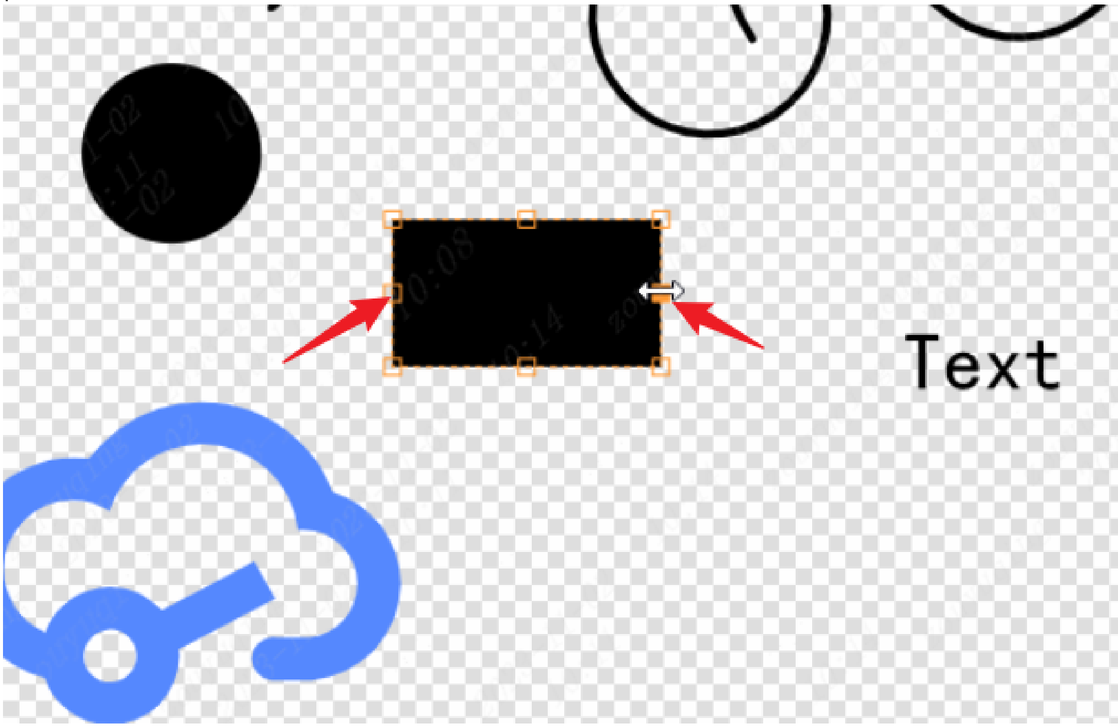
The smart dimension, auxiliary line, auxiliary point, and dimension of the assist drawing layer do not provide 8-point control.

Stretching

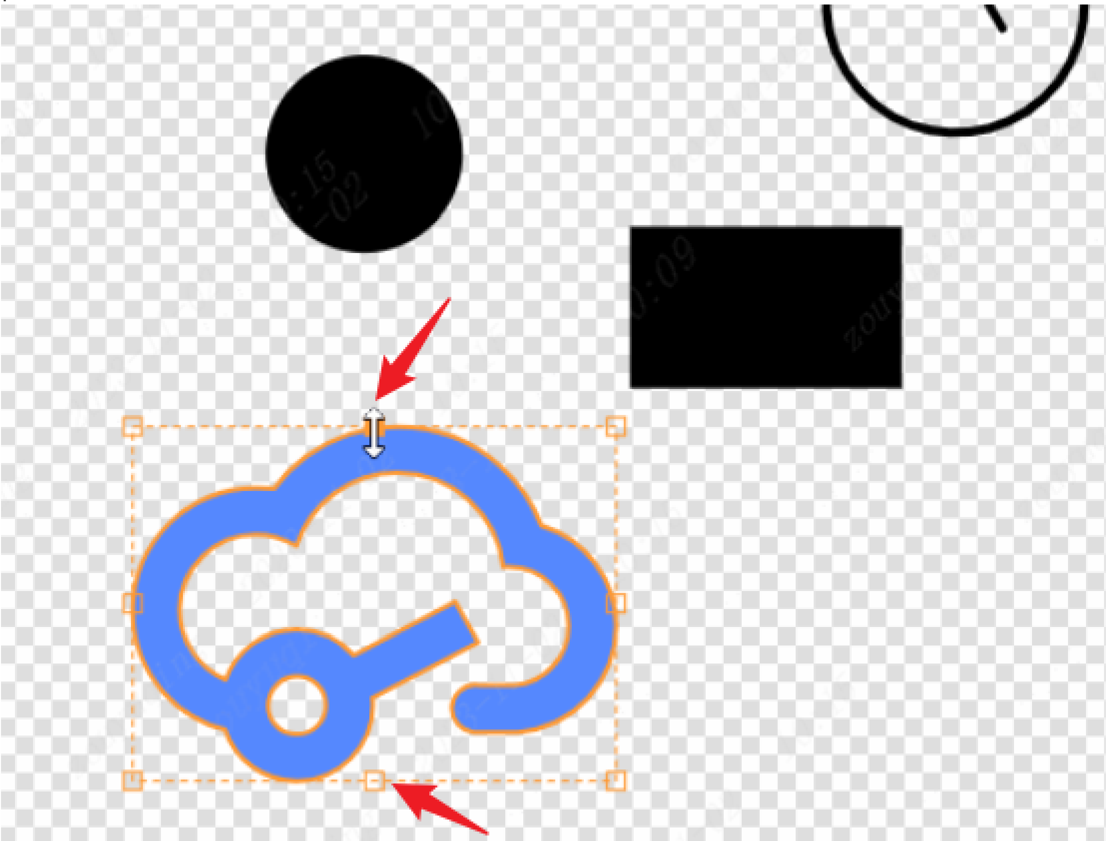
Eight control points can control the stretching of the primitive. Move the mouse to the corresponding control point to display the cursor of the corresponding stretching type, and hold down the left mouse button to perform the stretching operation.



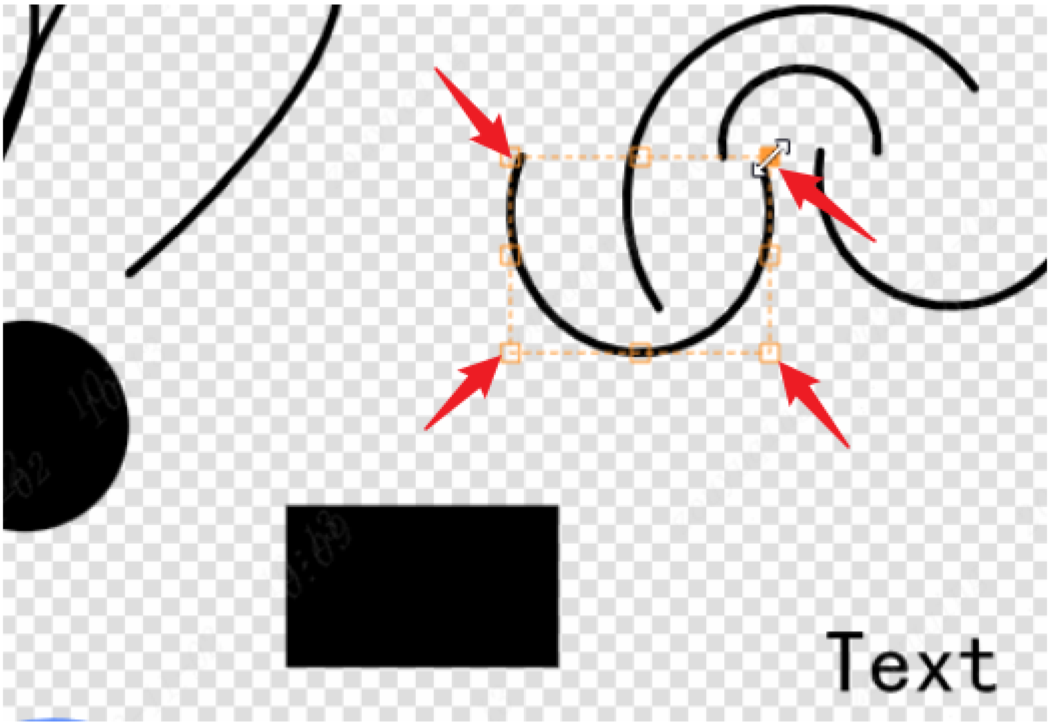
- The left and right center are the horizontal stretching control points, which will maintain the position of the symmetrical edge. The horizontal stretching changes the width of the primitive.



- The vertical stretching control points in the upper and lower middle will also maintain the position of the symmetrical edge, and the vertical stretching will change the height of the primitive.

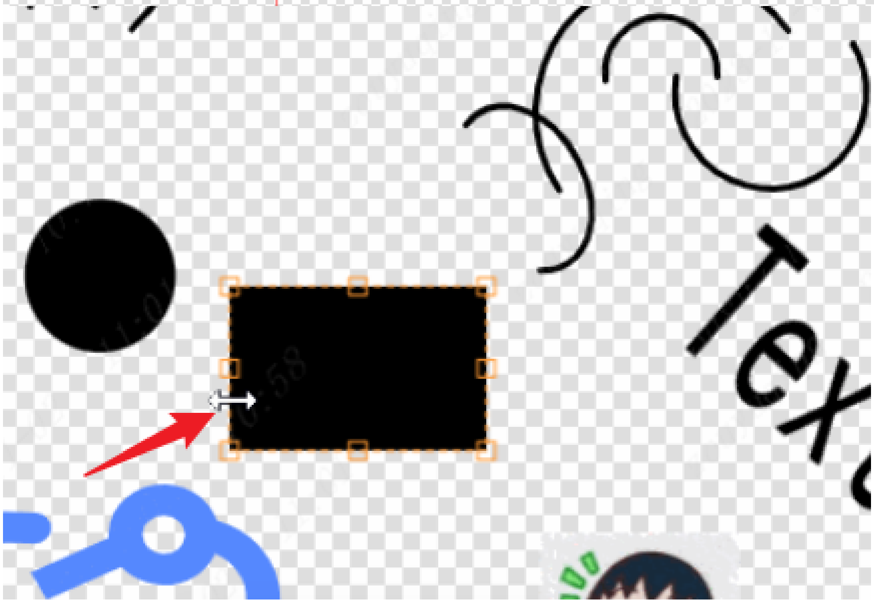


- Dragging the control point on vertex 4 for stretching will simultaneously change the width and height of the primitive and maintain the position of the symmetry point.



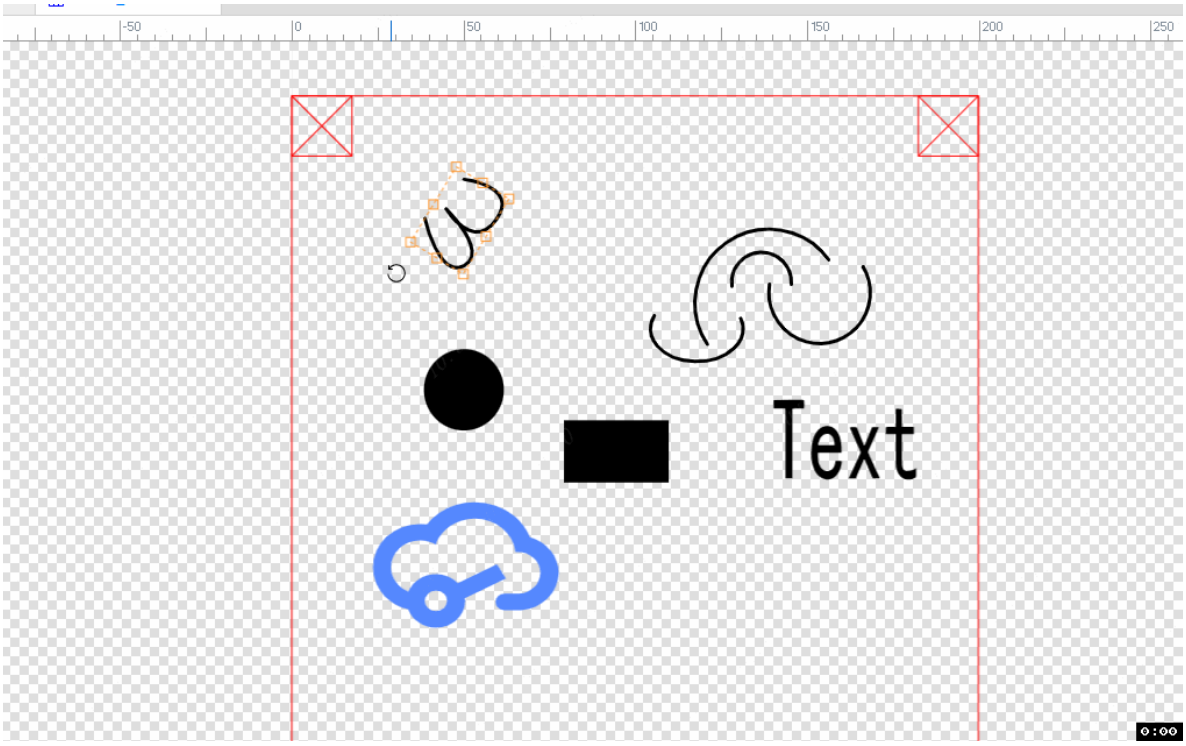
Hold the **key Shift** on the 4 vertices and drag it to keep the aspect ratio of the primitive stretched.

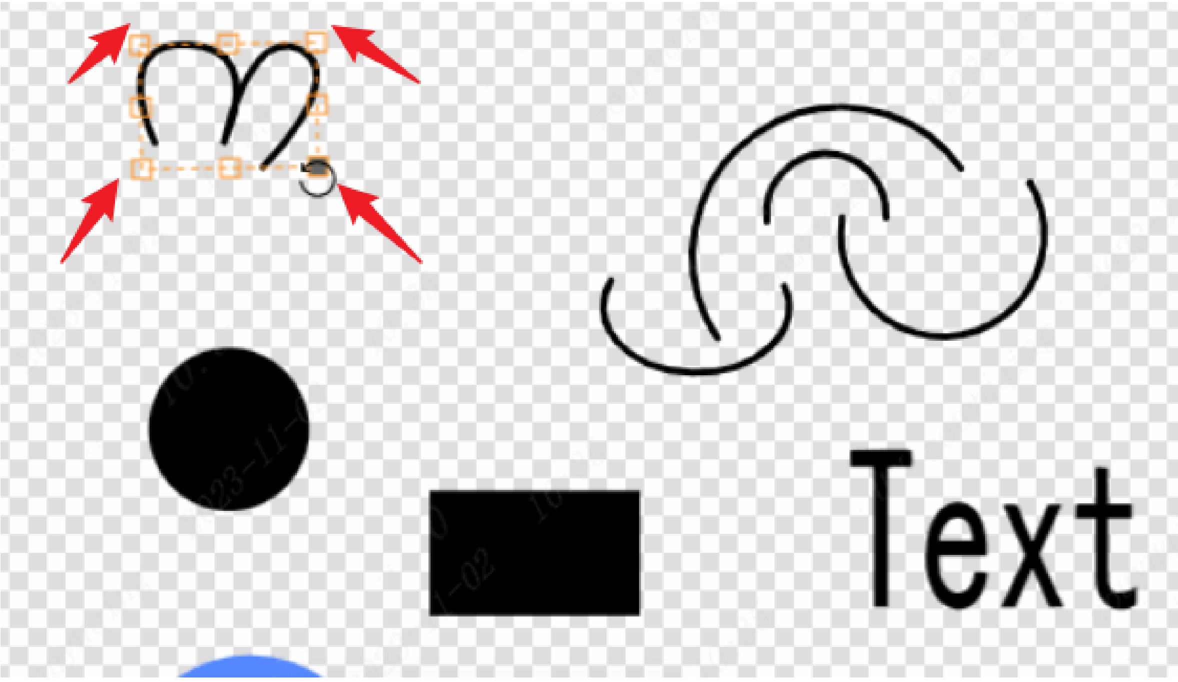
The border controlled by 8 points also supports resizing, and the function of adjusting the size of the control points is the same. Move the mouse over the border to display the corresponding type of stretched cursor.



Rotate

The rotation angle can be adjusted at the 4 vertices controlled at 8 o'clock. Move the mouse to the outside of the 4 vertices to display the rotating cursor. Press and hold the left mouse button to rotate.

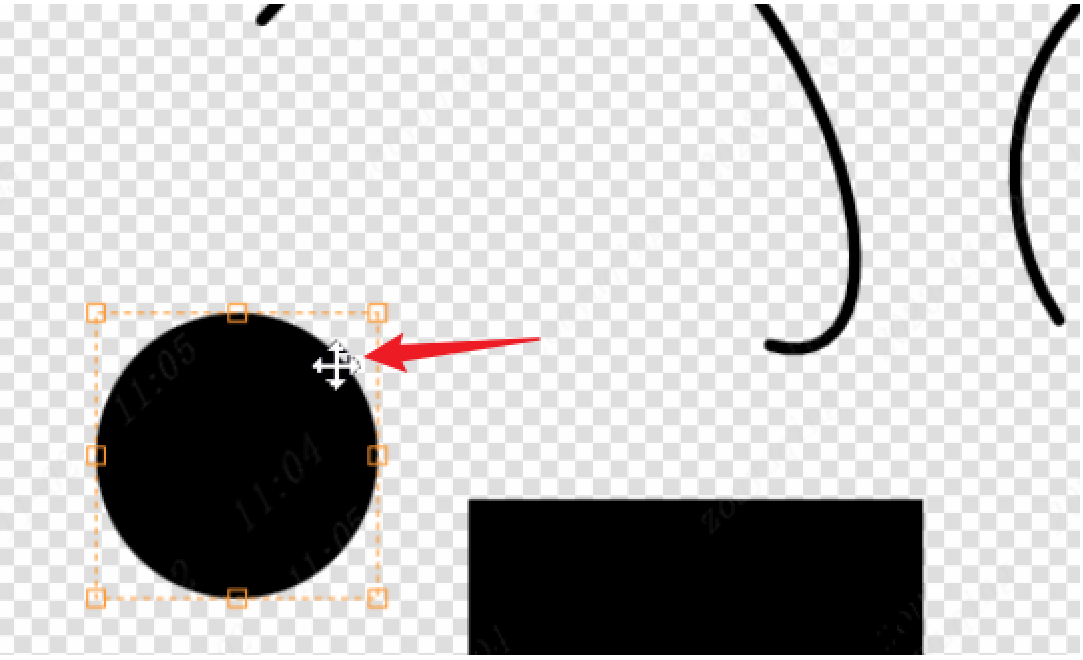




- When the mouse moves to the outside of the vertex 4 to display the rotating cursor, the Shift key can be rotated in 45 ° steps.

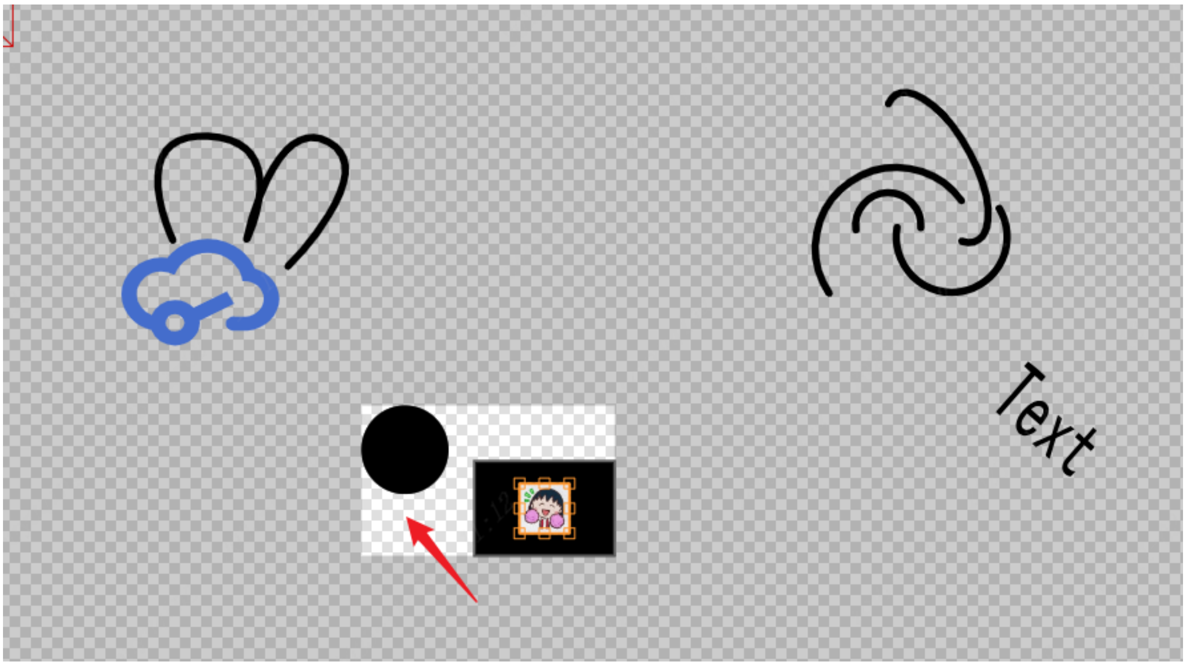
Drag

After selecting the element, the areas in the 8-point control box are all drag functional areas. Move the mouse to the 8-point control box to display the drag cursor, and hold down the left mouse button to drag the element.



Mask Mode

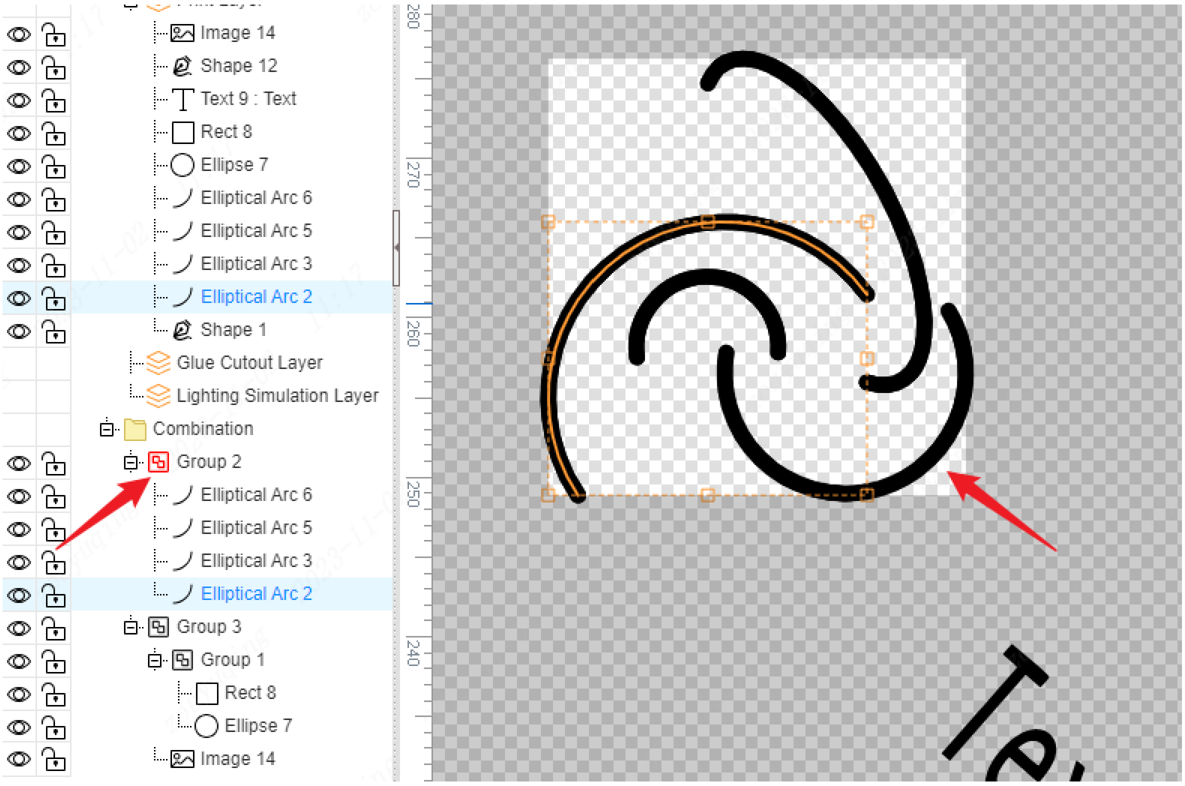
Currently, not only are there masks in the panel, but also masks for arcs and shapes. When entering the mask mode, the canvas displays a gray mask effect outside the mask area, and the normal brightness inside the mask is displayed.

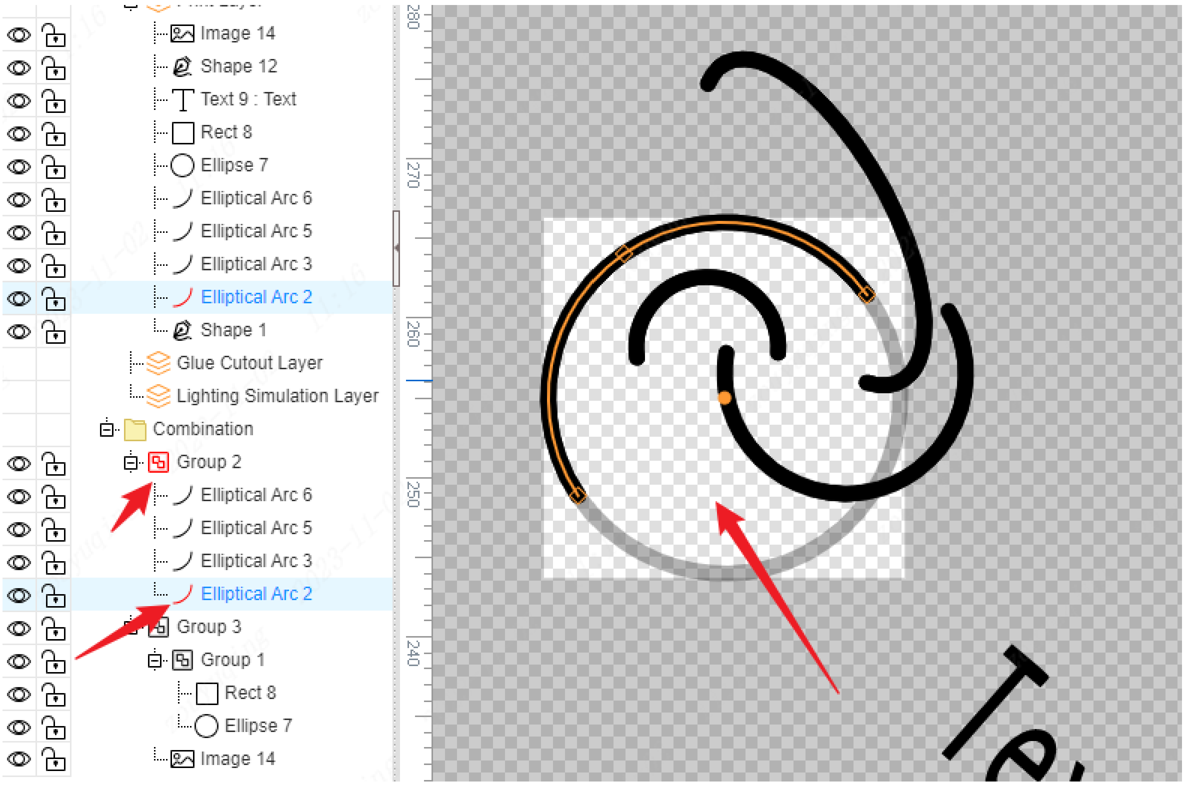


Masks are double-clicked to enter, double-clicked or pressed Esc to exit.

The elements inside the mask can be selected and edited normally, while the elements outside the mask cannot be selected and operated temporarily.

After combining arcs or shapes, double-click the first time to enter the combination mask, and double-click again to enter the arc or shape mask. Exiting will also double-click to exit the arc or shape mask first, and double-click again to exit the combination mask.





Draw Interaction Rules

Right click to exit drawing

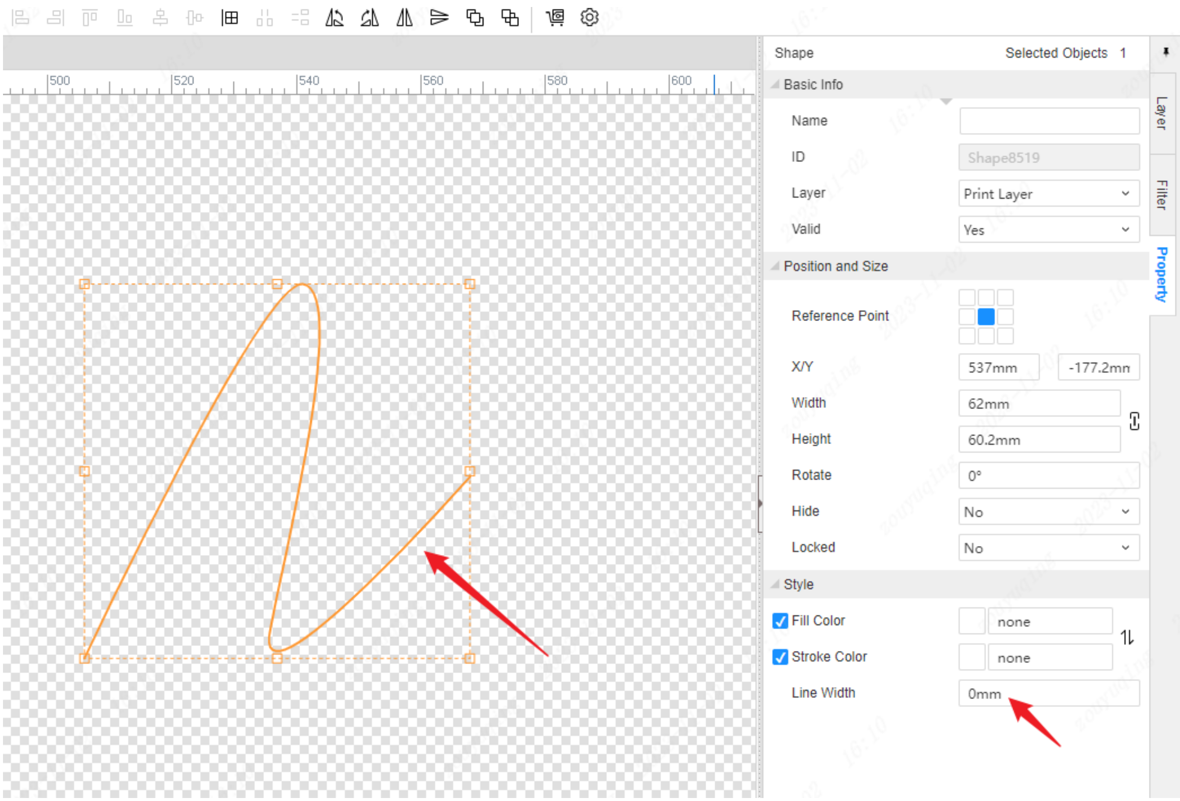
All primitive drawing in the panel starts with the left mouse button. The first right mouse button will exit the drawing of the primitive in the current drawing, and the second right mouse button will exit the drawing mode.

Note: Pressing Esc will directly exit the drawing mode.

Select Highlight

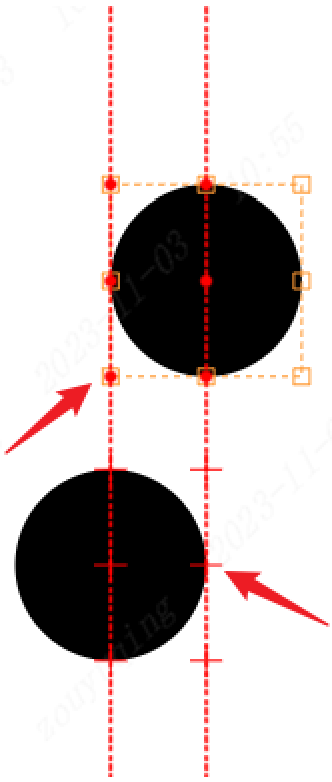
After the panel element is selected, the element outline path will be highlighted. The highlighted line width style is fixed and does not follow the element's own line width. The color follows the selected theme color, which can be customized in the settings pop-up window.

For example, as shown in the figure below, even if the line width is set to 0mm, it will still be highlighted.



Drag Adsorption

During the dragging process of the panel element, when the key point of the dragged object contacts the key point of other objects on the canvas, adsorption is triggered. A small red cross is displayed at the key point, and when the adsorption distance is reached, adsorption is performed and a small red dot is displayed.

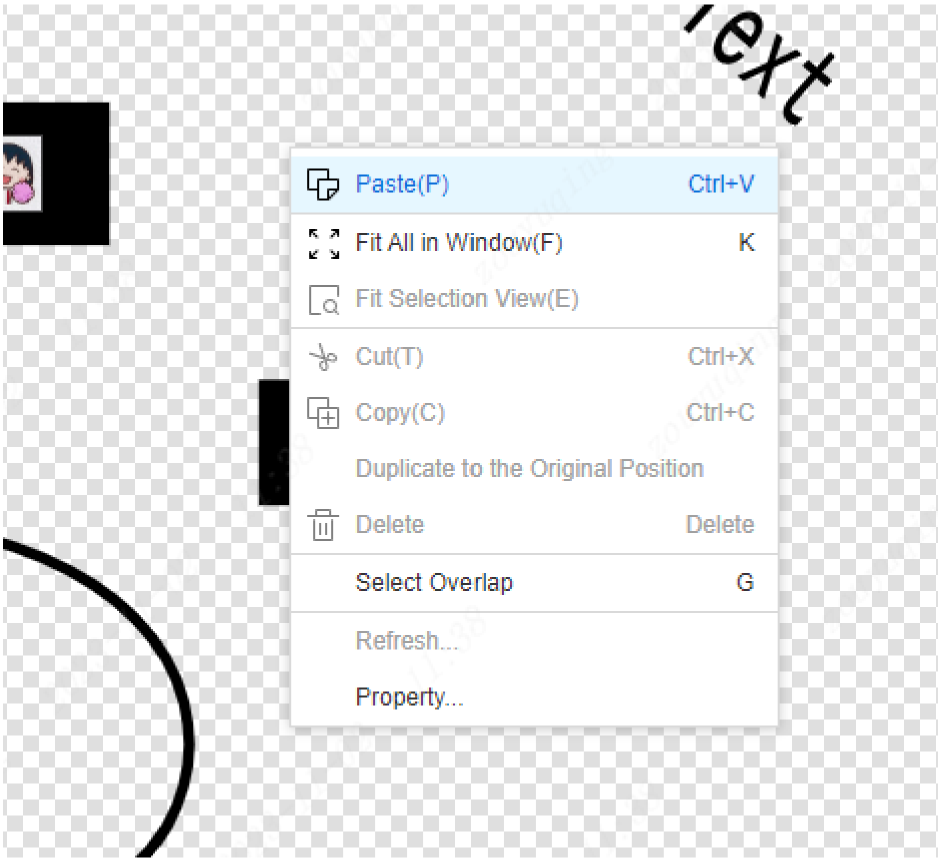


The virtual line that runs through the canvas is the adsorption auxiliary line.

Right-Click Menu

Canvas

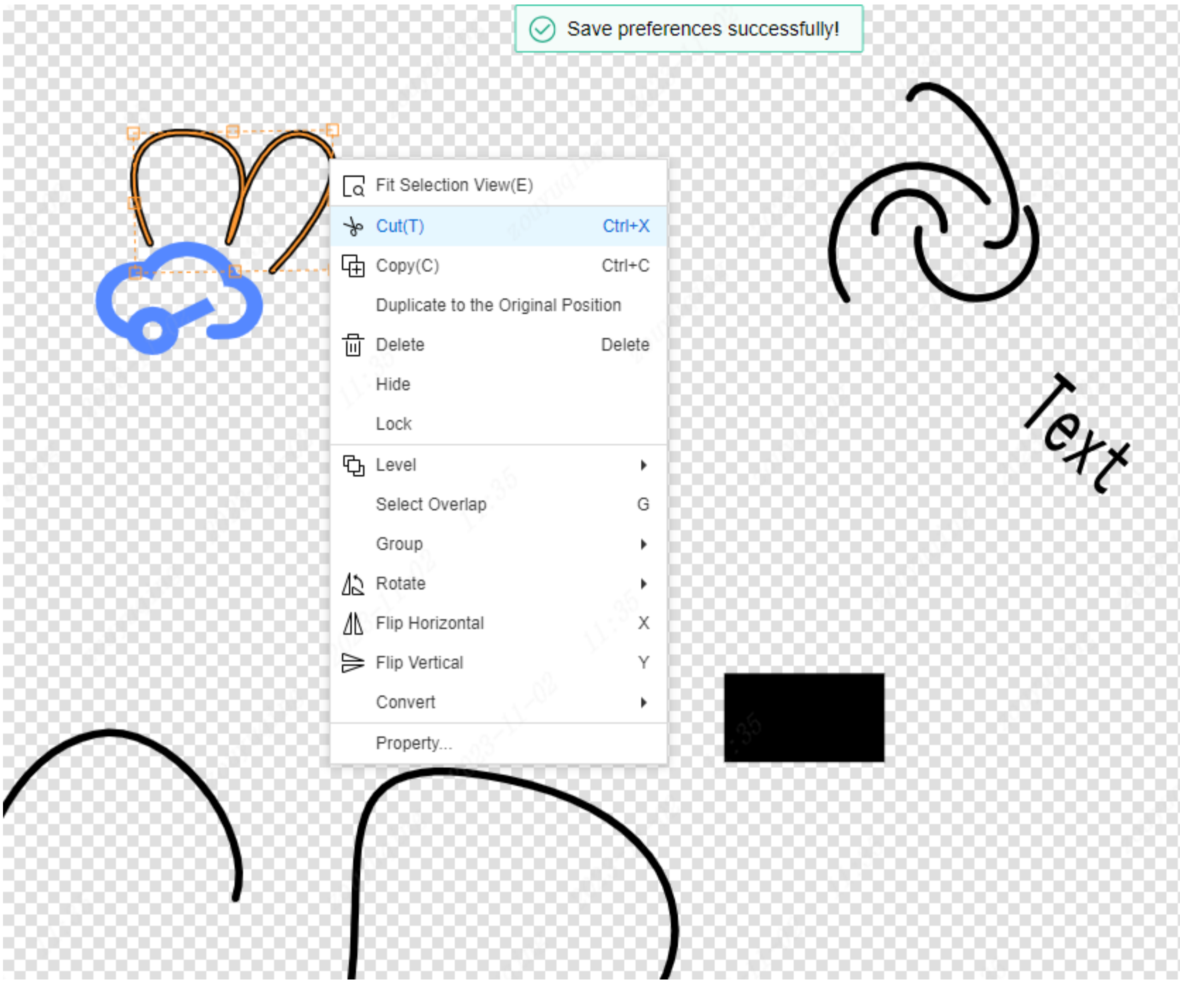
The item displayed in grey out does not support the operation.



- **Paste**: When there is content in the clipboard, the paste menu can be operated, otherwise grey out is displayed.
- **Fit All in Window**: all elements on the canvas will be centered and occupy the entire canvas display, you can see all elements on the current canvas.
- **Select Overlap**: will be selected by the mouse graphics overlap all graphics information is displayed, you can arbitrarily select the occluded graphics, easy to select.
- **Property...**:The property panel on the right will open to view the property information of the current page.

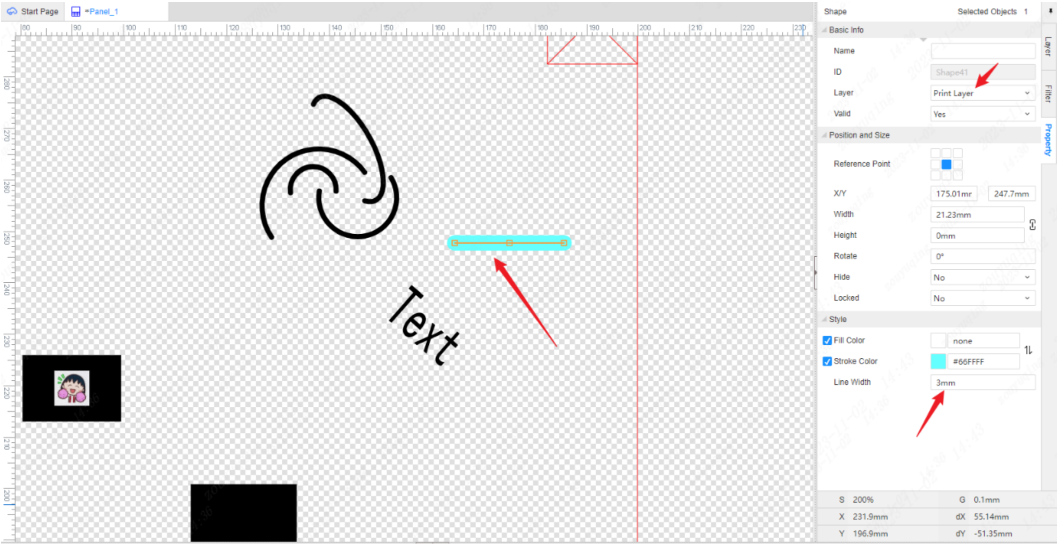
Ordinary primitive

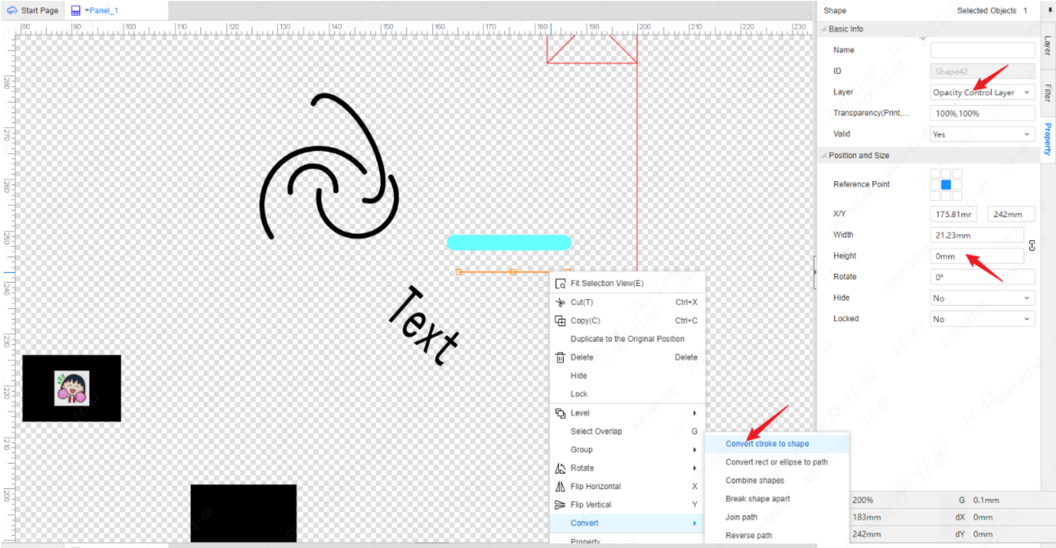
Shape、**Arc**、**Ellipse**、**Rectangle**、**Text**、**Image**these are called ordinary elements of the panel, and the right-click menu items are consistent.

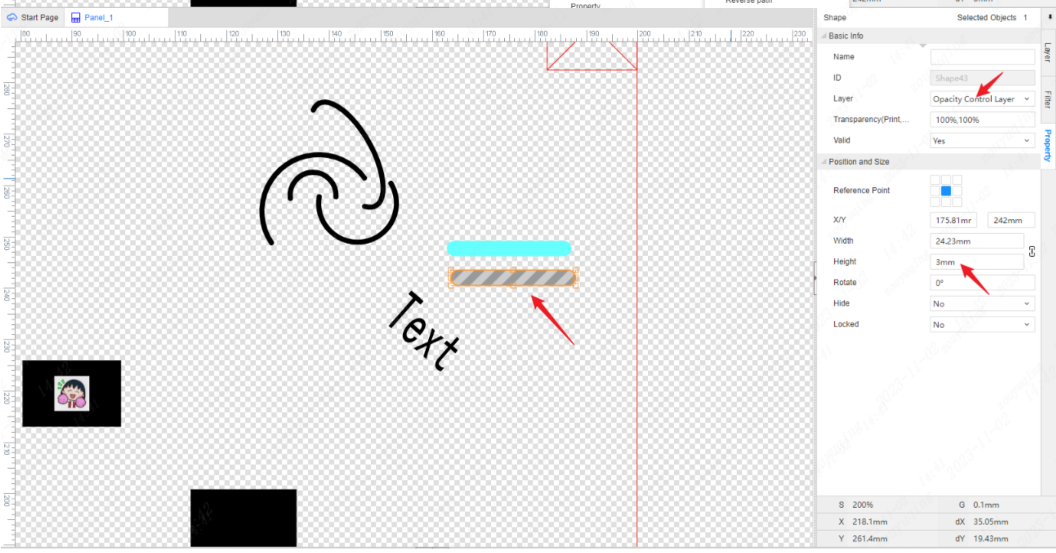


- **Fit Selection View**:Scale the canvas so that the selected object centers and fills the entire canvas display.
- **Cut**： Cut the selected primitive to the clipboard. The cut primitive disappears from the canvas.
- **Copy**：Copy the selected primitive to the clipboard. The copied primitive remains on the canvas.
- **Duplicate to the Original Position**： equivalent to copying and pasting the same graphic in the original position. Equivalent to operations Ctrl + C and Ctrl + V (paste in situ)
- **Delete**：Delete the selected element, usually directly click Delete to delete the operation.
- **Hide**： You can hide the elements on the canvas is not visible, the operation of the hidden menu is not unchecked, there are 8 control box, the right-click menu in the Show menu can be unhidden.
- **Lock** : You can lock the element on the canvas. Similarly, when the locked element is selected, the unlock menu in the right-click menu can be unlocked.
- **Level**： - You can move the graphics of the same layer. (Switch between levels of graphics within an editing layer)
  Note: If you want to display the elements of a certain layer at the top of the visual display, click on the layer on the right to select the corresponding layer as the current editing layer, and the graphics in the corresponding layer will be displayed at the top without being obscured by the graphics of other layers.
- **Select Overlap**: will be selected by the mouse graphics overlap all graphics information is displayed, you can arbitrarily select the occluded graphics, easy to select.
- **Group**：You can create a new combination or add an existing combination. You can also operate to remove the combination and cancel the combination.

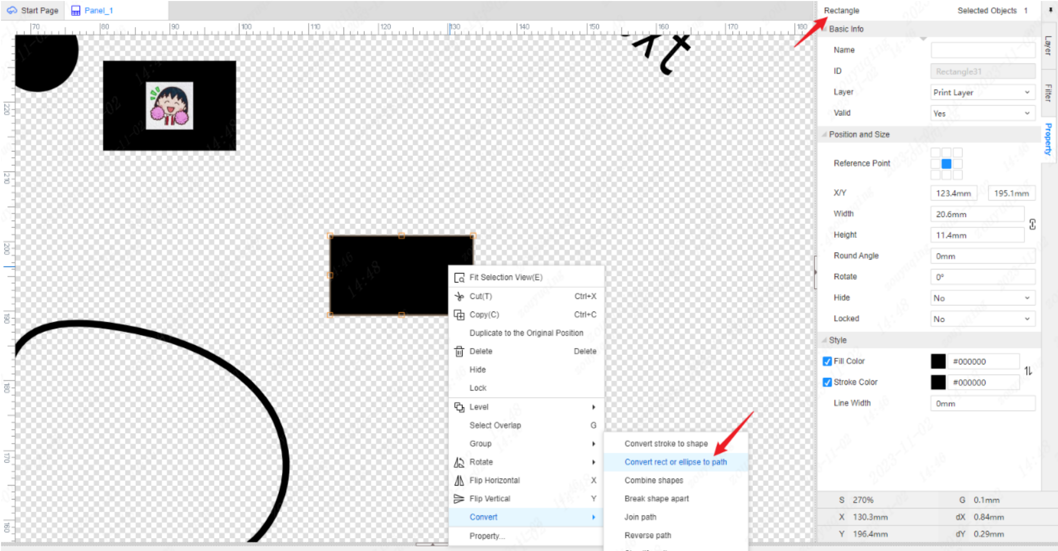
- **Rotate**: Select the "Rotate..." menu to rotate at any angle, and customize the rotation angle in the pop-up window. The left rotation and right rotation are rotated 90 ° each time, the left rotation is counterclockwise, and the right rotation is clockwise. Rotate in the direction.
- **Flip Horizontal, Flip Vertical**: Flip left and right is mirroring the vertical axis of the selected object, while flip up and down is mirroring the horizontal axis of the selected object.
- **Convert**:
    - **Convert Stroke to Shape**: Strokes can be converted to fill shapes, which can be set to the transparent control layer.
      For example, if you want to set the transparency of a shape with a line width of 3mm on a printed layer, copy the shape and switch the layer to the transparent control layer. After discovering that the stroke of the displayed shape is unfilled, right-click to convert the stroke to a filled shape, and the original stroke area will be converted to a filled area to achieve the expected effect.

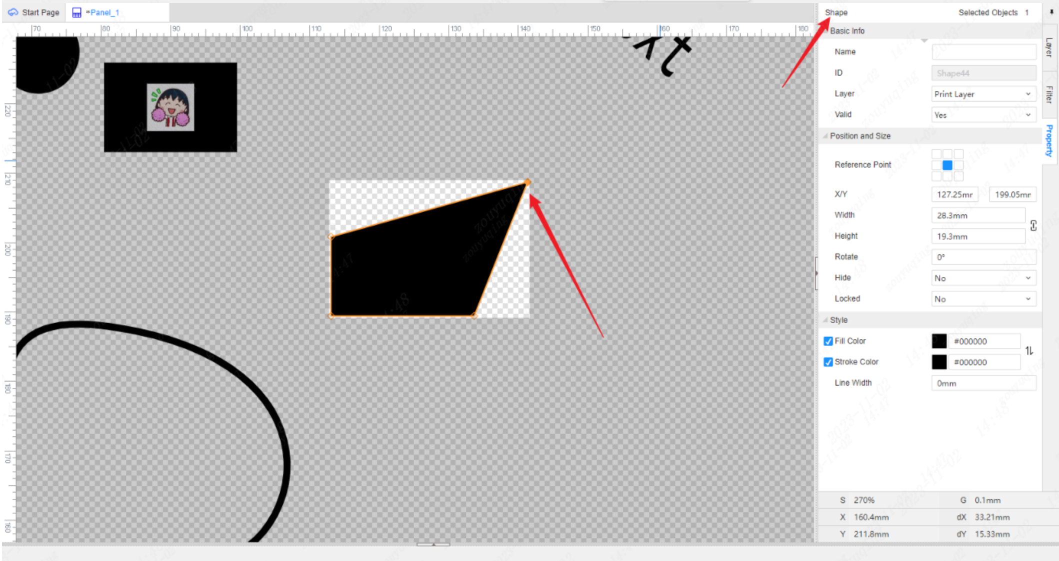




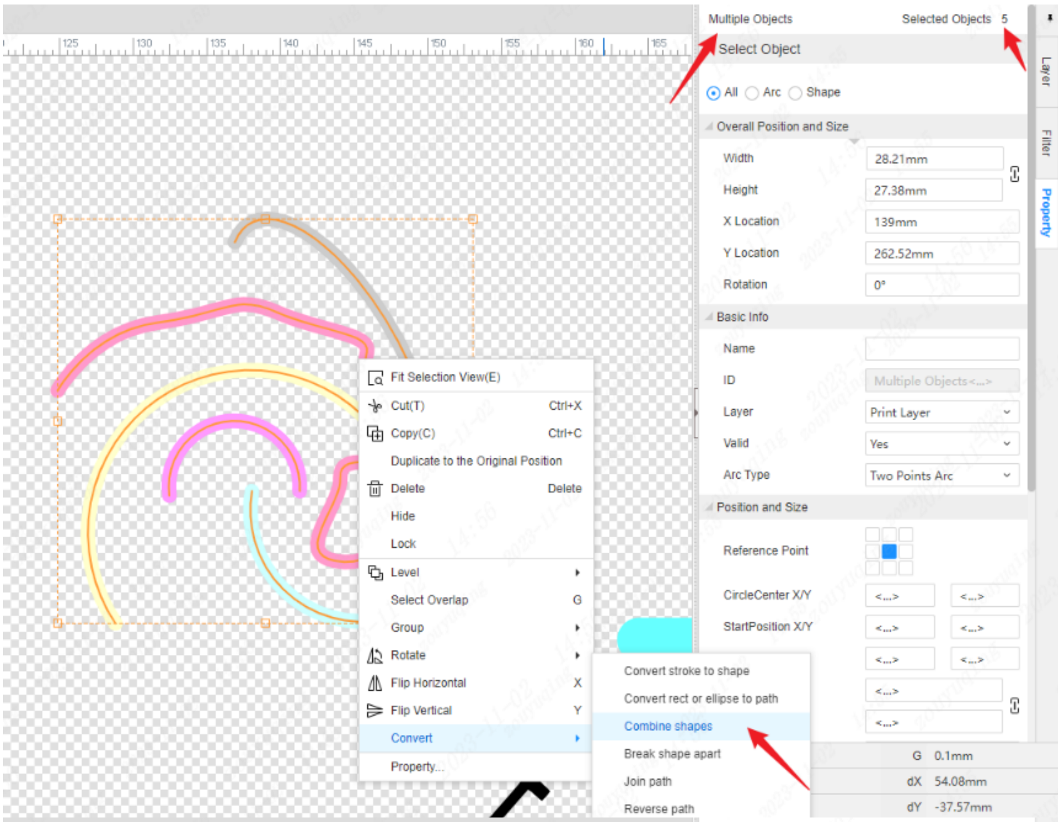


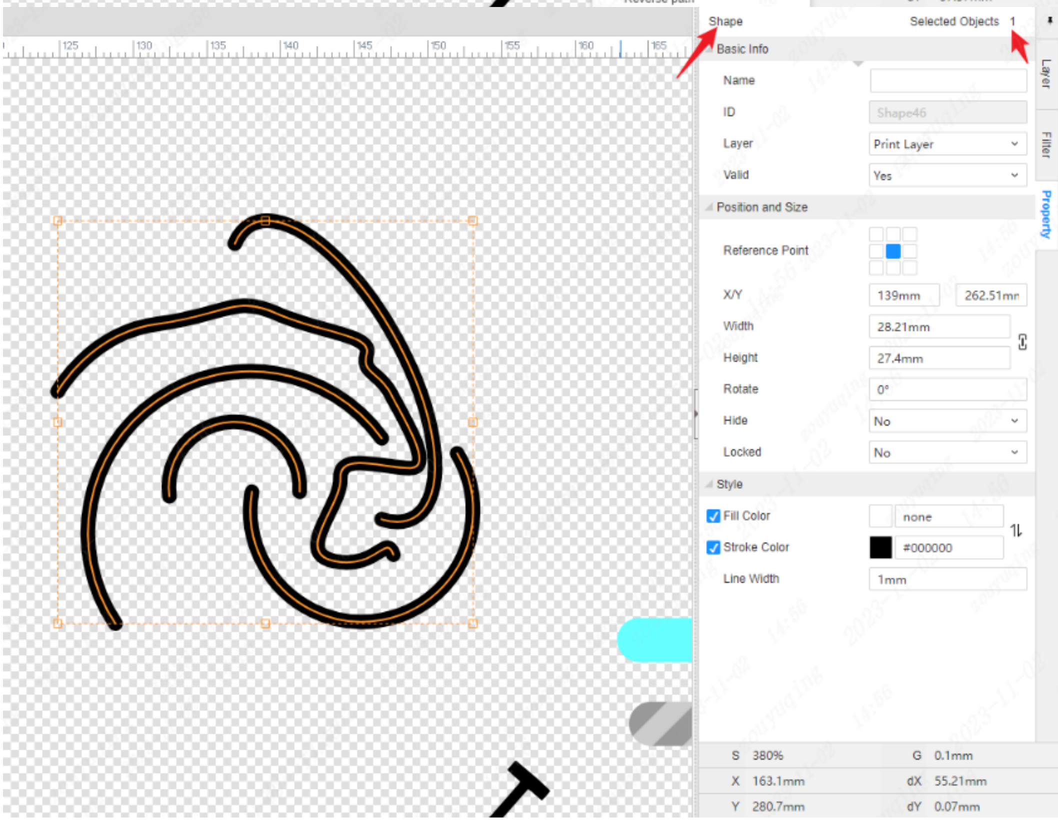
- **Convert Rect or Ellipse to Path**： You can convert elements whose properties are rectangles or ellipses into shapes. After having shape properties, you can enter the shape mask for further editing.



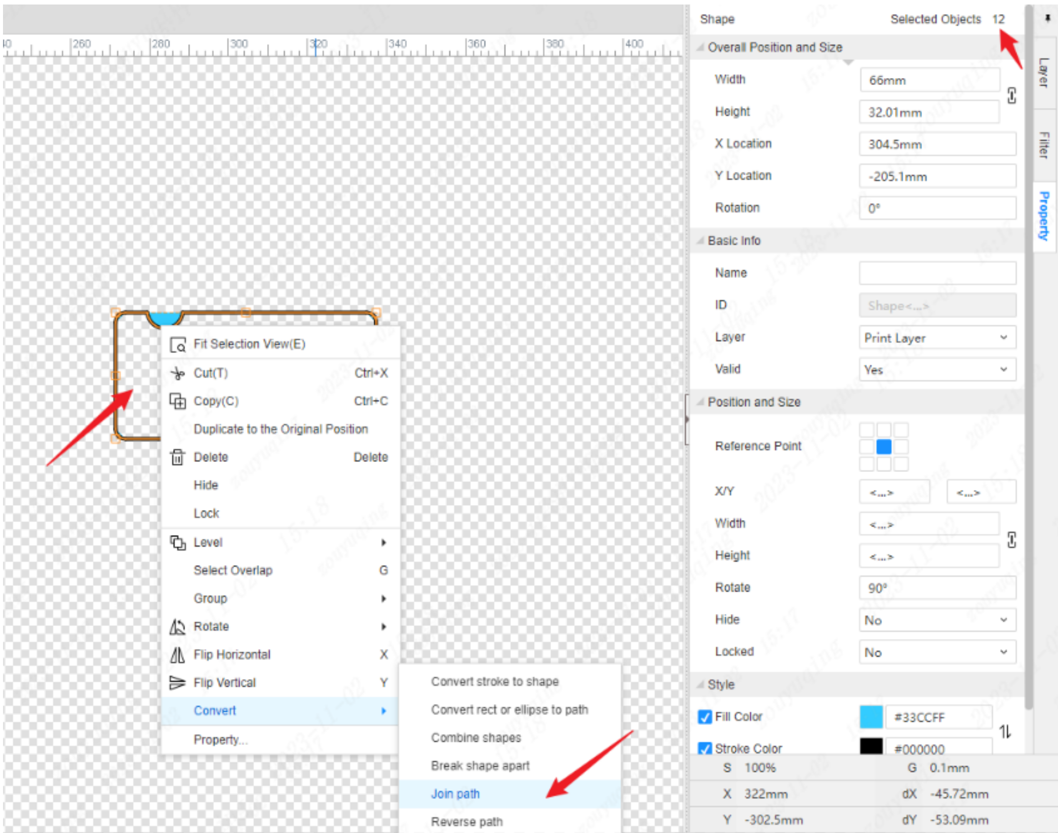


- **Combine shapes**: Multiple independent elements can be merged into one shape element, retaining the original contour path, which is different from the merged area of Boolean operations.

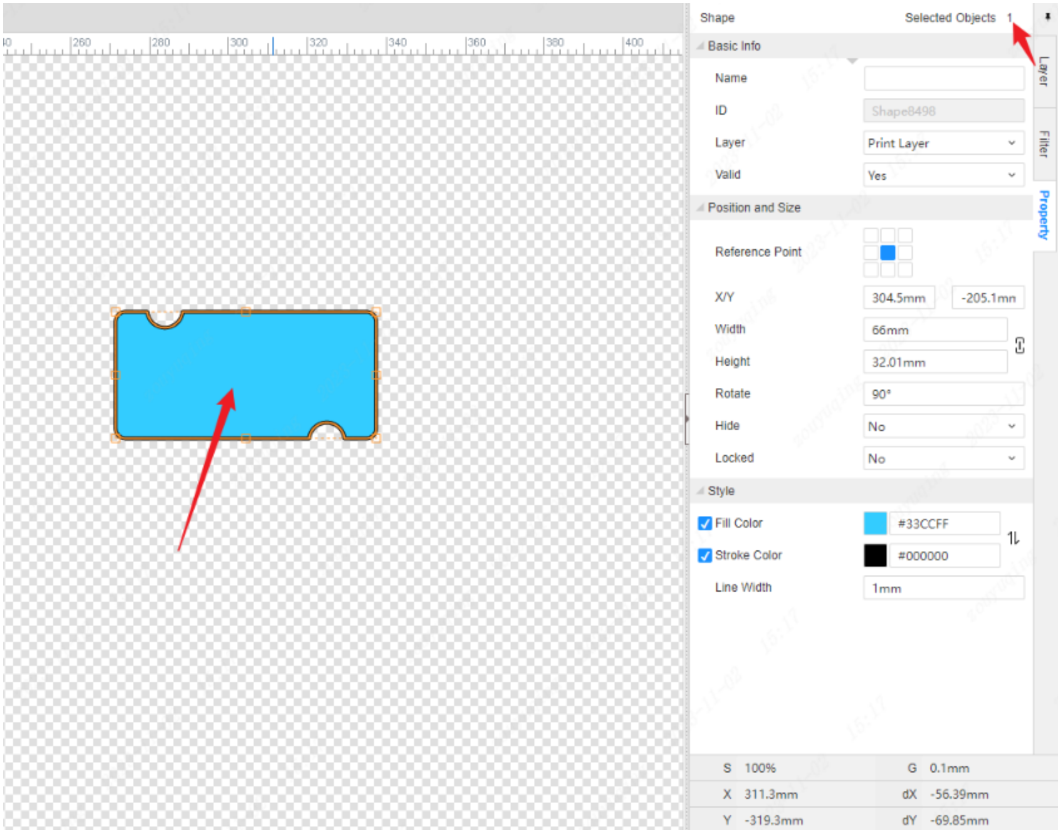




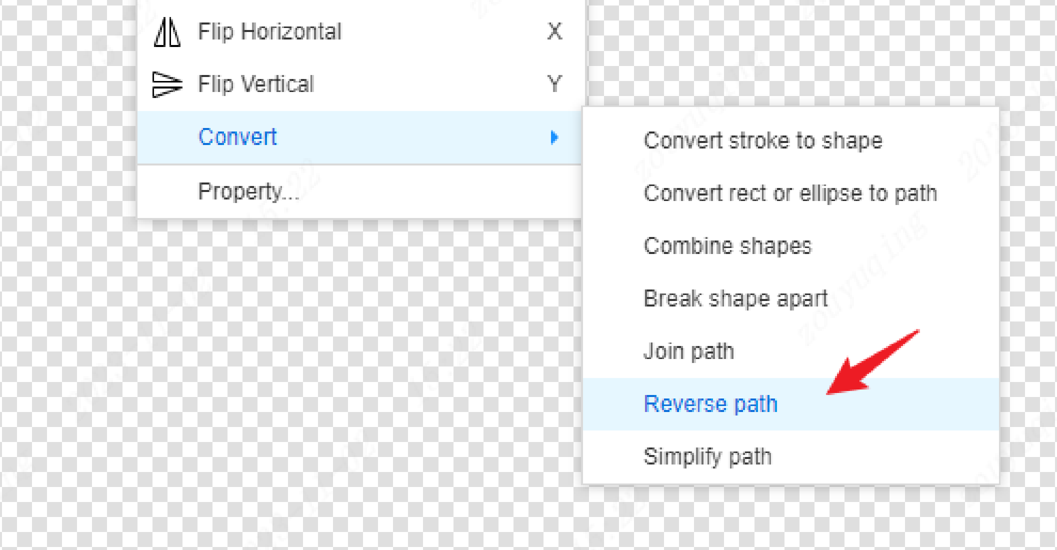
- **Join Path**： can manually close the path. Usually imported DXF There will be many disconnected paths, which cannot be produced abnormally after export. After selecting the path, we can right-click and operate continuous segmented path connection to automatically connect discontinuous paths into closed paths.
  If filling is performed when the path is not closed, unexpected filling will be generated as shown in the figure below.



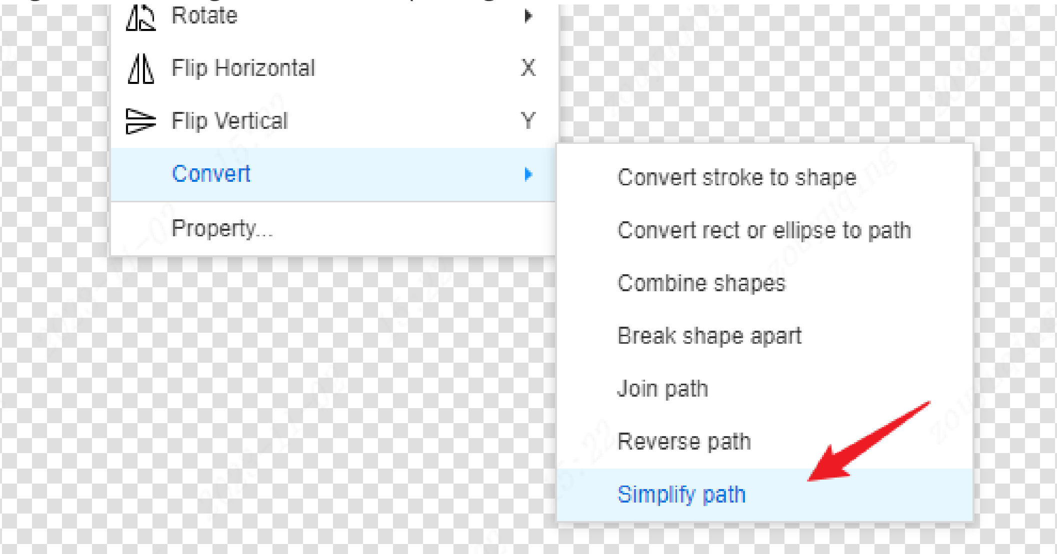
After the path is closed, the filling effect is in line with expectations.



- **Reverse Path**: a shape of the path has a starting point and an end point, reversal is to reverse the starting point and the end point, the naked eye can not see the difference, but the corresponding data changes.



- **Simplify Path**: The original path may have 100 points, and some points will be deleted if they have no effect on the shape. After simplification, only 50 points may be left. (Significant editing effect when importing DXF)

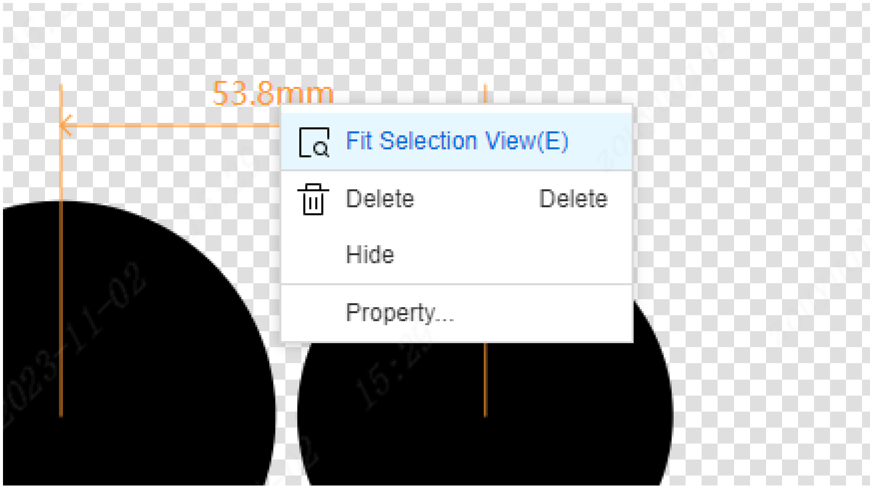


- **Property...**: You can open the property panel on the right to view the property information of the selected object.

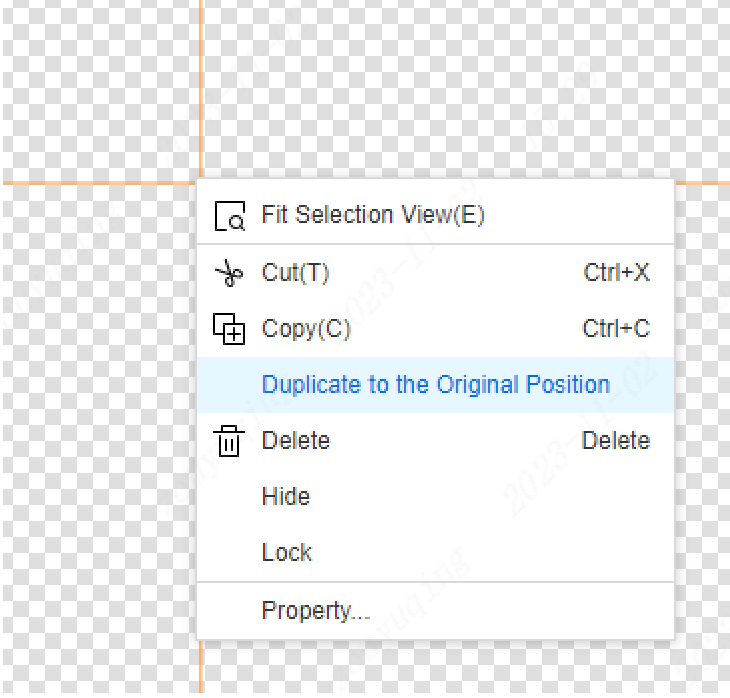
Auxiliary primitive

The intelligent dimensions, auxiliary lines, auxiliary points, and dimensions in the panel are all auxiliary elements, and the right-click menu varies slightly according to their respective characteristics.

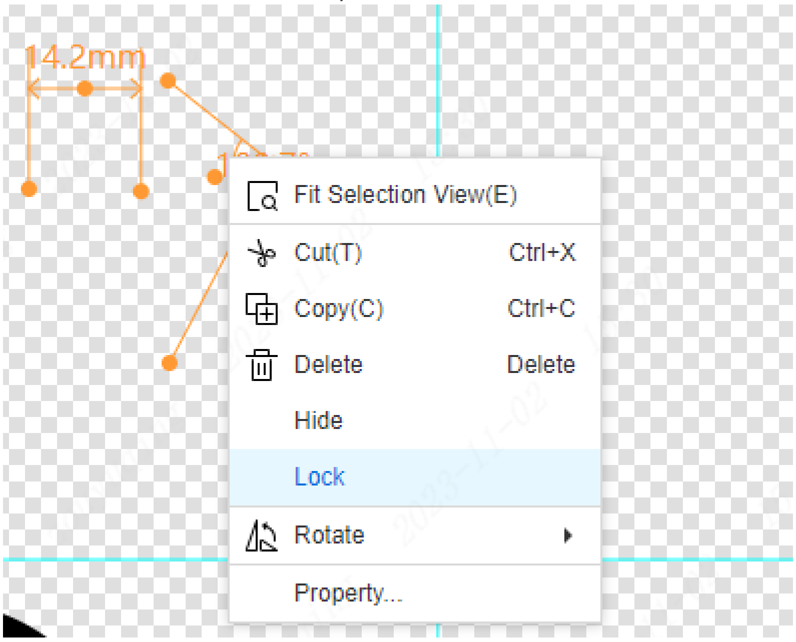
- **Smart Dimension**: Right-click menu can operate fewer items, support hiding smart size.



- **Auxiliary Line, Auxiliary Point**： can be cut and copied or repeated to the original position and other operations, support locking auxiliary lines, auxiliary points, easy to assist alignment.

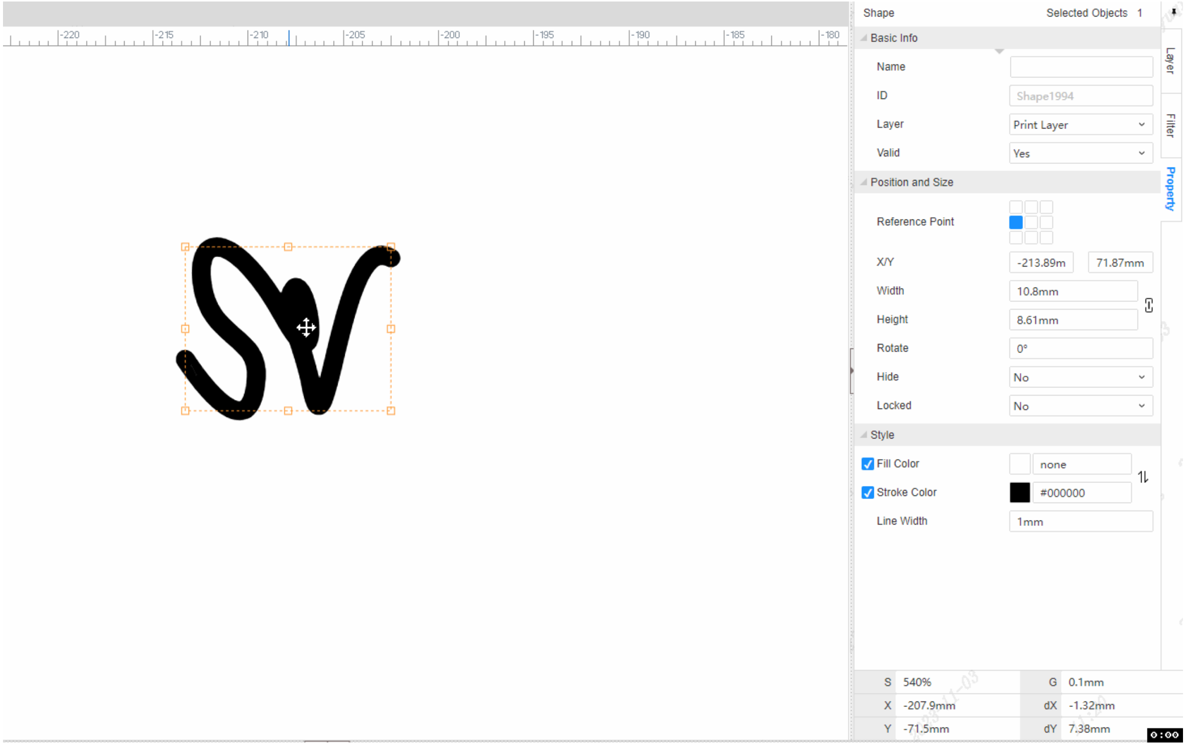


- **Dimension**： You can manipulate the size element to rotate with the measured element.

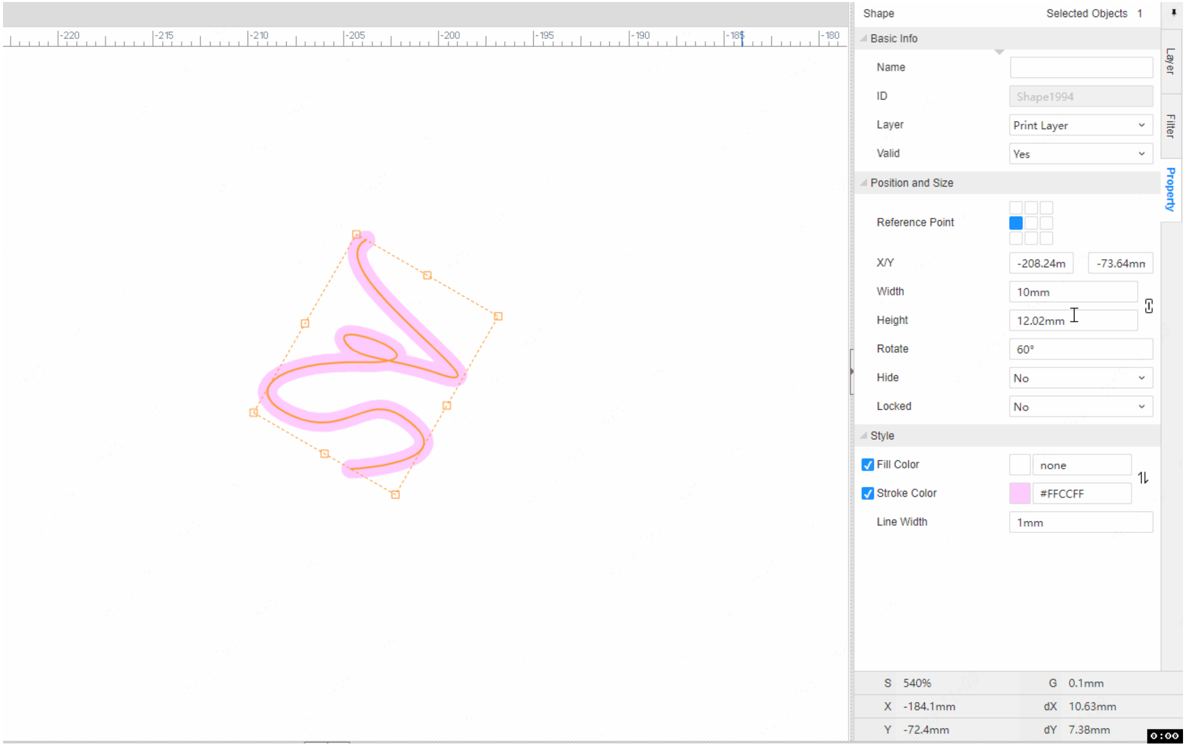


Canvas & Right Properties

- When manipulating elements on the canvas (including adjusting position, size, angle, etc.), the properties on the right side are kept updated in real time.



- Adjust the properties in the property panel on the right, and the corresponding canvas will also be updated in real time.



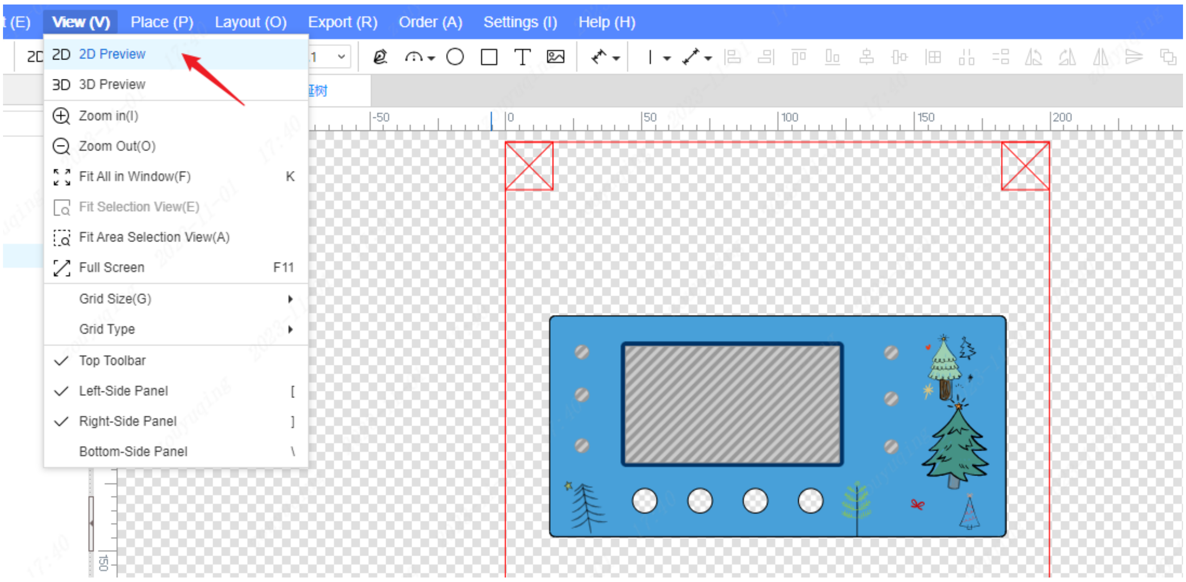
2D Preview

2D Preview

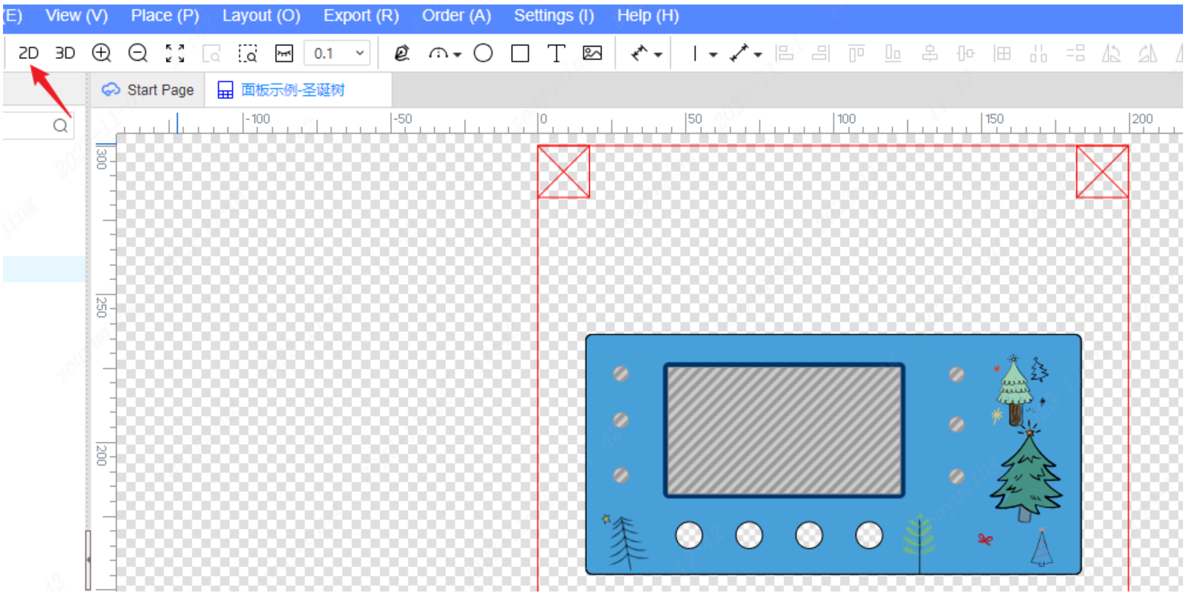
The panel design supports 2D preview viewing effect. The current 2D preview is not very perfect, and EasyEDA Pro will complete 2D preview and 3D preview later.

Function entry

- Top Menu - View - 2D Preview

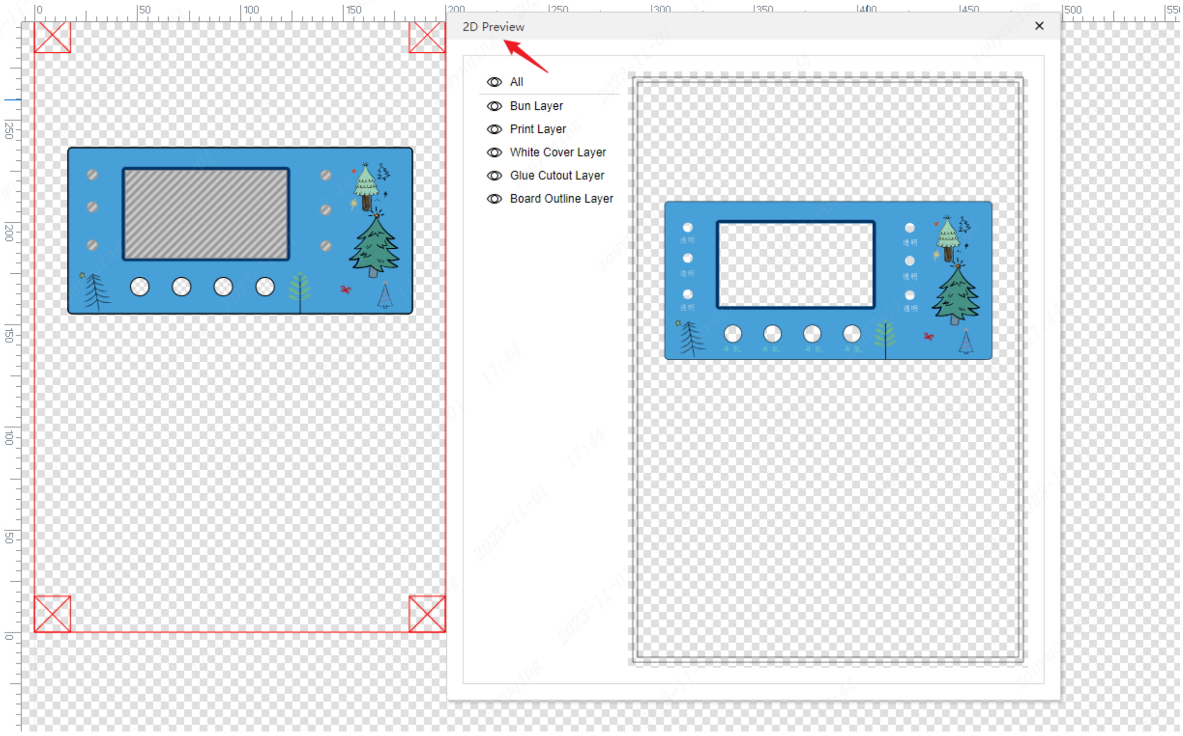


- Directly click the 2D Preview icon in the top toolbar to quickly open the 2D preview.



2D preview pop-up

Because the preview needs to be close to the actual production effect, it will involve the calculation of graphics. The more complex the graphics, the more time it takes for the preview to calculate, render and load.



Click the button above to control the display and hide of the layer.

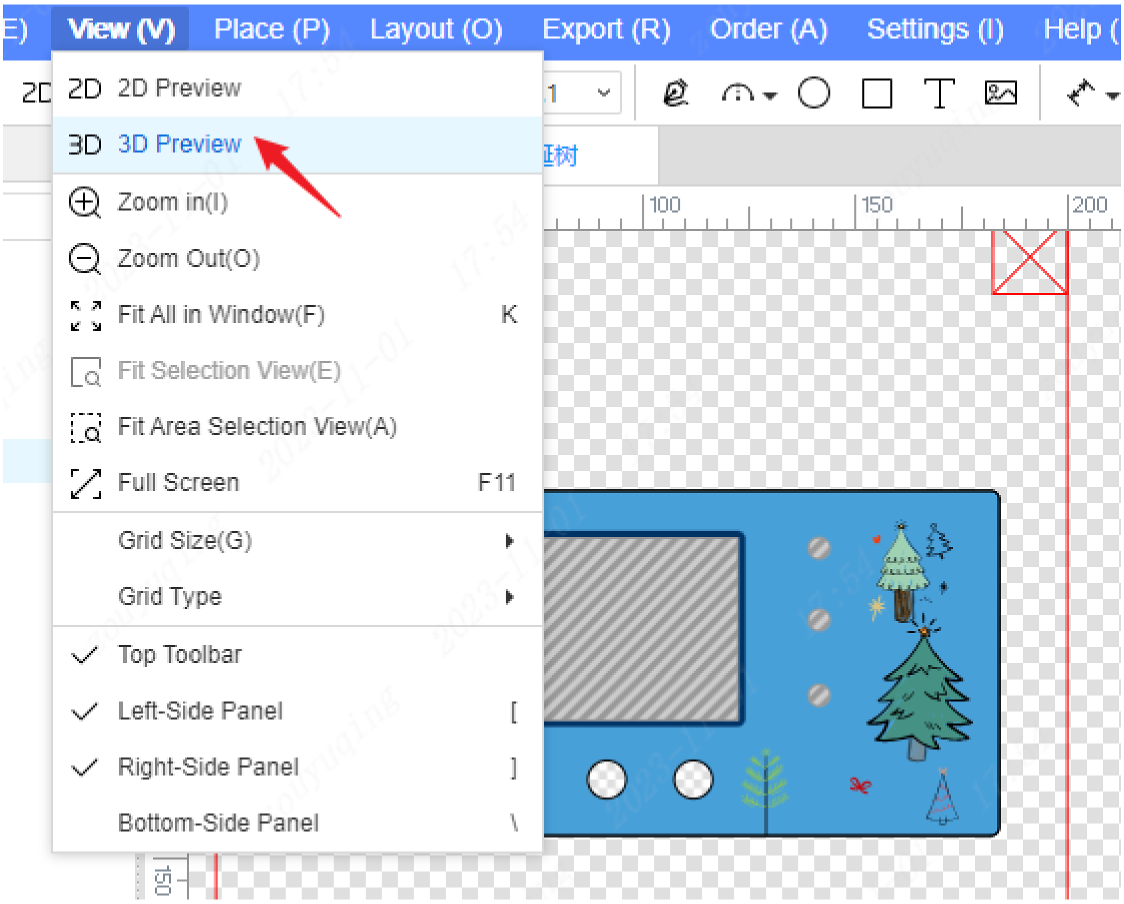
3D Preview

3D Preview

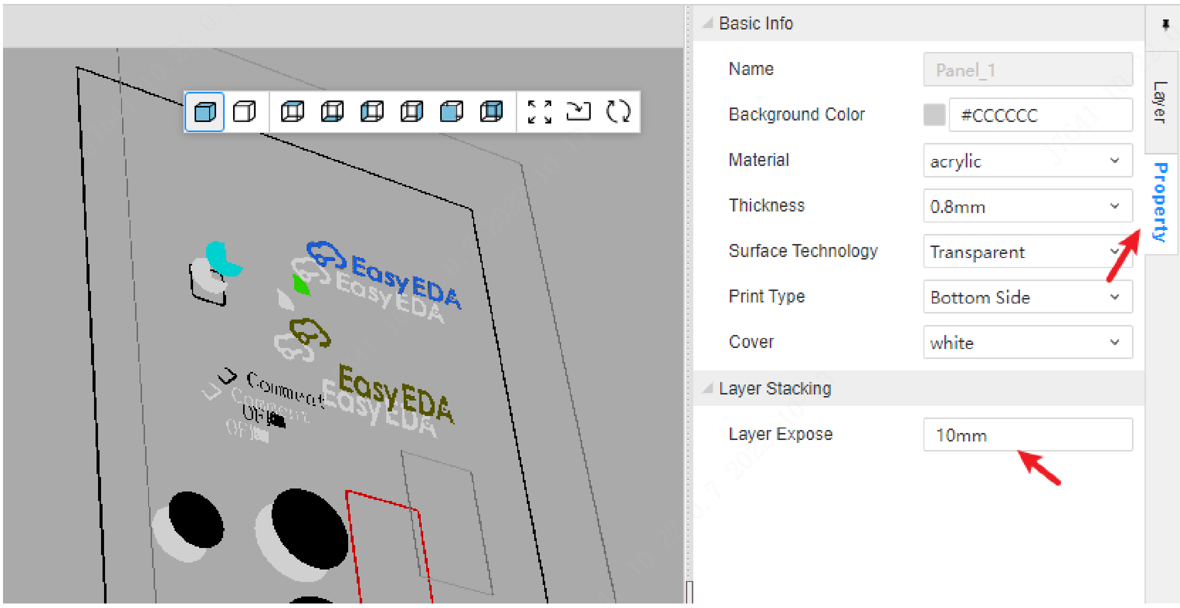
The panel design supports 3D preview viewing effect, which can avoid some design mistakes, and can also see the final design sample of the panel.

功能入口

- Top Menu - View - 3D Preview

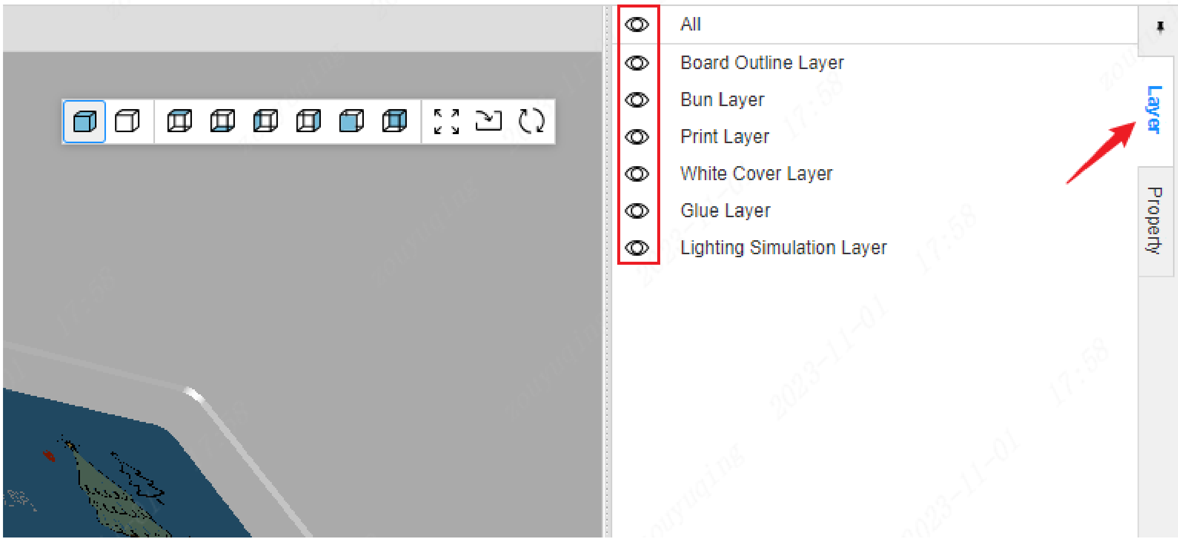


Because the preview needs to be close to the actual production effect, it will involve the calculation of graphics. The more complex the graphics, the more time it takes for the preview to calculate, render and load.



Click the buttons above to control the orientation of the preview, import changes, refresh the view, and more.

In the property panel on the right, you can set the property parameters of the 3D preview, and hide and display layers.



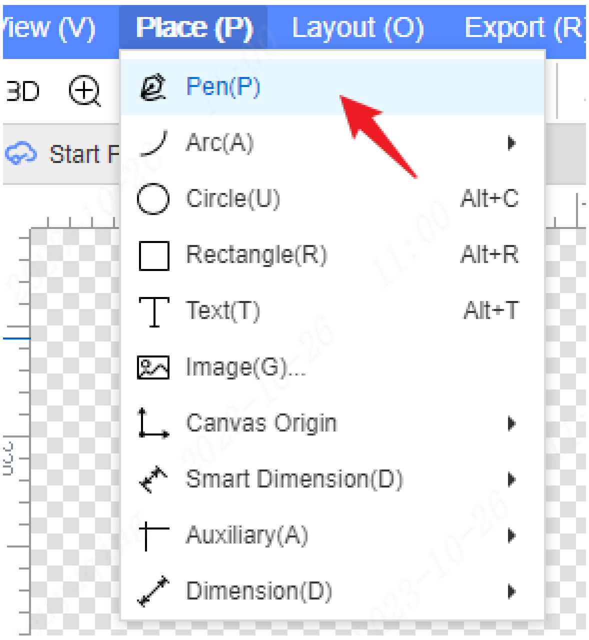
Pen

Pen

The polyline of the panel is slightly different from the polyline of the schematic diagram. In the panel, it is a path shape as a whole, which can be seen by checking its properties panel. Lines drawn with the pen tool are called shapes in the panel and can be any line or curve. The default line is drawn, and holding down the left button can switch to curve drawing.

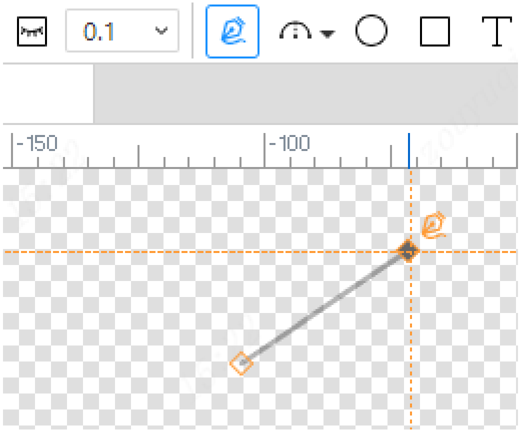
Operation steps

- Top Menu - Placement - Pen Tool
- Menu shortcut: P + P

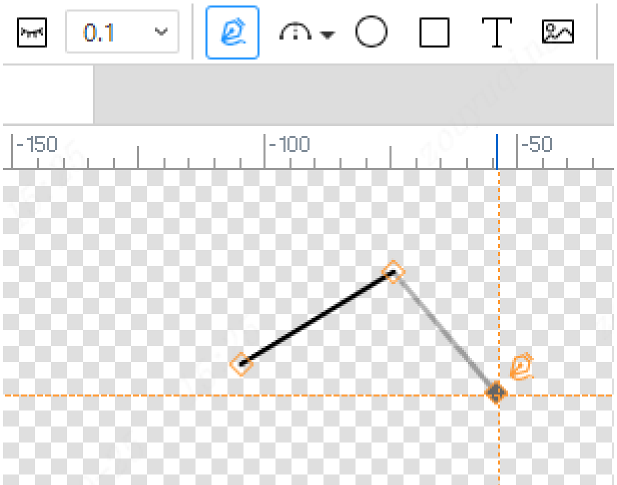


Draw a straight line

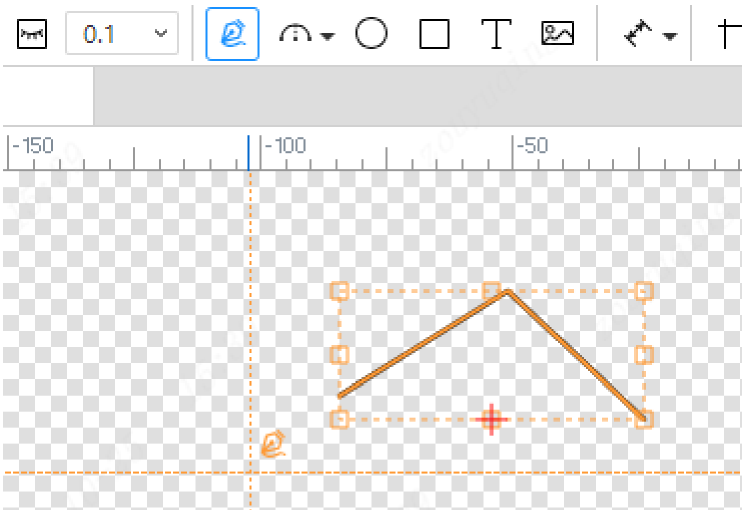
After entering the pen tool drawing mode, the first left mouse button on the canvas determines the starting point, which is the first anchor point, and enters the drawing state of the second anchor point, displaying a segment to be determined.



Continue to determine the line segment path with the left mouse button, complete the drawing of a polyline, enter the drawing state of the next anchor point, and display the segment to be determined between the last left mouse button and the current cursor (Note: To distinguish from the determined segment , the display of the segment to be determined increases transparency)



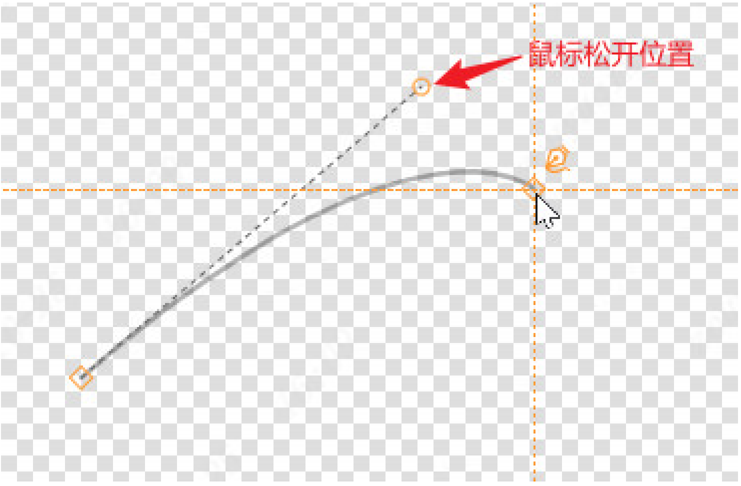
It can draw any number of polylines continuously. The right mouse button or button Esc will exit the drawing of the current polyline and enter the state to be drawn of the next polyline



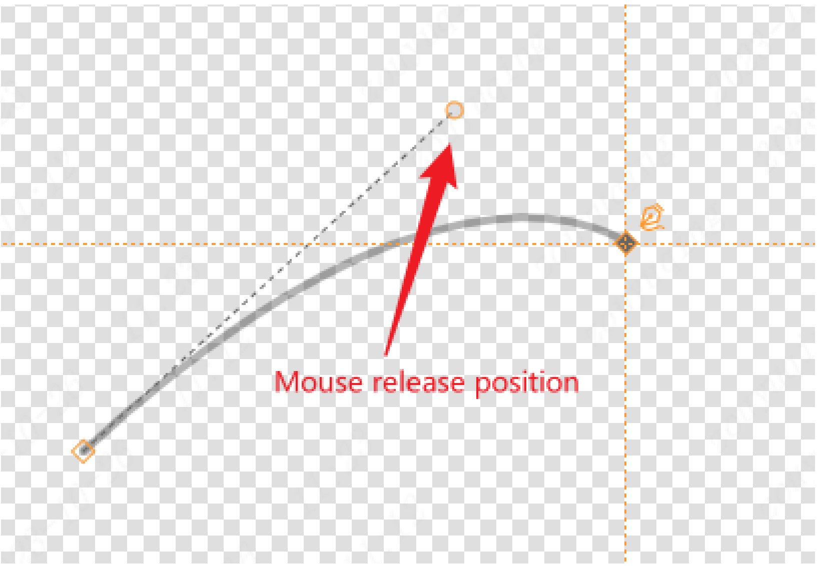
Right-click or press Esc to exit the pen tool drawing mode, that is, the upper right corner of the cursor no longer follows the display pen tool icon

Draw curves

If you hold down and drag the left mouse button on the canvas, an anchor point will be generated at the pressed position, and a handle will be generated to follow the mouse until the left mouse button is released. The handle of the first anchor point will be generated at the position where the mouse is released and enter the drawing state of the second anchor point.

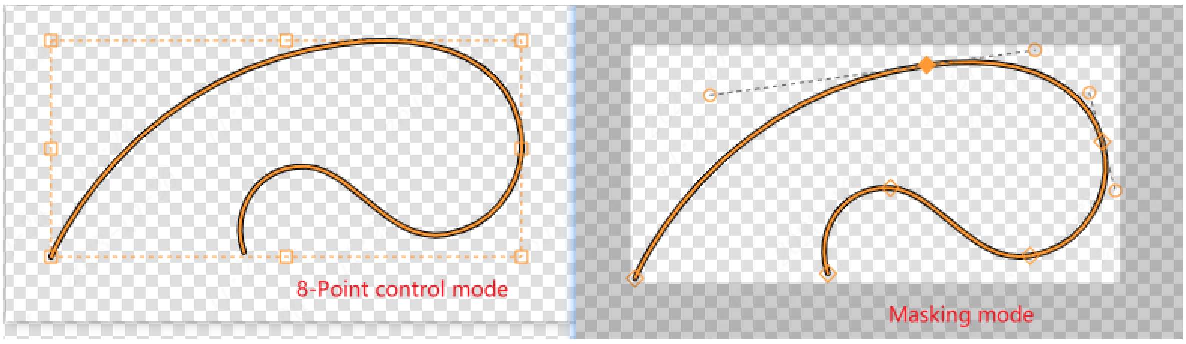


After determining the starting point, the left mouse button is also used to determine the anchor point. Hold down the left mouse button and drag to determine the handle. Subsequent dragging of the mouse will generate two handles based on anchor symmetry. That is, one handle of the anchor point is generated at the position where the mouse is released, and the other is generated at the position based on anchor symmetry.

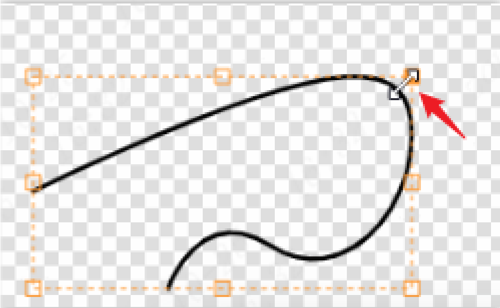


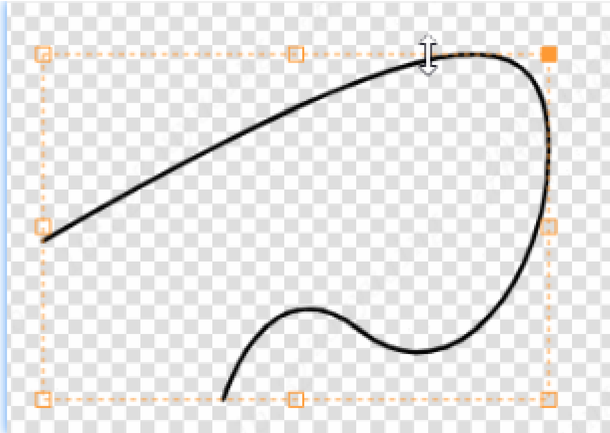
8-Point control mode & mask mode

Directly click or box to draw the completed shape, which is the 8-point control box mode. Double-clicking the shape will enter its mask mode.

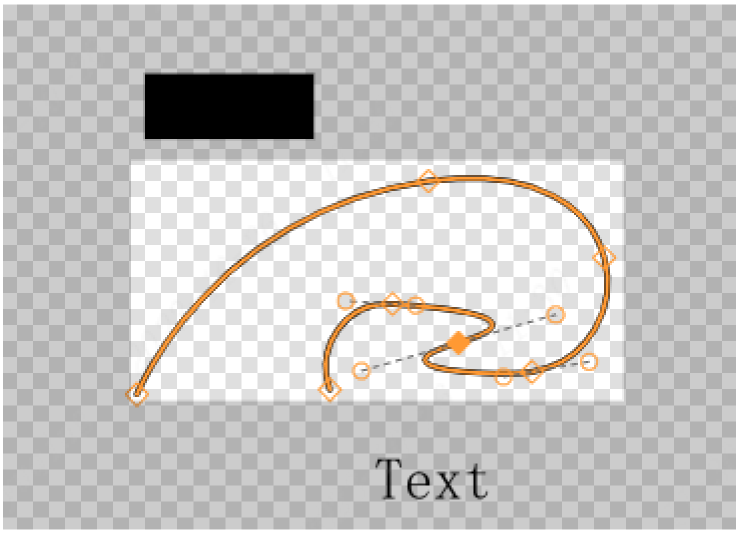


In the 8-point control mode, you can drag the stretch control points or edges on the bounding frame to adjust the size of the shape



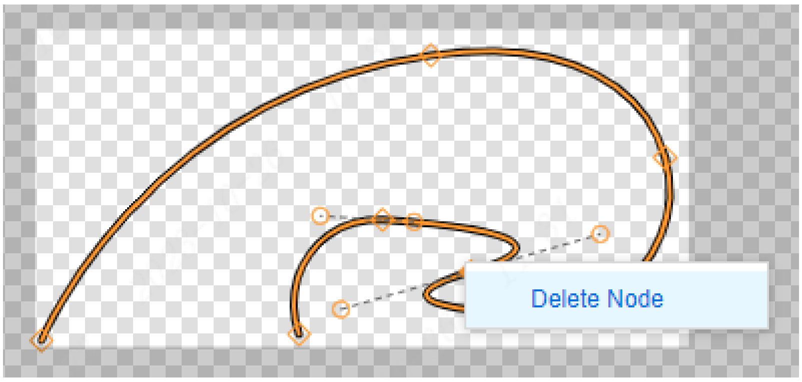


When adjusting the shape of the anchor or handle, you must double-click to enter its mask. Double-click the shape to enter the mask, as shown in the figure below, you can see the gray masking effect in the non-shape area, which will shield the elements outside the shape mask, that is, you can only edit to that shape.

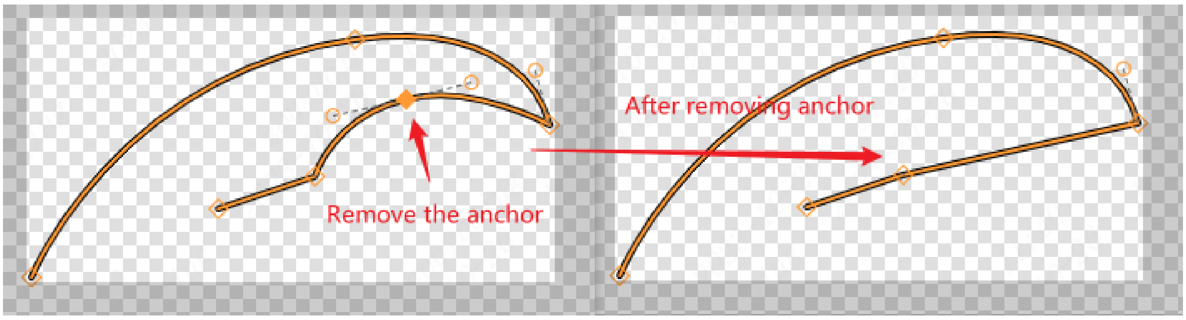


Delete node

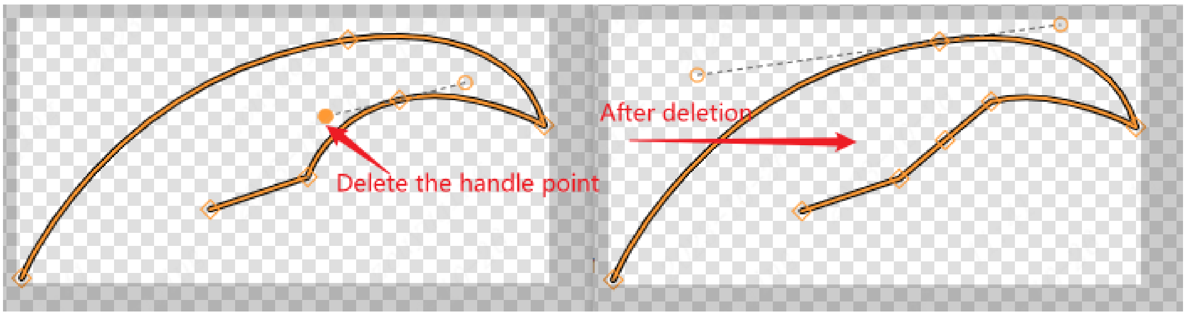
Click to select an anchor or handle point, right-click the delete node menu, or double-click the anchor or handle point to delete



The adjacent anchor points of the deleted anchor point will be connected in a straight line, and the associated handle point will be deleted at the same time as the anchor point.



Operating the delete node at the handle point will only delete the current handle point, and the corresponding curve part will become a straight line

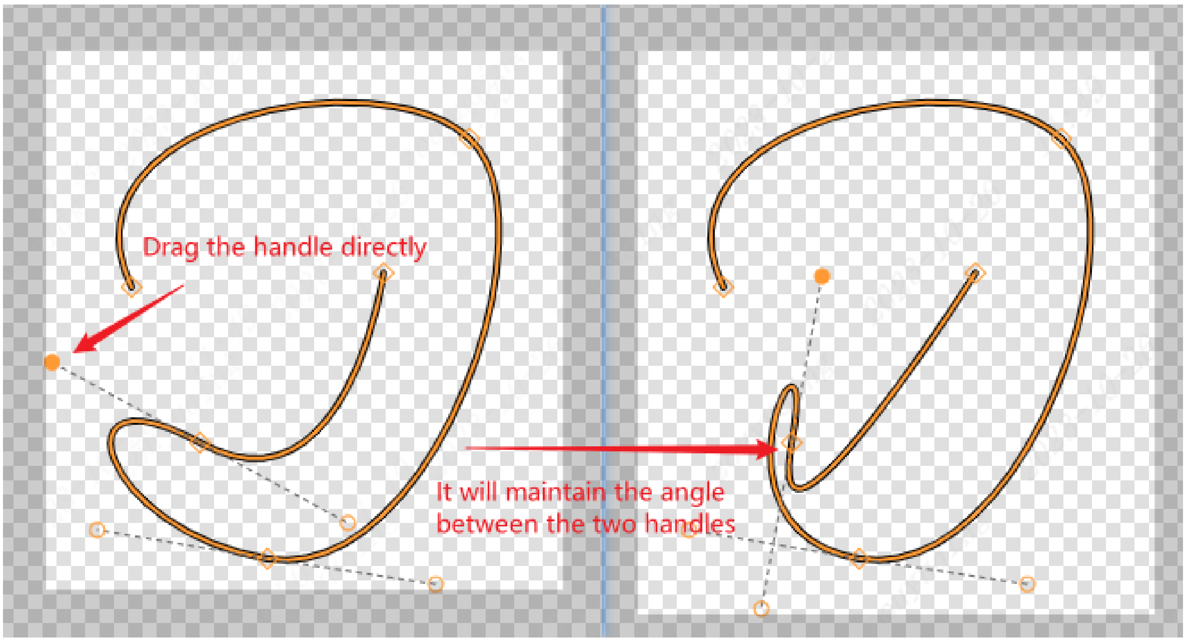


Add a node

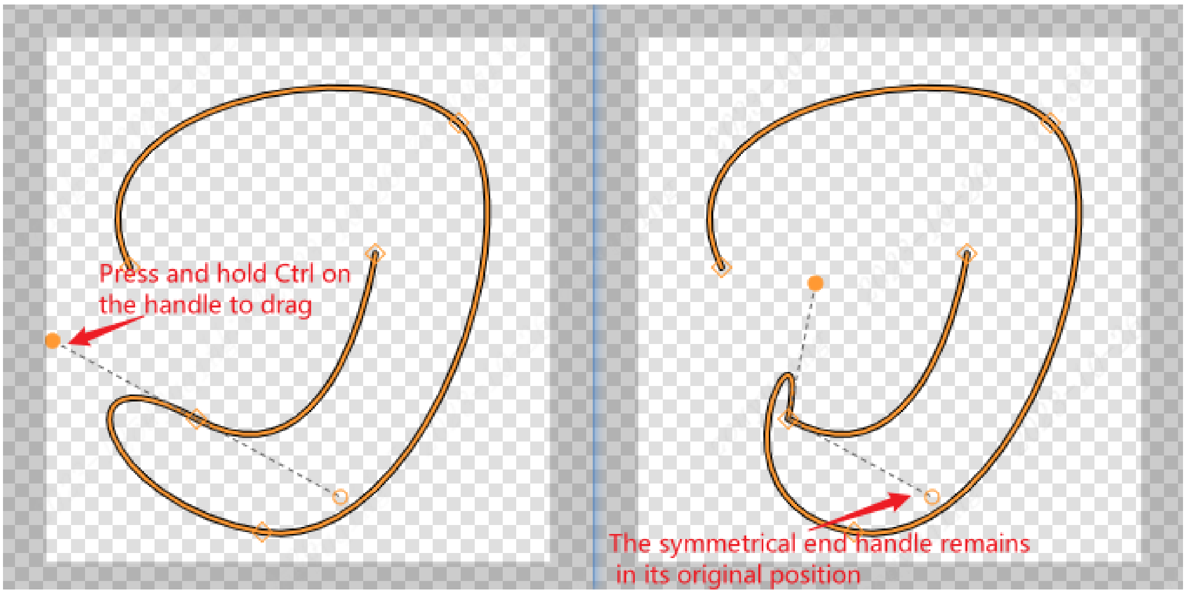
Add an anchor point, select and move the cursor to hover over the shape. An anchor point waiting to take effect will be displayed between every two adjacent anchor points. Click to generate an anchor point at the corresponding position.
Add a handle, hold Ctrl on the anchor point where you want to add a handle and drag it directly to generate the corresponding handle.

Adjusting handle

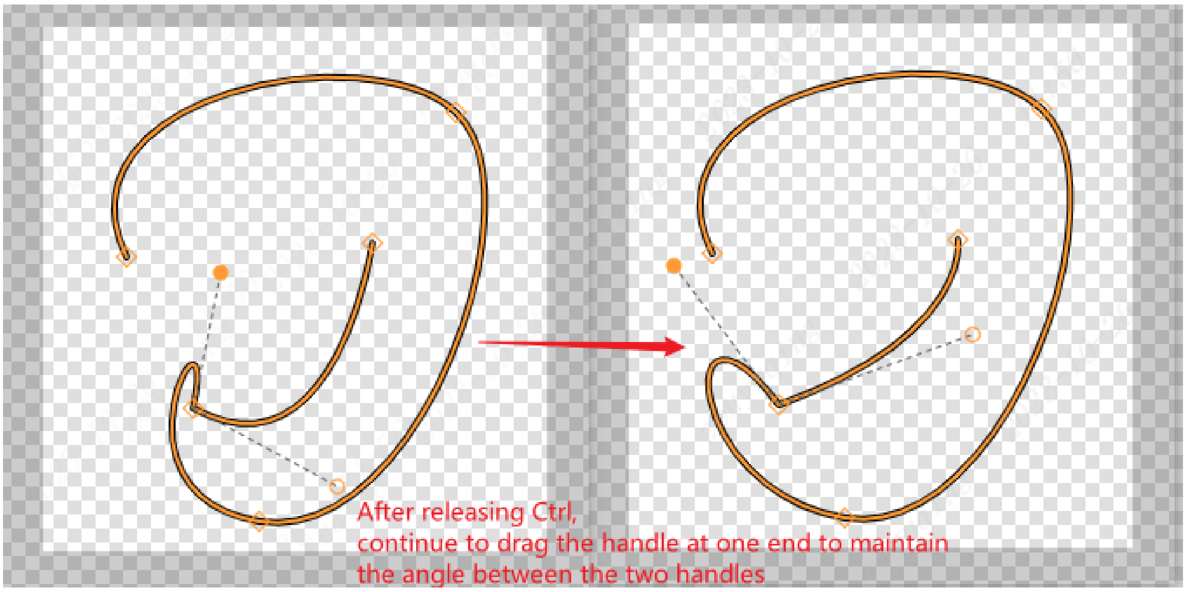
Directly dragging one end of the handle to move will maintain the angle between the two handles, and the handle on the symmetrical end will move synchronously. Adjusting the direction of the handle can control the direction of curve bending.



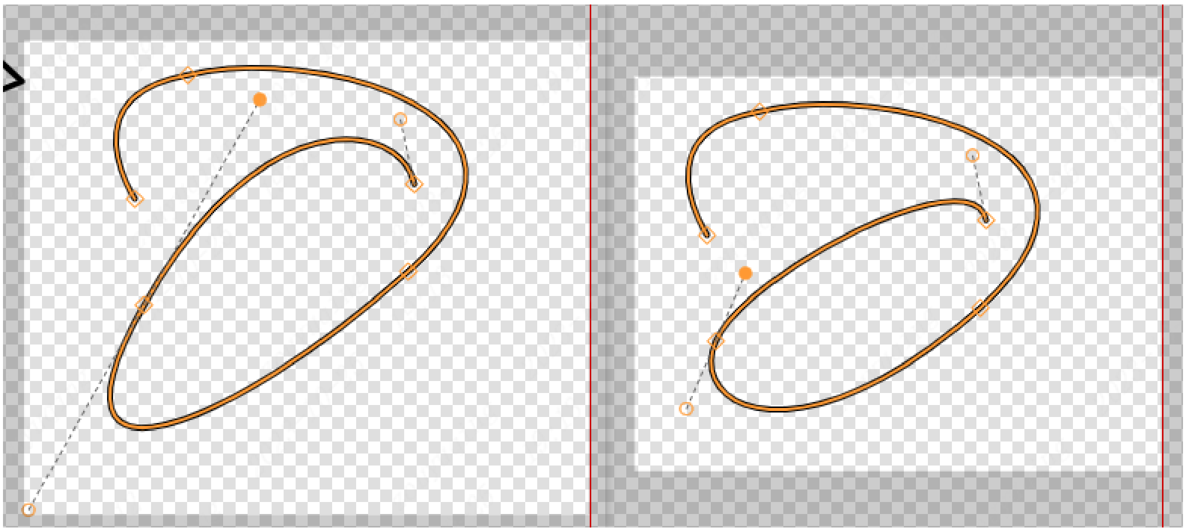
Press and hold Ctrl, drag the handle on one end, and the handle on the symmetrical end will remain in its original position.



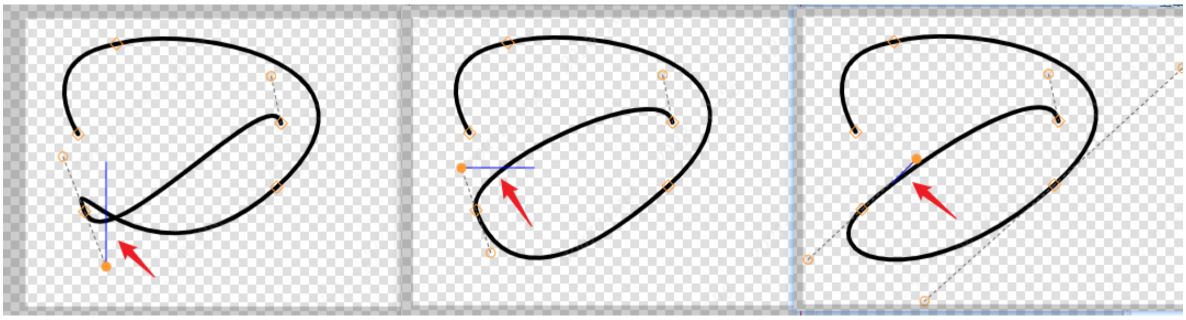
After releasing Ctrl and continuing to drag one end of the handle, the angle between the two handles will still be maintained, and the other end of the handle will move synchronously.



Similarly, by directly moving the handle point to change the length of the handle, the lengths of the two handles will change symmetrically. Holding down Ctrl to move the handle point can only change the length of one end of the handle, while keeping the length of the other end. Adjusting the length of the handle can affect the curvature of the curve.

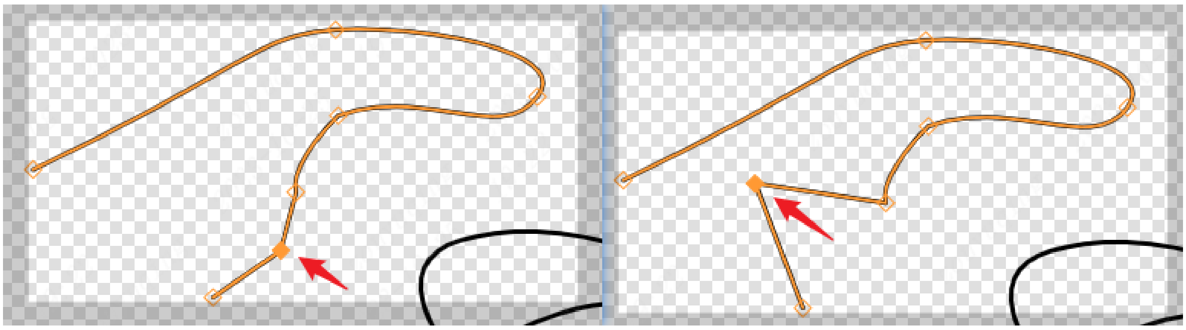


When holding down Shift, whether it is the anchor point or the handle point, it can only be at a multiple angle of 45 °



Regulating anchor

Hold down the left mouse button on the anchor point and move it, only changing the position of the anchor point, affecting only the two lines connected to the anchor point, and the remaining anchor point line segments will remain



Place arc

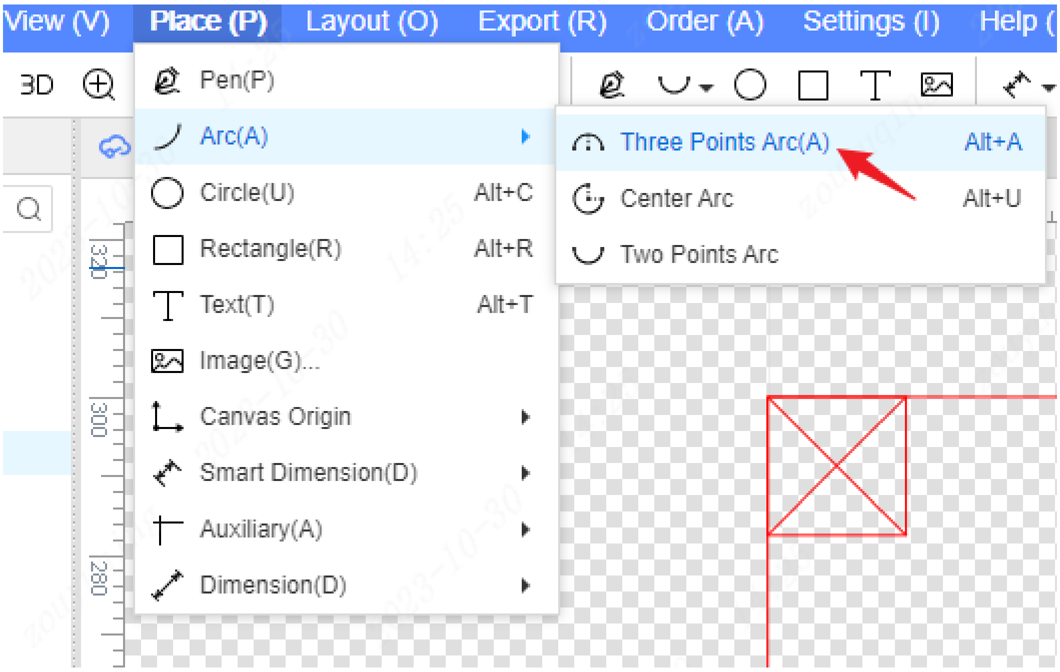
arc

The panel provides three tools for drawing arcs, just like PCB. Three points arc and two points arc are only drawn differently, and the attribute classification names are collectively referred to as two points arc. The difference between the center arc and the two points arc lies in the different adjustment rules on the canvas, and the attribute items are the same.

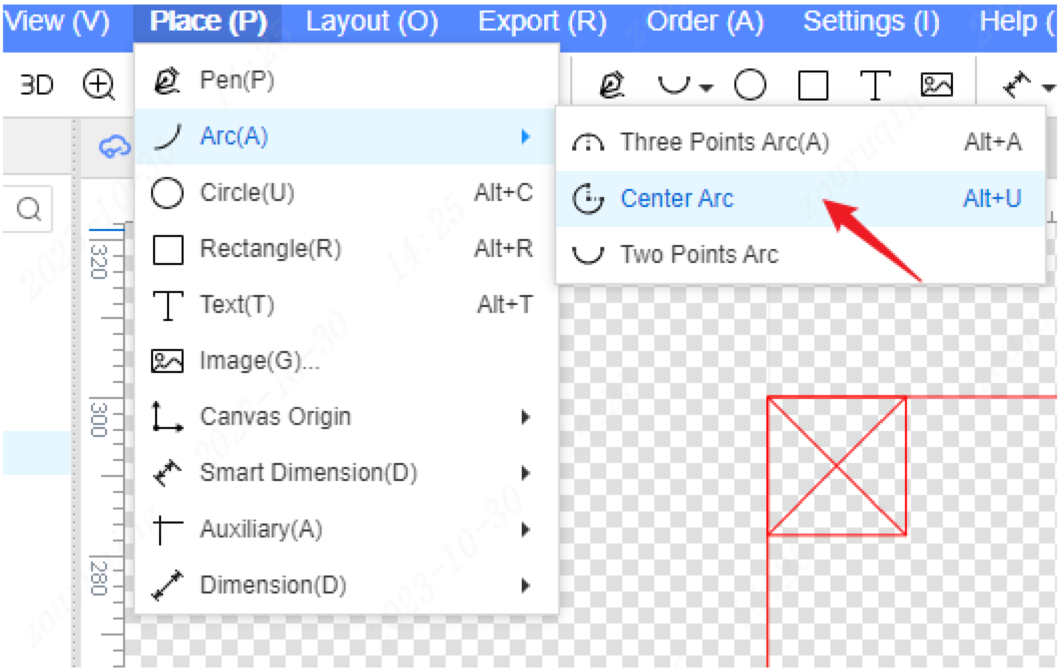
Function entry

Top menu

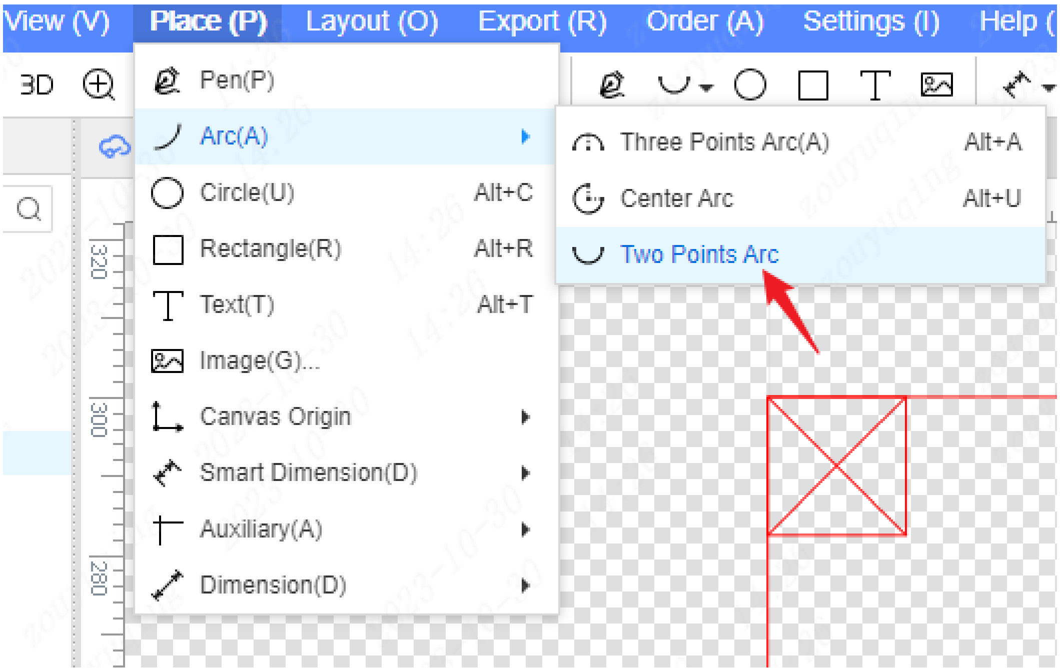
- Top Menu - Place - Arc - Three Points Arc
- menu shortcut： P + A + A
- global shortcut： Alt + A



- Top Menu - Place - Arc - Center Arc
- global shortcut： Alt + U

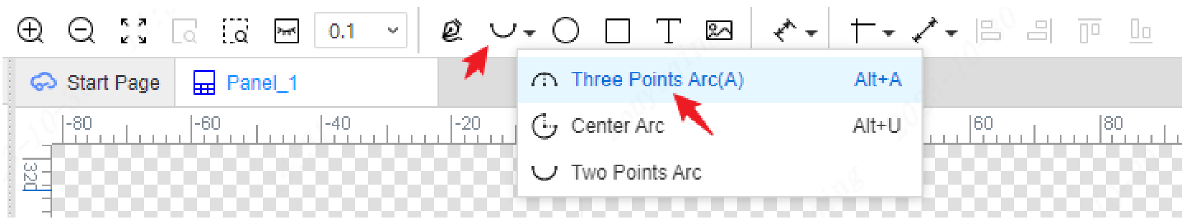


- Top Menu - Place - Arc - Two Points Arc



Top toolbar

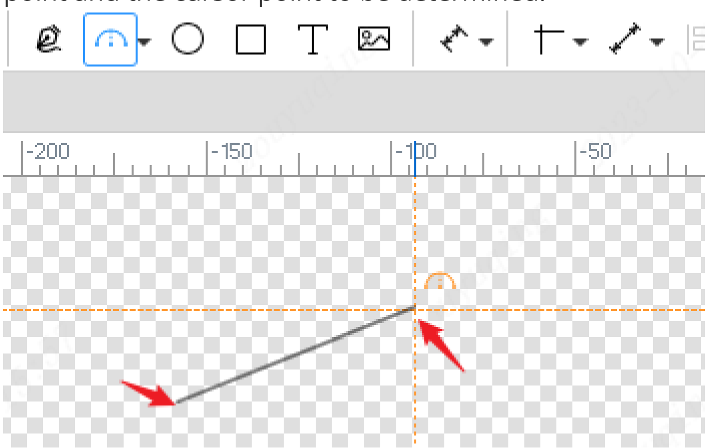
- Directly click the arc icon in the top toolbar, the default is a three-point arc, you can also expand the drop-down options to switch to choose other ways of drawing arcs.



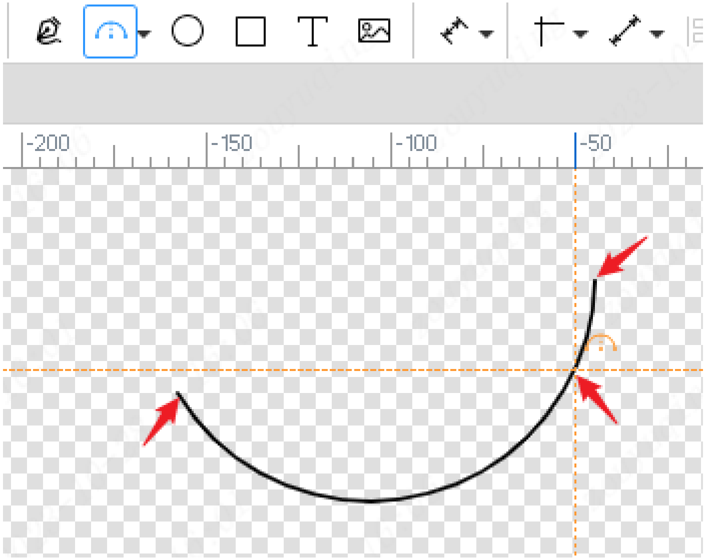
Operation steps

Three points arc

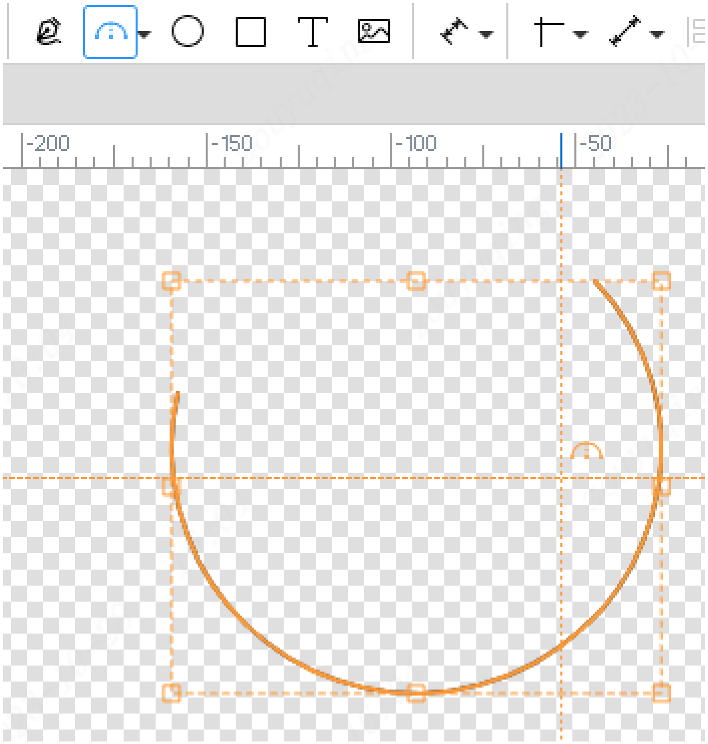
- After selecting the "three points arc" drawing method at the function entrance, the "three points arc" type icon will be displayed in the upper right corner of the cursor. Click the left mouse button anywhere on the canvas for the first time to determine the starting point of the arc, move the cursor, and a transparent indicator line will be visible between the starting point and the cursor point to be determined.



- Click the left mouse button again to determine the end point of the arc at any position on the canvas, move the cursor, and display the arc starting point, end point, and the arc line to be determined by the cursor point.

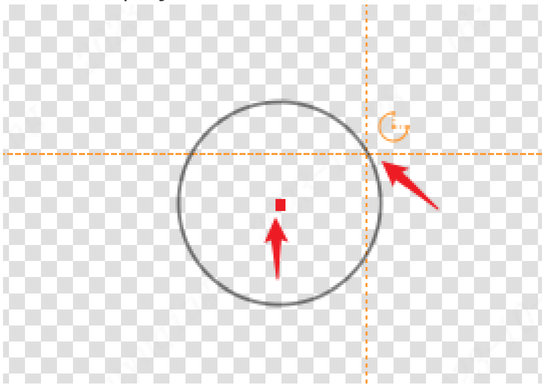


- The third left mouse button can determine the radius of the arc, complete the drawing of this arc, and enter the drawing of the next three points arc.

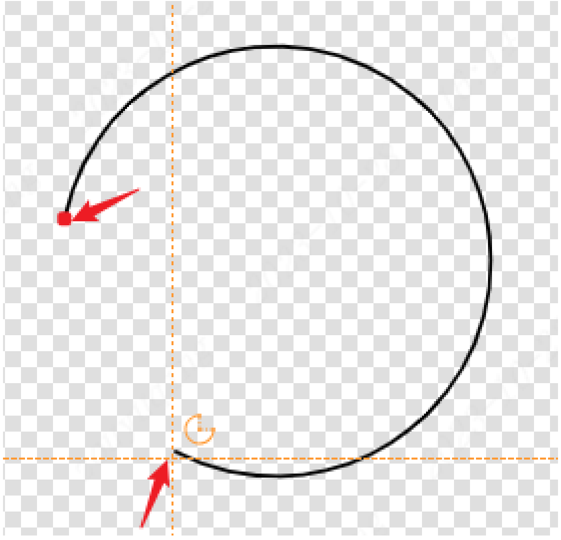


Center arc

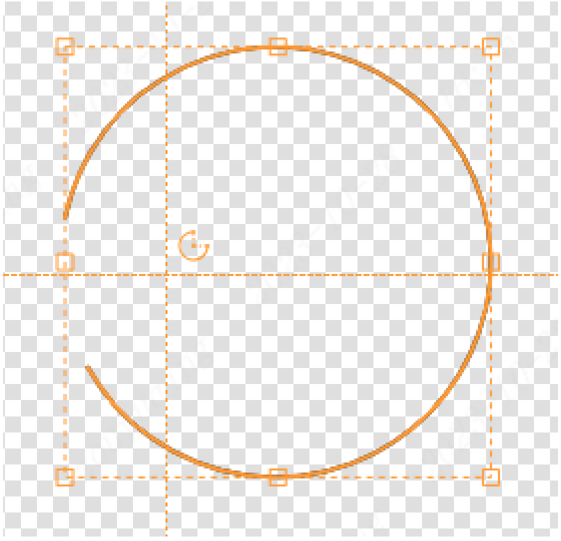
- After selecting the "Center Arc" drawing method at the function entrance, follow the "Center Arc" type icon in the upper right corner of the cursor. The first left mouse button at any position on the canvas determines the center of the arc, move the cursor, and a circular indicator line with a radius from the center of the circle to the cursor point to be determined will be displayed.



- Press the left mouse button again to determine the starting point and radius of the arc, and display the arc line from the starting point to the cursor point to be determined.

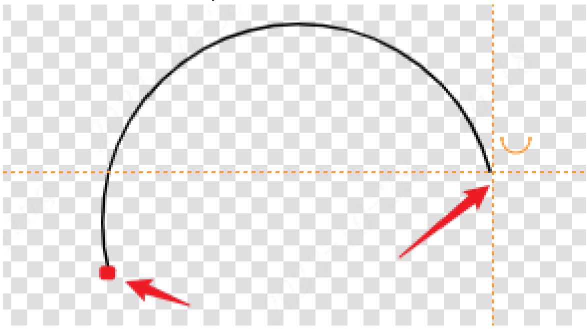


- The third left mouse button can determine the end point of the arc, complete the drawing of this arc, and enter the drawing of the next central arc.

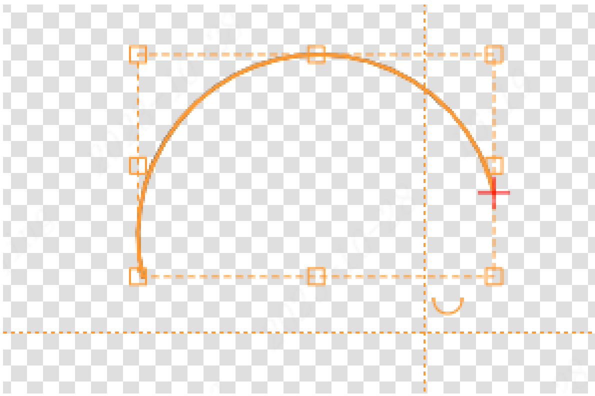


Two points arc

- After selecting the "two points arc" drawing method at the function entrance, the "two points arc" type icon will be displayed in the upper right corner of the cursor. The first left mouse button at any position on the canvas determines the starting point of the arc. Move the cursor to display an arc with the diameter of the distance from the starting point of the arc to the cursor point to be determined.

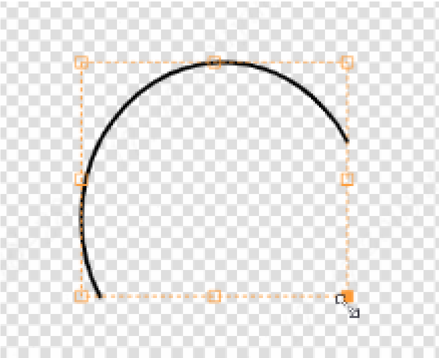


- The second left mouse button can determine the end point and radius of the arc, complete the drawing of this arc, and enter the drawing of the next two-point arc.

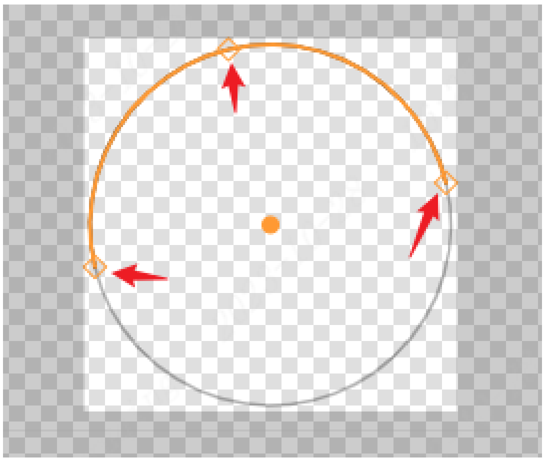


Arc mask

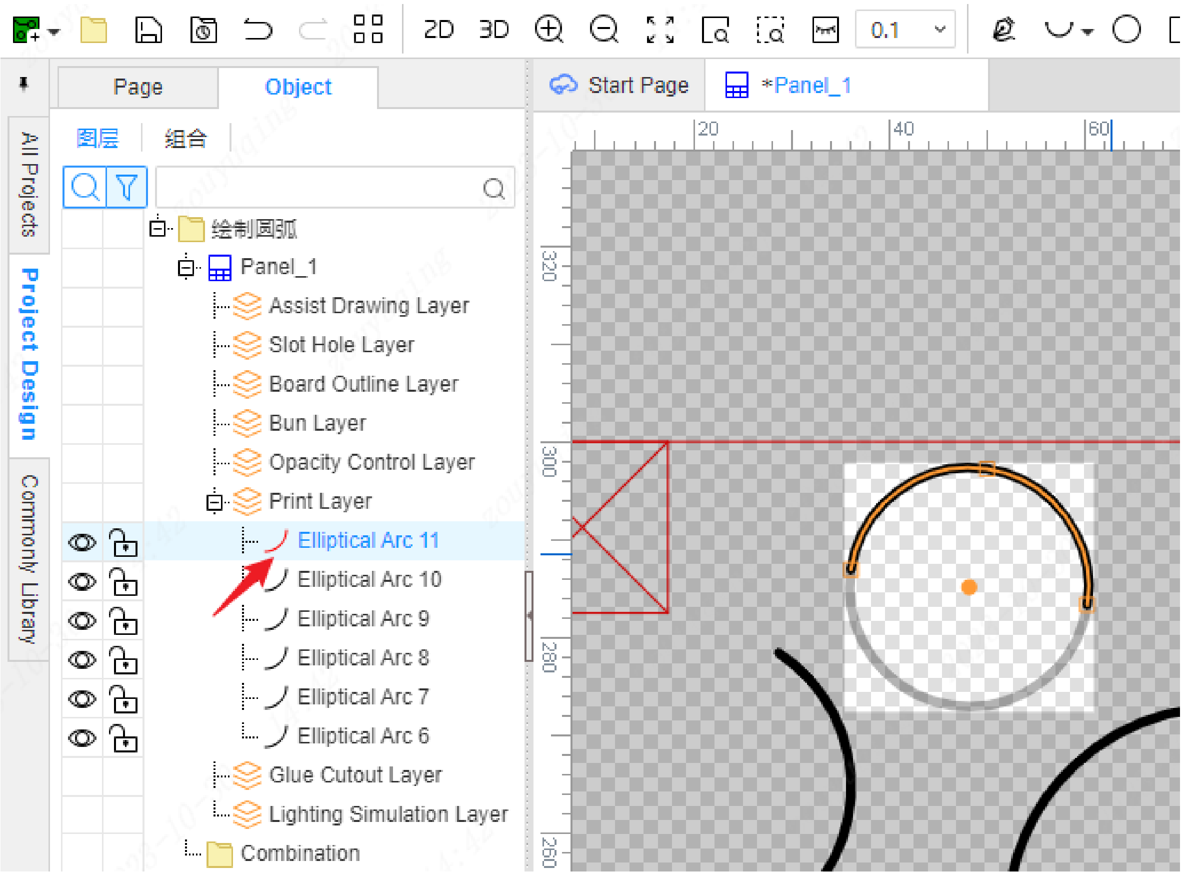
Directly selecting the arc in the panel is an 8-point control mode, which can stretch, rotate, flip and other operations on the arc.



If you want to adjust the starting point, end point, or radius of an arc on the canvas, you need to double-click the arc to enter its mask mode. Double-clicking the mouse and pressing Esc can exit the arc mask.



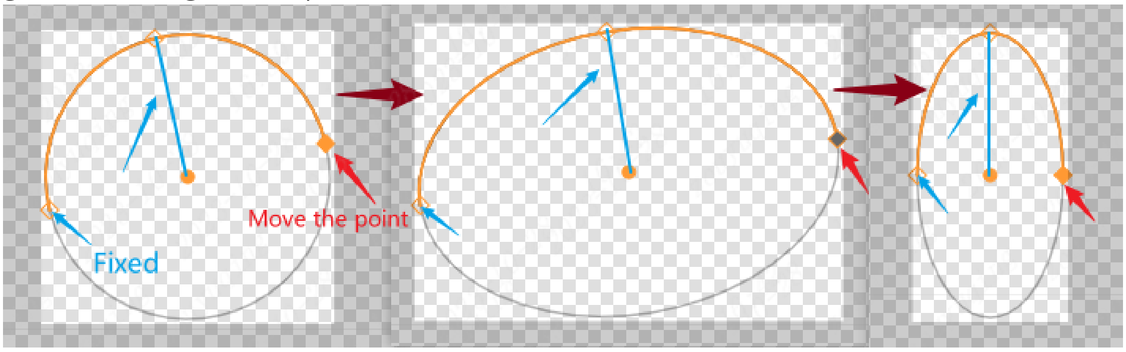
In the object tree on the left, when it is visible under a certain arc mask, the corresponding icon will be displayed in red as a prompt.



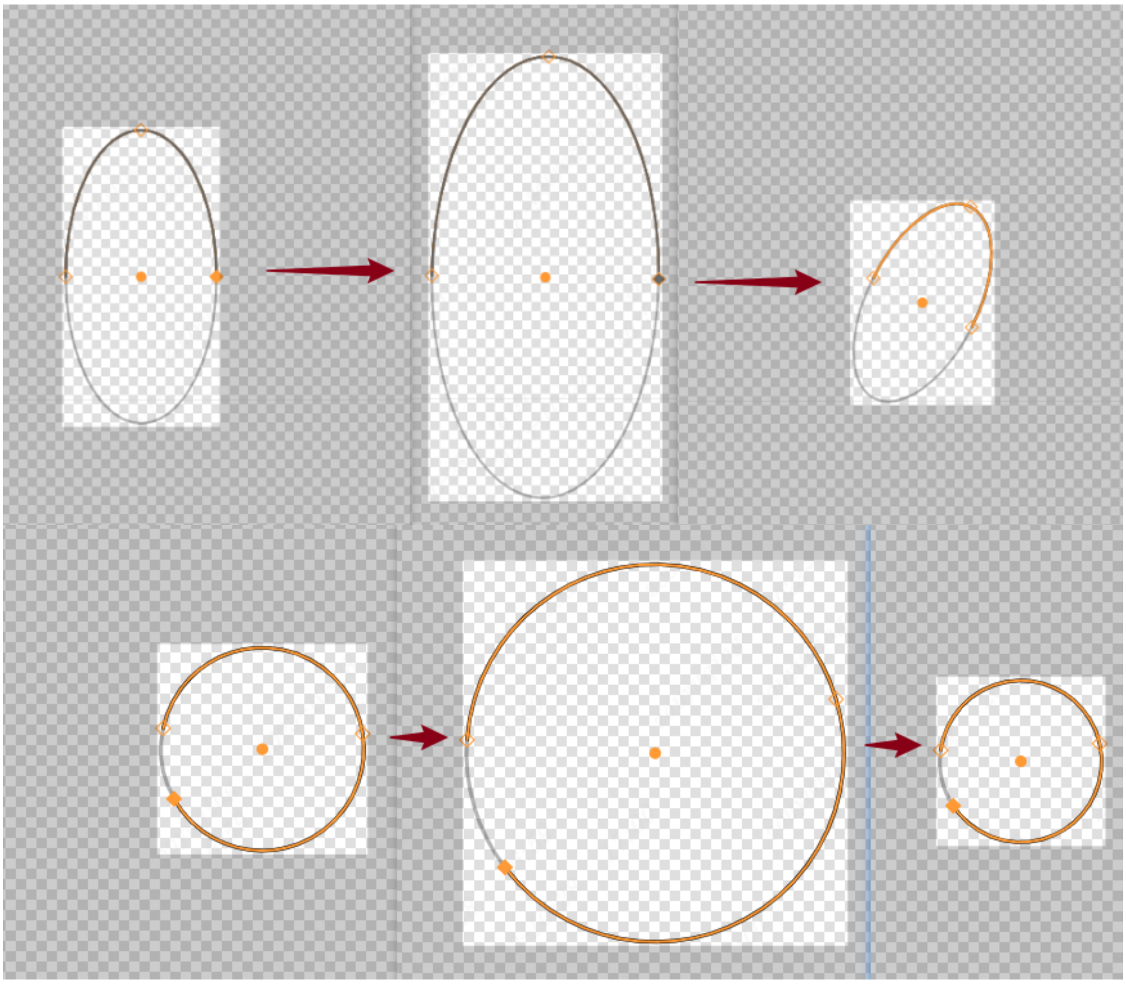
Arc control point

Start point, End point

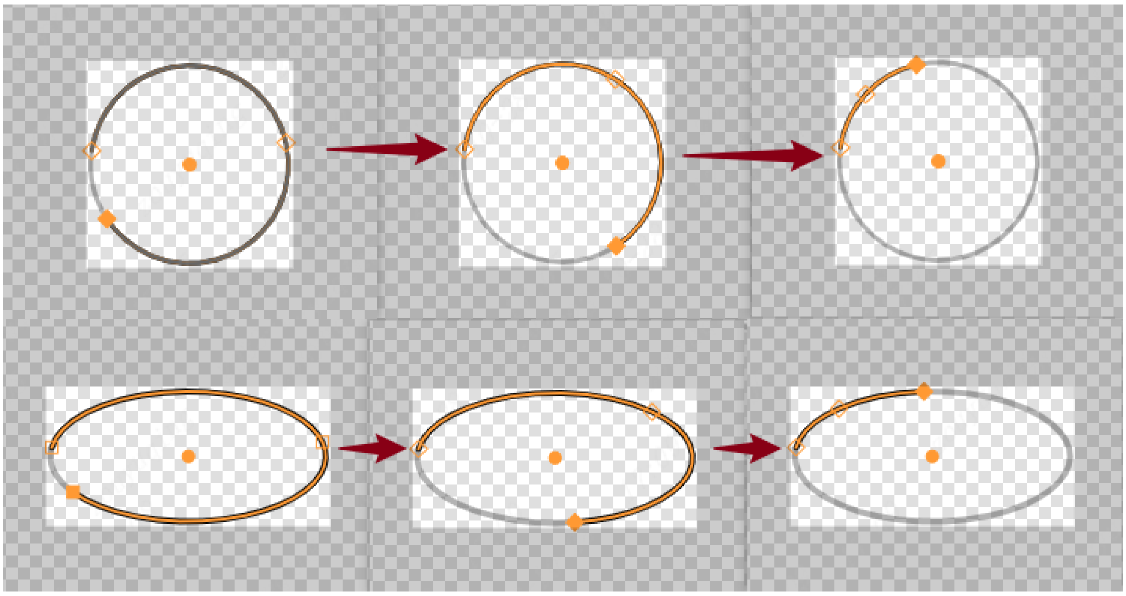
- The control points at the starting and ending points of the **two points arc** can be dragged arbitrarily. If the starting point of the arc is moved, the end point of the arc will remain, and the center of the circle will be recalculated and updated in real time. Conversely, the same goes for moving the end point of the arc.



Press Shift to move the starting point or end point of the arc, which will maintain the radius ratio of the arc, that is, it is always a positive arc or an elliptical arc.



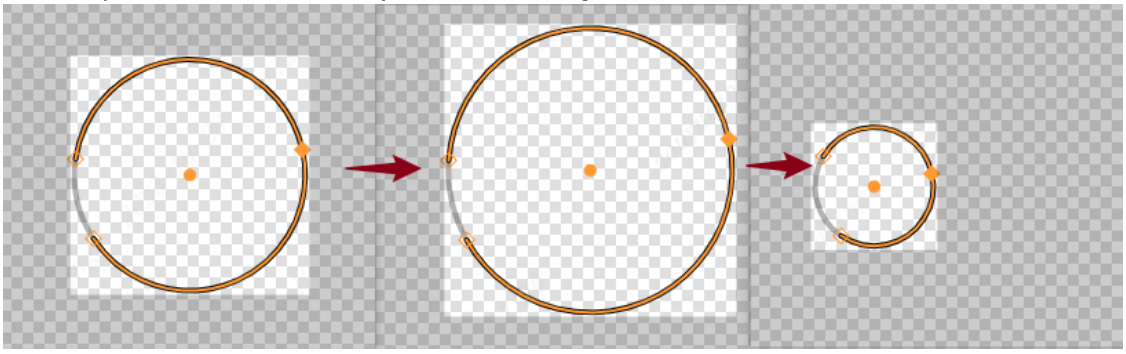
- The control points at the starting and ending points of the **central arc** can only move on the circumference, changing the starting and ending angles of the arc, and the length and radius of the arc are maintained.



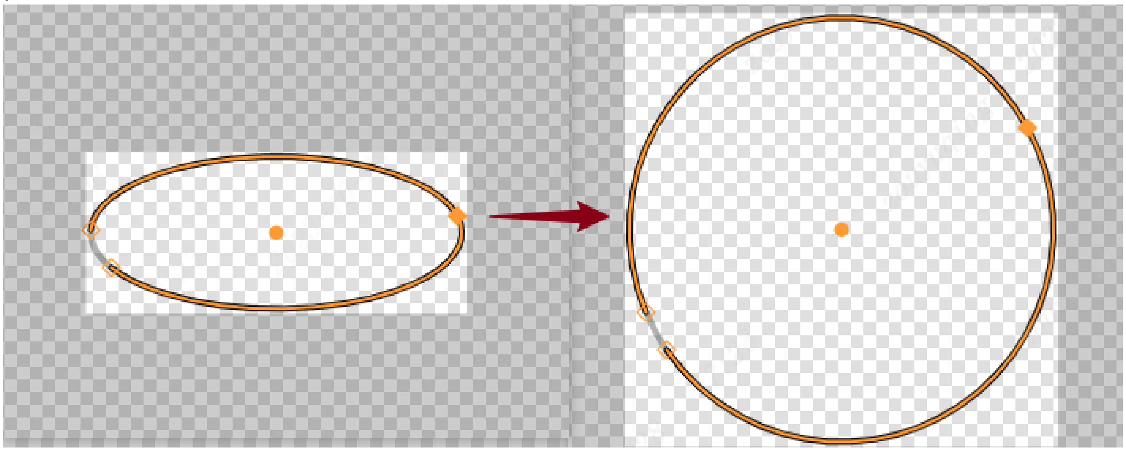
- **Ctrl switching type** , hold down the Ctrl key at the starting point and end point of the two-point arc to drag it to keep moving on the circumference, that is, temporarily switch to the center arc mode. (Conversely the same)

Midpoint of circular arc

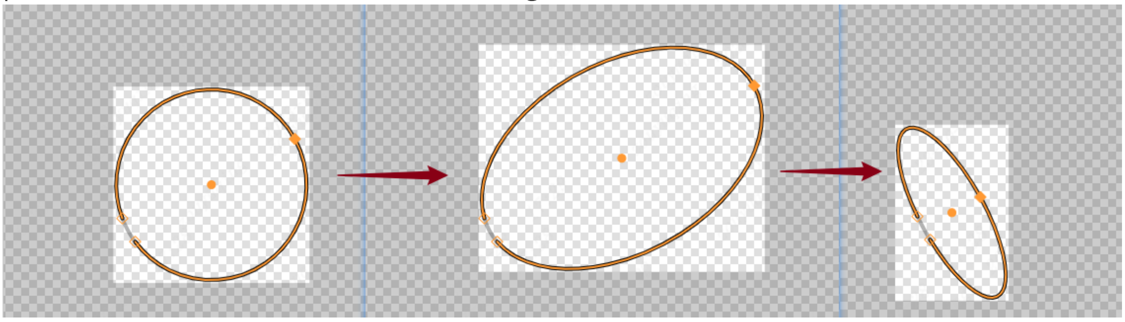
- The midpoint of the arc can only move in a straight line with the center of the circle.



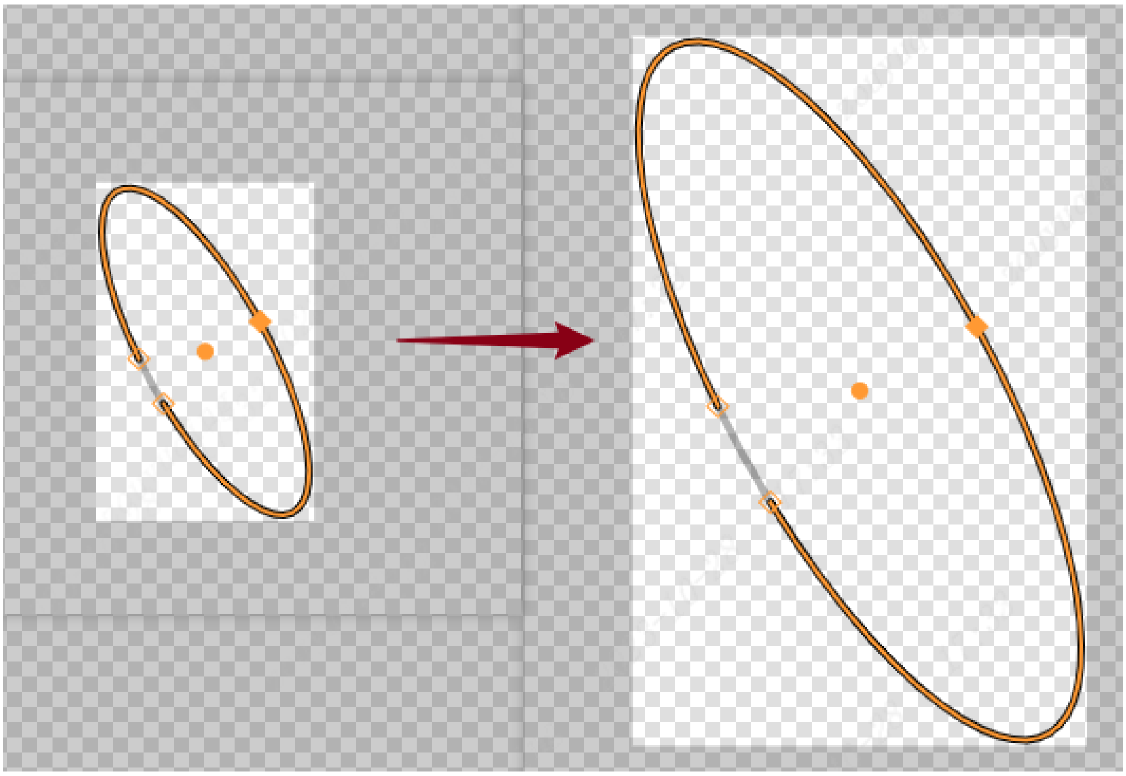
- Directly drag the midpoint of the arc, and the elliptical arc will automatically adjust to a positive arc.



- **Hold down the Alt key** and drag the midpoint of the arc to keep the starting and ending positions of the arc, and the ratio of the length and radius of the arc is not maintained.



- **Press and hold Alt key + Shift key** to drag the midpoint of the arc, it will keep the ratio of the length and radius of the arc, the same as the effect of center scaling.

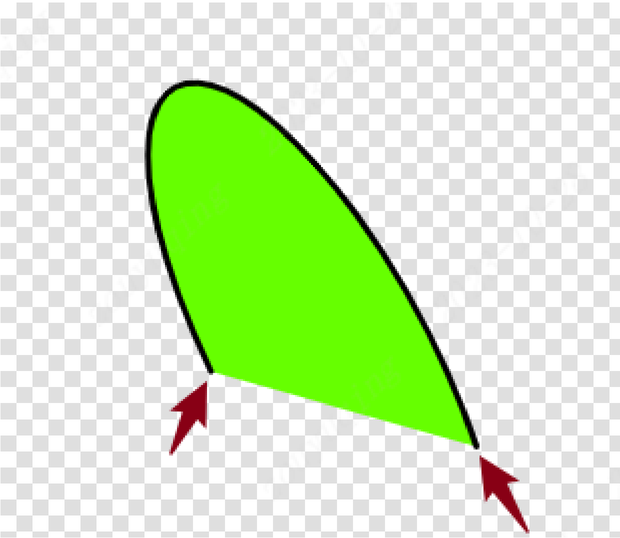


Center of circle

- Drag the center of the circle to move the entire arc in arc mask mode. You can also drag the entire arc at the arc line position of non-control points.

Arc filling

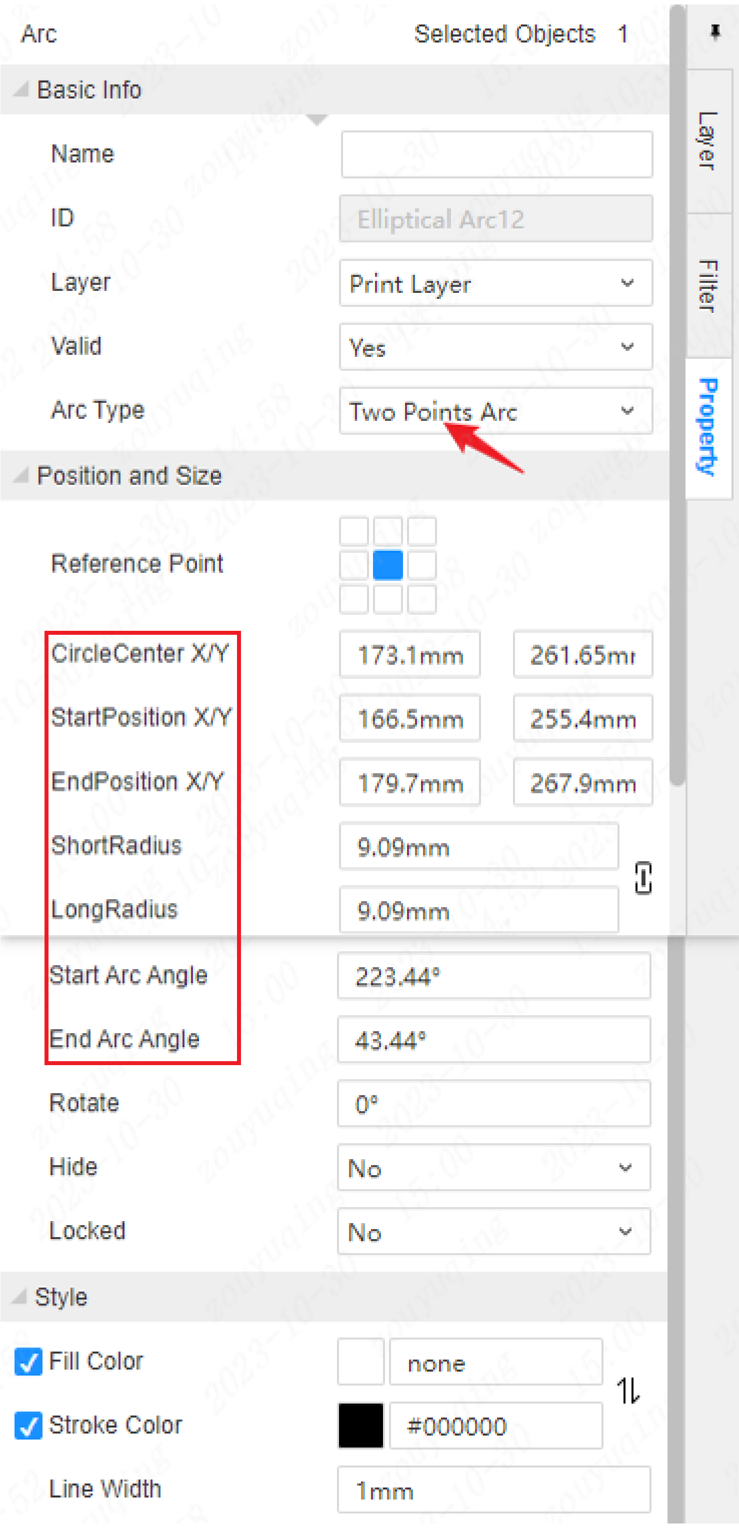
The filling area of the arc in the panel is the area within the closed path formed by the arc line connecting the starting and ending points.



Right property

The two points arc and the center arc can be switched and adjusted through the "Arc Type" attribute setting item on the right.

In the attribute panel on the right, you can also adjust the coordinates, length radius, start angle, end angle, and other attribute values of the arc control point.

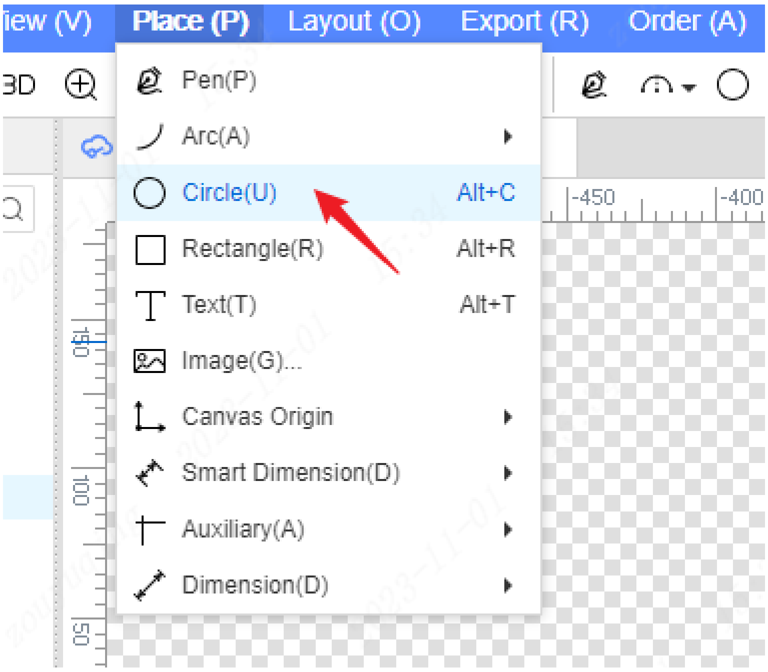


Circle

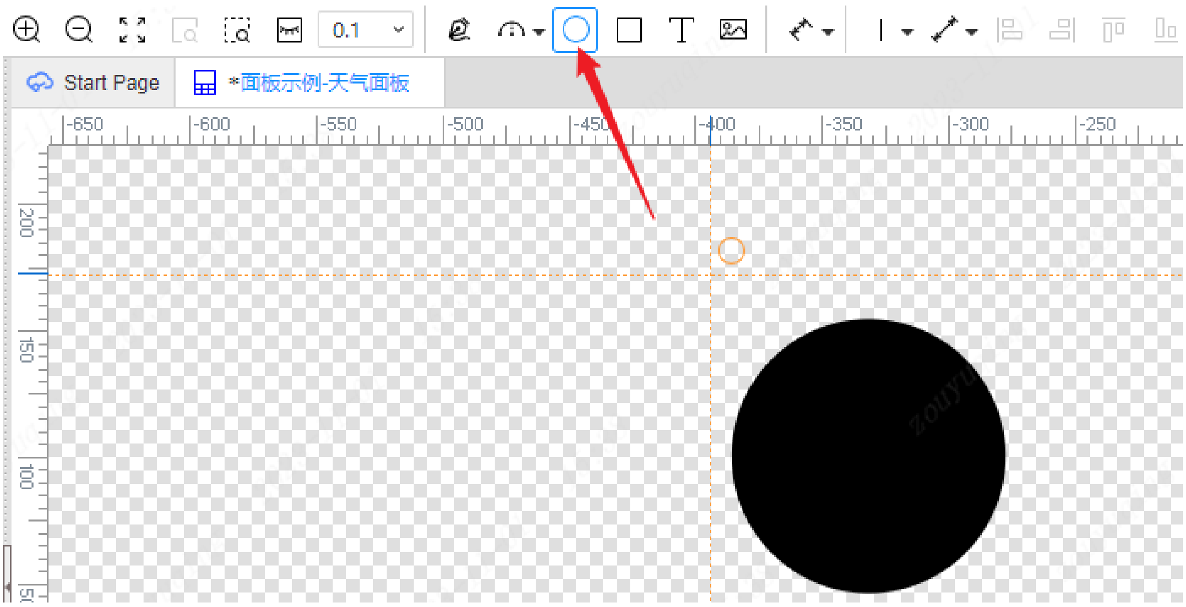
Place Circle

Support drawing circles on the panel. Circles and other graphics can be combined into multiple shapes.

- Top Menu Bar - Place - Circle
- Global shortcut：`Alt`+`C`

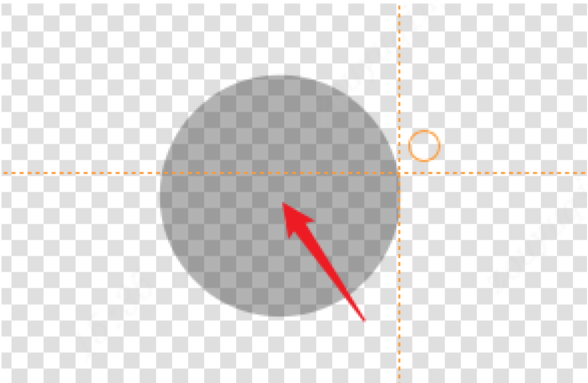


- By clicking the circular icon on the top toolbar, you can also quickly enter the circular drawing mode.

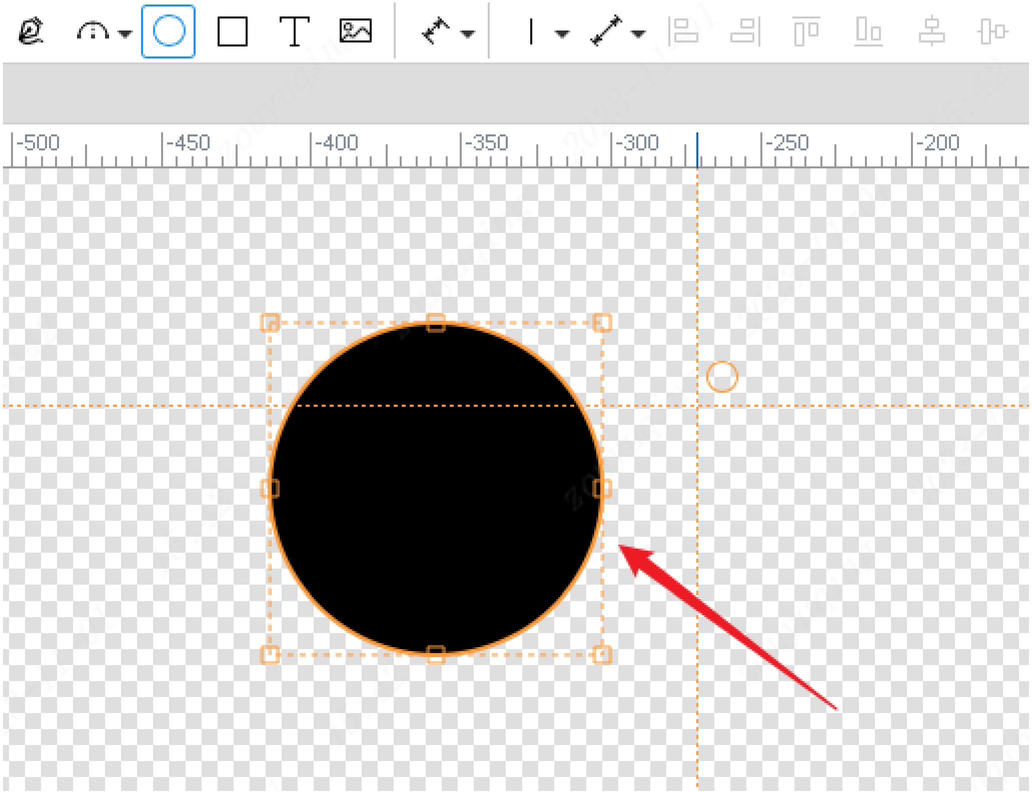


Operation steps

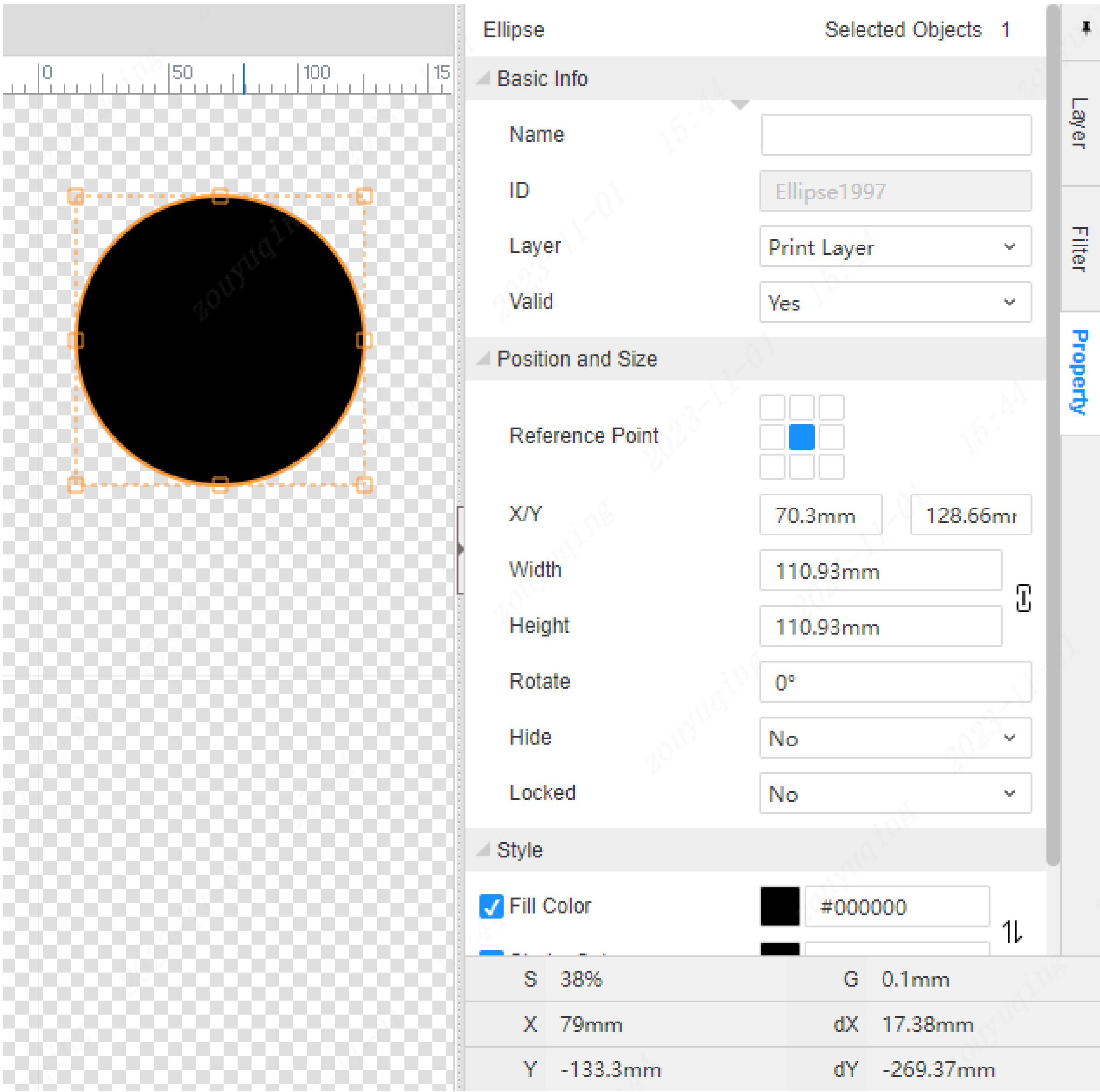
After entering the circular drawing mode, move the cursor to the canvas, follow the circular icon in the upper right corner, and the center of the circle can be determined by the first left mouse button at any position, displaying a circle to be determined with the distance from the center of the circle to the cursor as the radius.



Click the left mouse button again to determine the radius of the circle, complete the drawing of this circle, and enter the state of the next circle to be drawn.



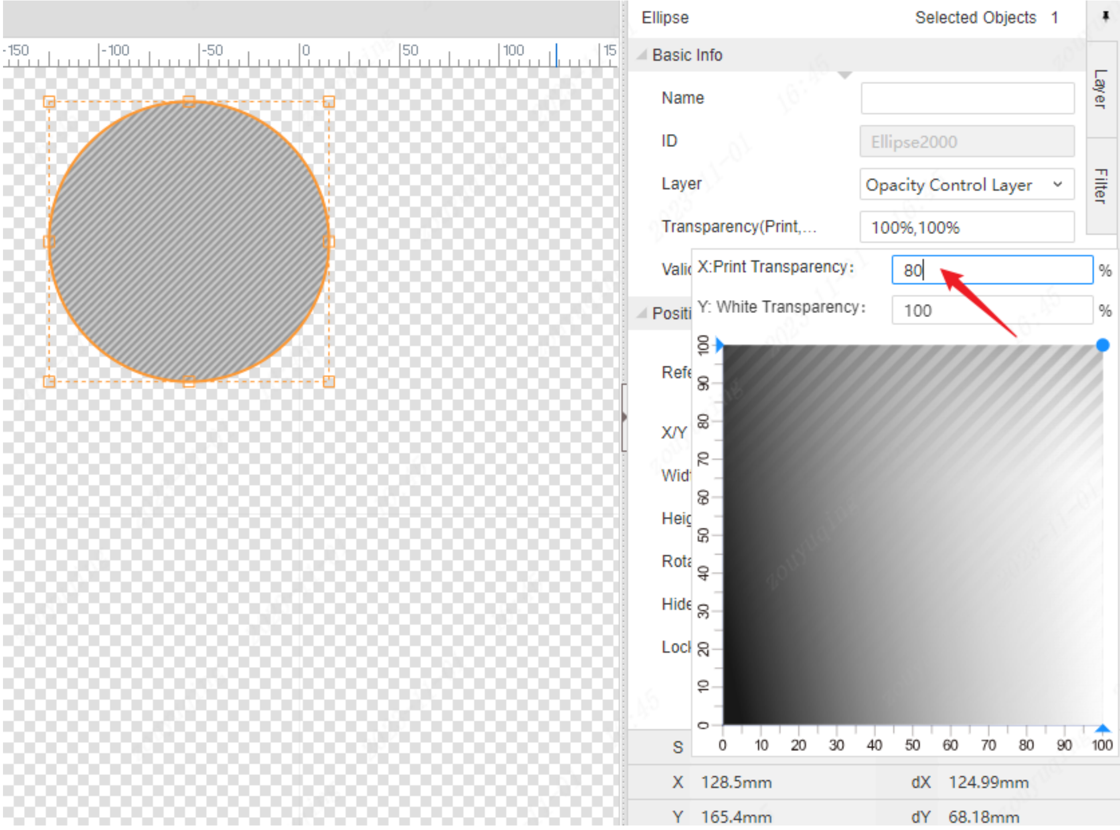
Right property



Basic Info

- **Name** : The name of the drawing, default is empty.
- **ID** : The exclusive ID of the drawing, which cannot be modified.
- **Layer** : The graphic can be switched to the desired layer to complete the corresponding function. For example, changing to a frame layer is cutting the outline line. Changing to a print layer will become a printable graphic. If you want to set the same graphic to a different layer, you can right-click - repeat to the original position, and then change the properties of one of them to another layer.
- **Transparency(Print,White)** : used to set the printing layer and the white layer transparent, you can directly enter the specific value, you can also quickly set the relevant transparent

parameters through the following coordinate selector.

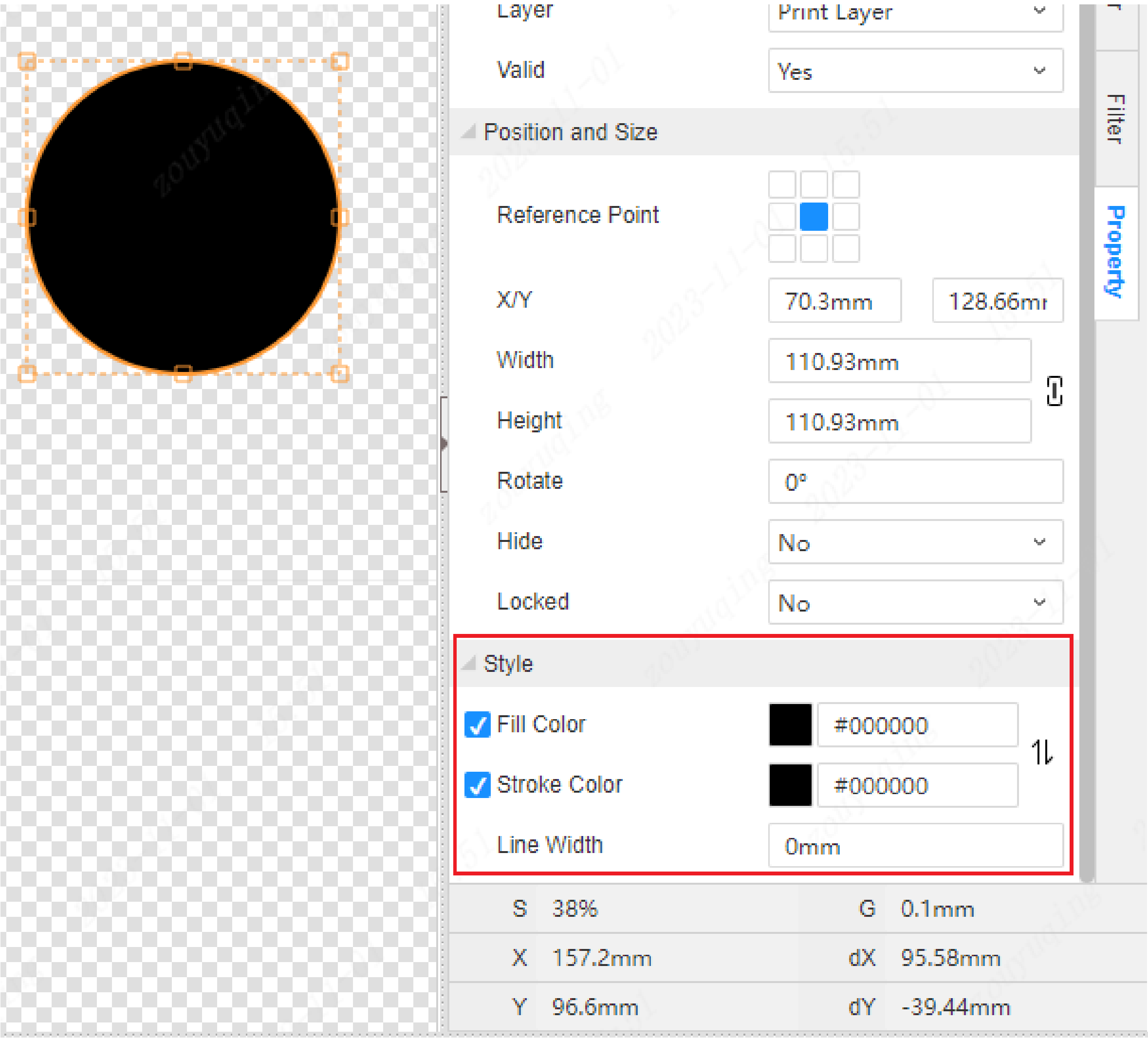


- **Valid** : You can choose whether this graphic is effective and in the design and production. Select **Yes** to produce the corresponding graphic. Select **No** side will not.

Position and Size

- **Reference point** : Set the middle point as the reference point, and adjust the concentric circle.
- **X/Y** : The X and Y axis coordinate values at the reference point of the graphic on the design drawing.
- **Width, Height** : The shape's axis aligns with the width and height of the bounding rectangle.
- **Rotate** : Set the rotation angle of the drawing.
- **Hide** : Sets whether the drawing is hidden or not on the canvas.
- **Locked** : Sets whether the drawing is locked on the canvas or not.

Style



- **Fill Color** : Set the fill color of the graphic, check whether to apply the fill color to the graphic, if not checked, there will be no fill
- **Stroke Color** : Set the stroke color of the graphic, check whether to apply the stroke color to the graphic, if not checked, there will be no stroke
- **Line Width** : The width setting of the drawing stroke.

Rectangle

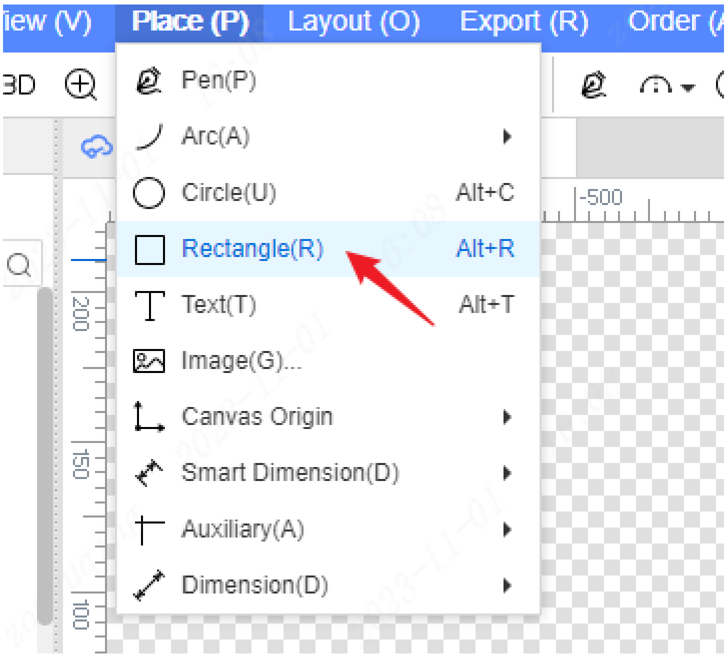
Rectangle

Rectangles can be drawn on the panel. Rectangles and other graphics can be combined into multiple shapes.

Function entry

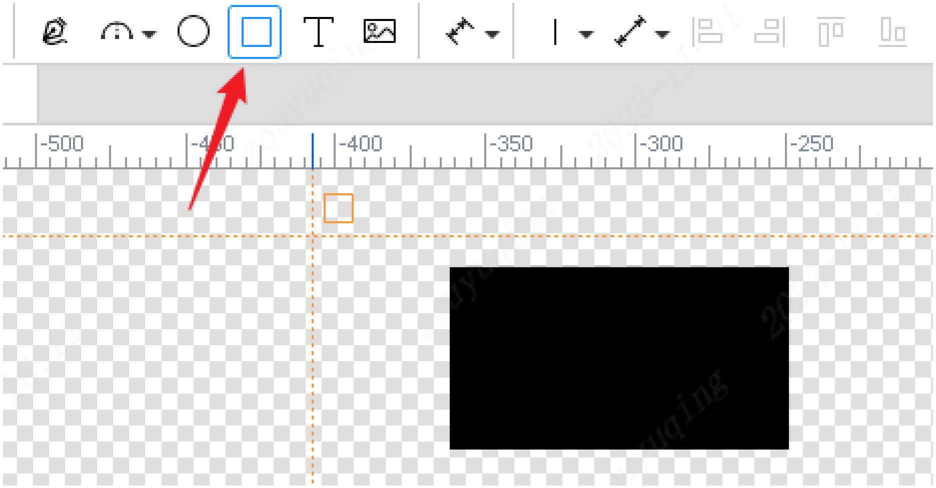
Top menu

- Top Menu - Place - Rectangle
- Menu shortcut: P+R
- Global shortcut: Alt+R



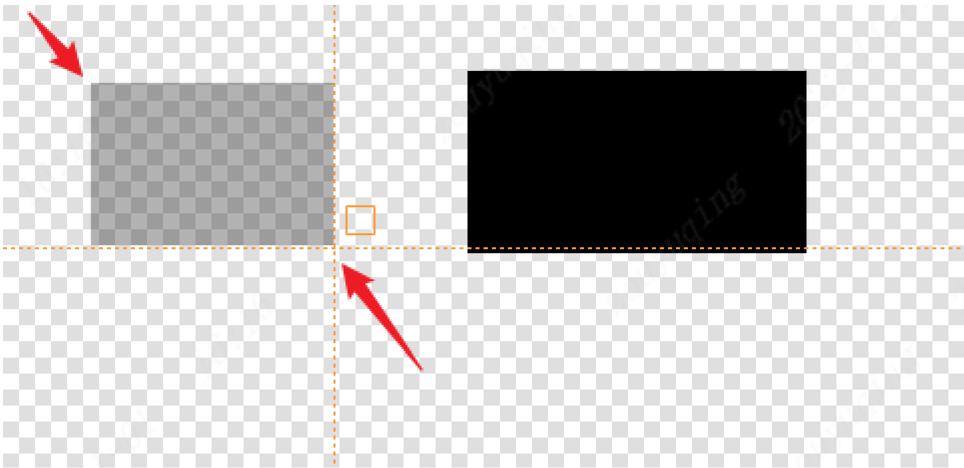
Top toolbar

- You can also quickly enter the rectangular drawing mode by clicking the rectangle icon on the top toolbar.

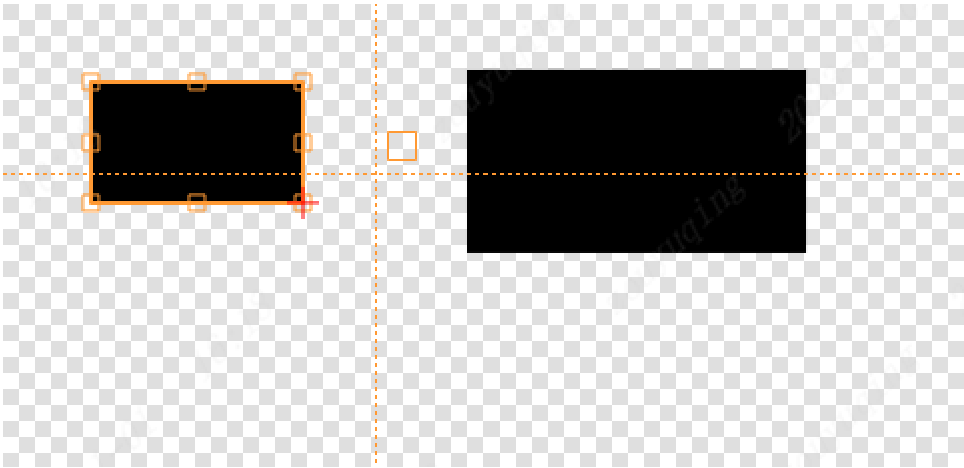


Operation steps

After entering the drawing mode of the rectangle, moving the cursor to the upper right corner of the canvas will display the rectangle icon. The first left mouse button at any position can determine a vertex of the rectangle, move the cursor, and display a rectangle to be determined between the determined vertex and the cursor point.



Click the left mouse button again to determine another symmetrical vertex of the rectangle, complete the drawing of this rectangle, and enter the state of the next rectangle to be drawn.



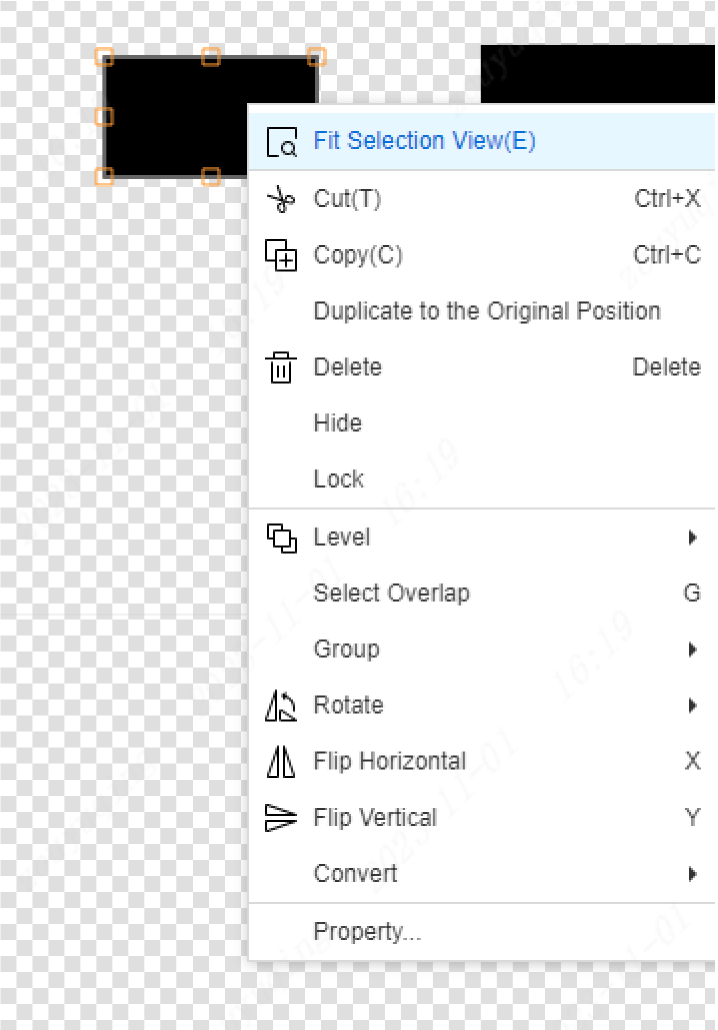
Right-click menu

To edit the relevant properties of a rectangle on the canvas, right-click the rectangle and operate the corresponding right-click menu item for editing.

Hide, lock

The "Hide" menu displayed by the right-click menu represents the current display state of the object, and operating this menu item will hide the selected object.
Similarly, the "Lock" menu displayed by the right-click menu represents the current unlocked state of the object, and operating this menu item will lock the selected object.

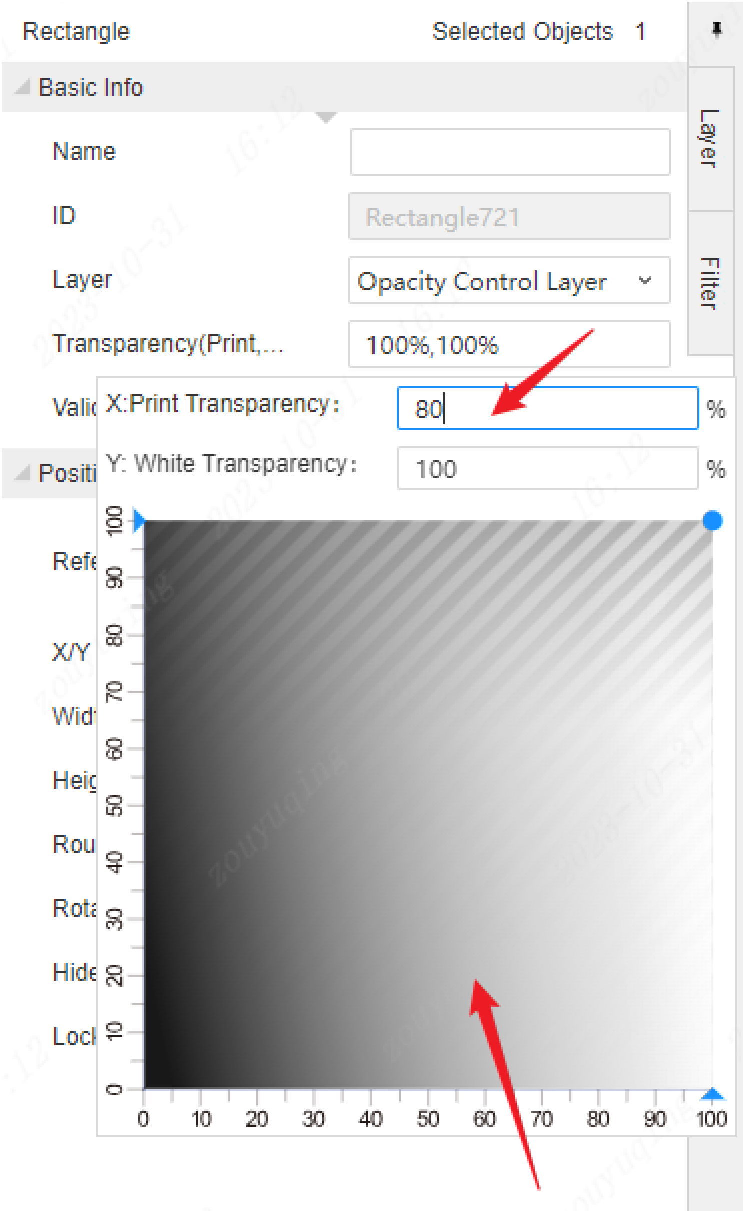
Right Property



Basic Info

- **Name** : The name of the drawing, default is empty.
- **ID** : The exclusive ID of the drawing, which cannot be modified.
- **Layer** : The graphic can be switched to the desired layer to complete the corresponding function. For example, changing to a frame layer is cutting the outline line. Changing to a print layer will become a printable graphic. If you want to set the same graphic to a different layer, you can right-click - repeat to the original position, and then change the properties of one of them to another layer.
- **Transparency(Print,White)** : used to set the printing layer and the white layer transparent, you can directly enter the specific value, you can also quickly set the relevant transparent

parameters through the following coordinate selector.

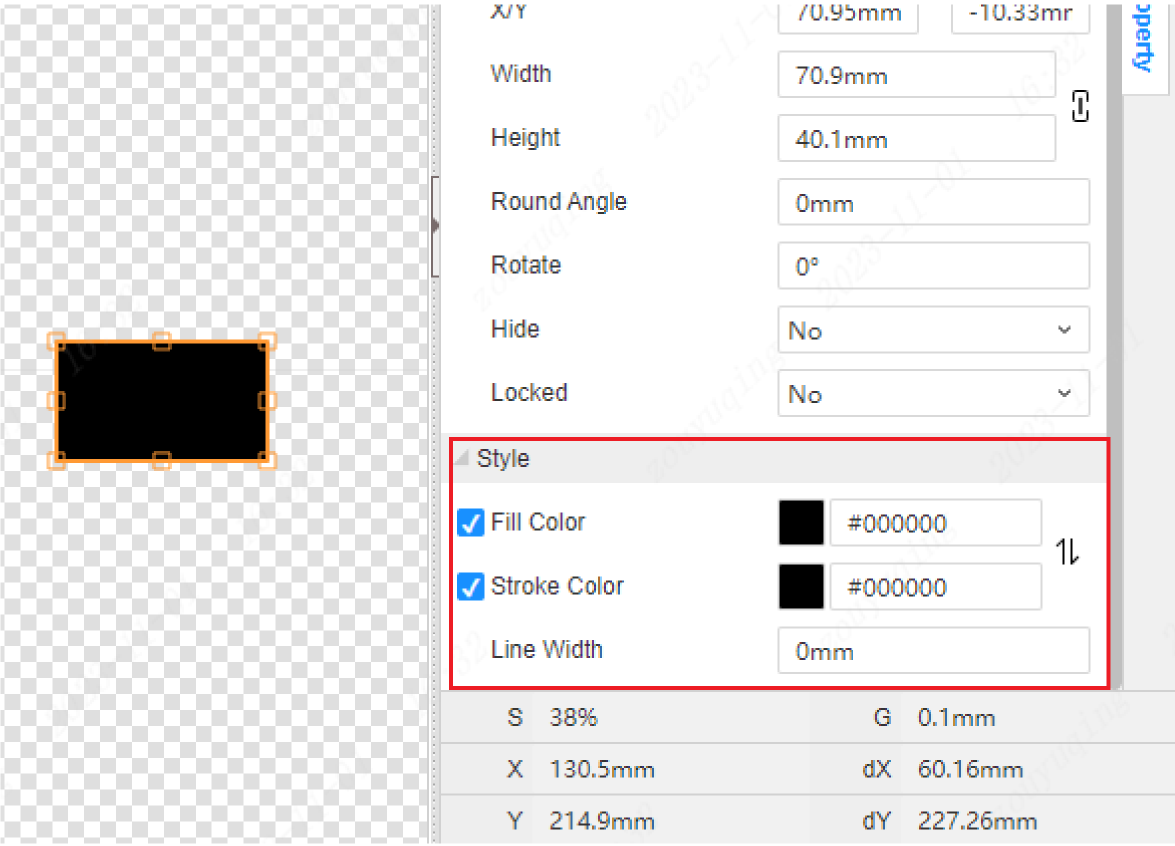


- **Valid** : You can choose whether this graphic is effective and in the design and production. Select **Yes** to produce the corresponding graphic. Select **No** side will not.

Position and Size

- **Reference point** : Set the middle point as the reference point, and adjust the concentric circle.
- **X/Y** : The X and Y axis coordinate values at the reference point of the graphic on the design drawing.
- **Width, Height** : The shape's axis aligns with the width and height of the bounding rectangle.
- **Rotate** : Set the rotation angle of the drawing.
- **Hide** : Sets whether the drawing is hidden or not on the canvas.
- **Locked** : Sets whether the drawing is locked on the canvas or not.

Style



- **Fill Color** : Set the fill color of the graphic, check whether to apply the fill color to the graphic, if not checked, there will be no fill
- **Stroke Color** : Set the stroke color of the graphic, check whether to apply the stroke color to the graphic, if not checked, there will be no stroke
- **Line Width** : The width setting of the drawing stroke.

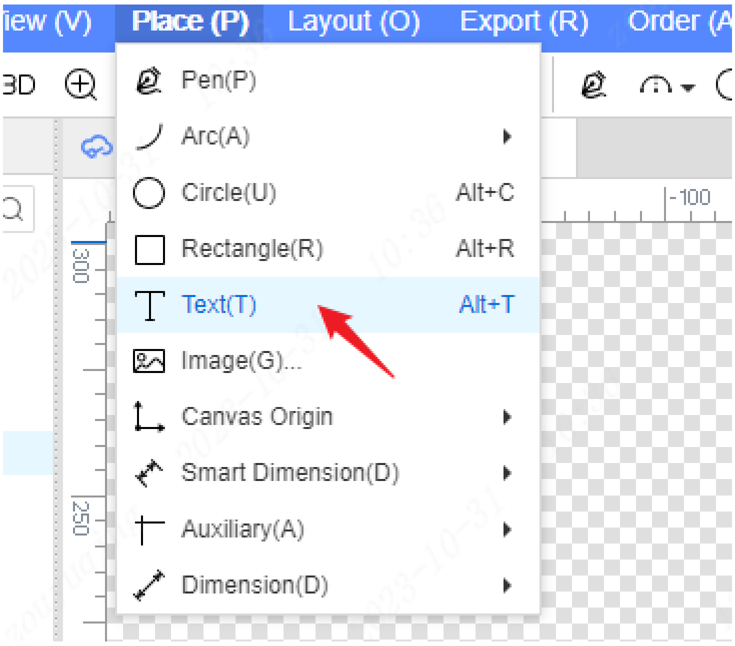
Text

Text

Place text when designing the panel

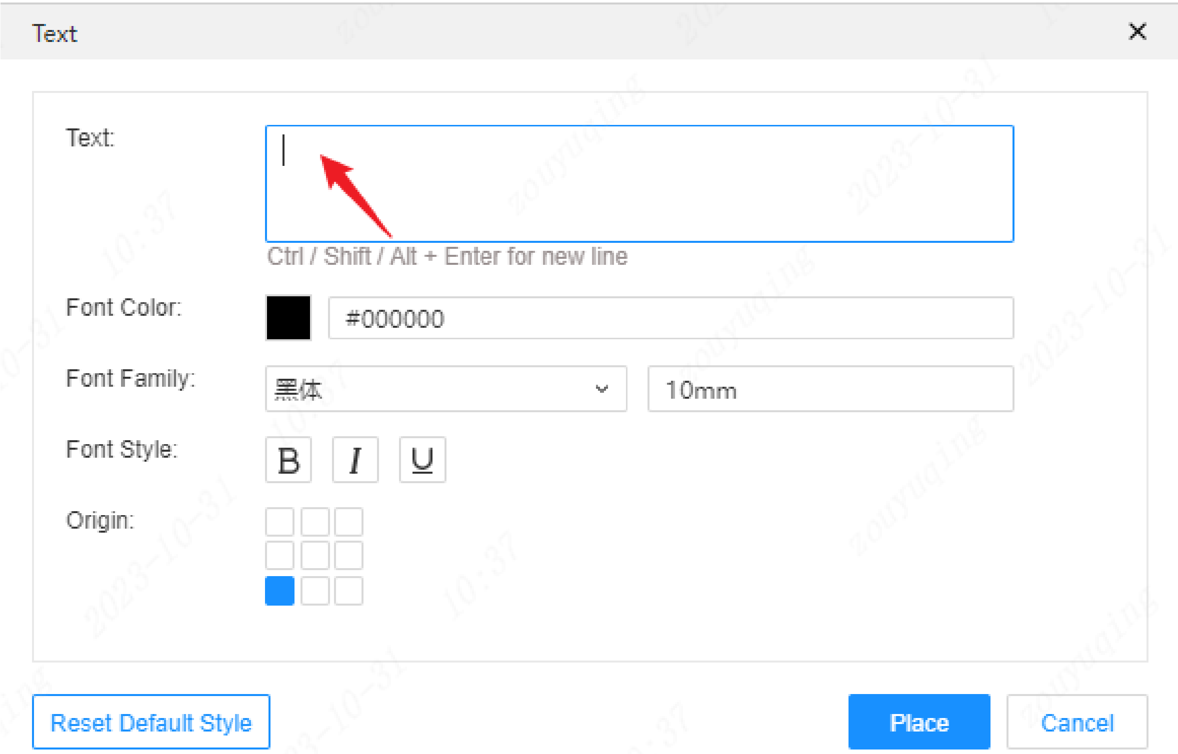
Operation steps

- Top Menu Bar - Place - Text
- Menu shortcut: P+T
- Global shortcut: Alt+T



Text pop-up

After the operation places the text, a text creation pop-up window will pop up.



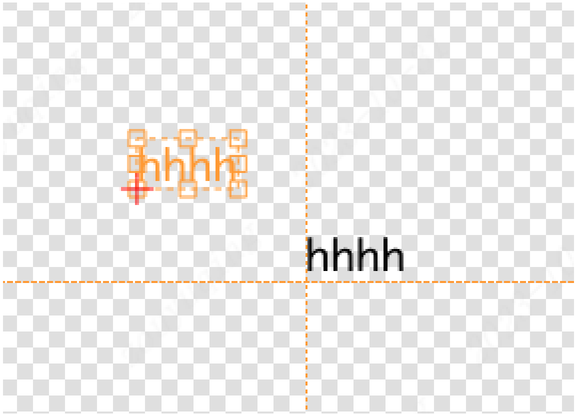
- Text wrap shortcut： `Ctrl` + `Enter` or `Shift` + `Enter` or `Alt` + `Enter`

Font color, font, font size, font style, and font origin all support custom adjustments.

The Restore Default Style button in the lower left corner of the pop-up window can reset the default style of the text, that is, restore the theme style settings of the font.

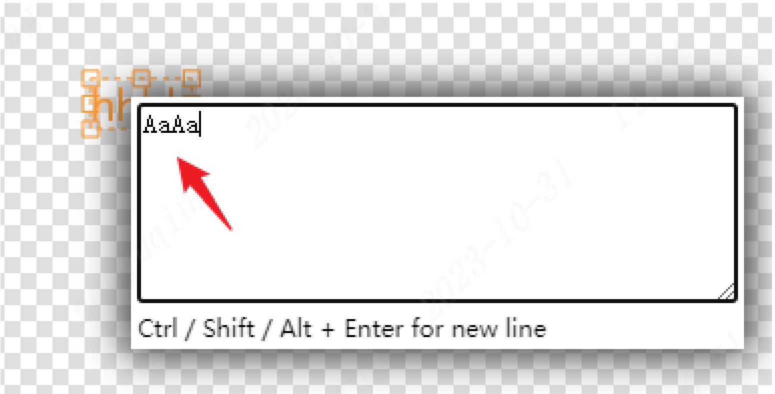
Place text

After clicking the Place button, the display text follows the cursor. The left mouse button on the canvas can complete the placement of one text and enter the to-be-placed state of the next text. The right mouse button will exit the text placement mode.

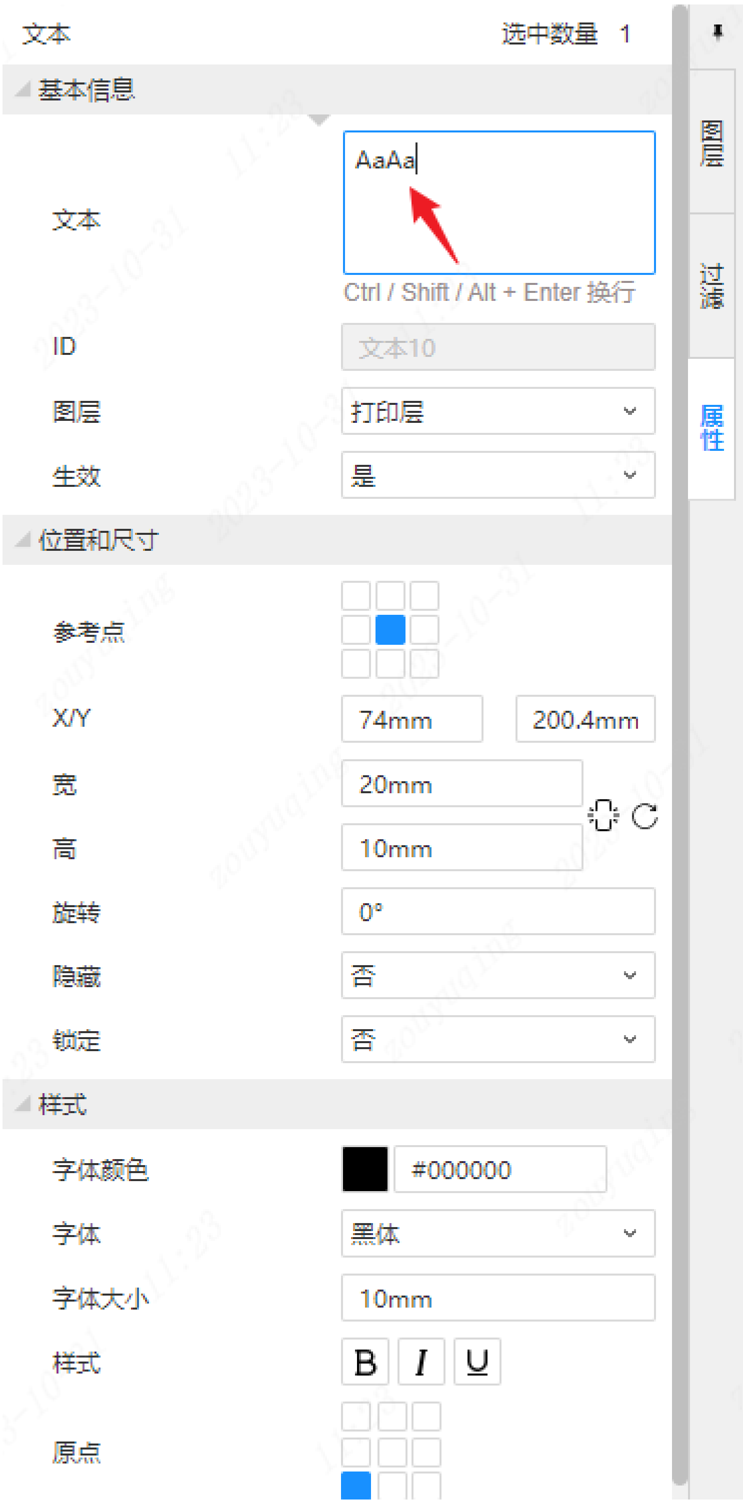


Modify text content

Double-click the text on the canvas to pop up the text modification pop-up window to adjust the text content.



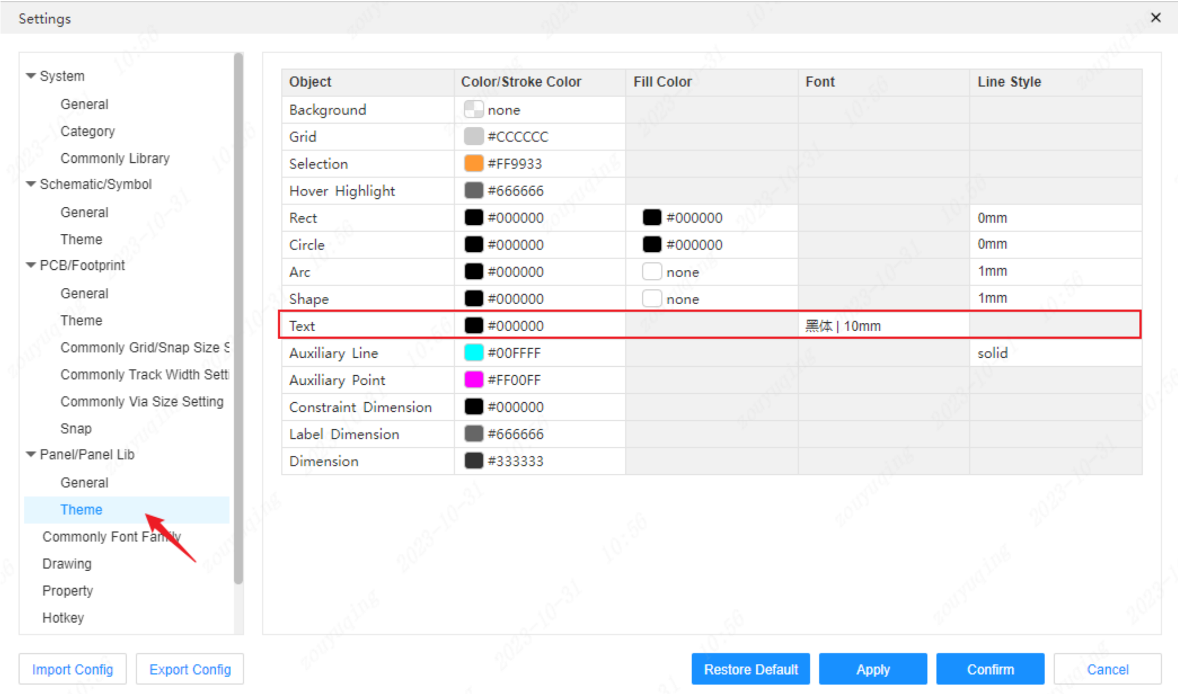
The properties panel on the right also supports modifying text content, which will be synchronously updated to the canvas after completion.



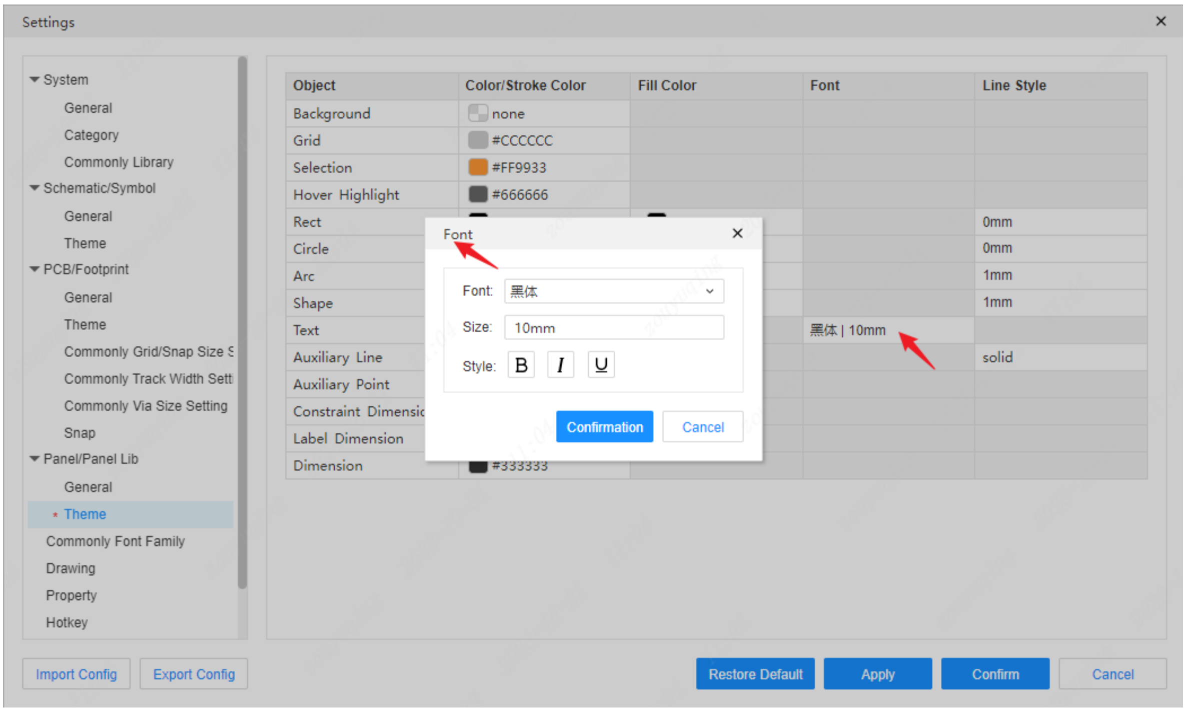
Other style settings of the text can also be adjusted through the properties on the right, which will be synchronously updated to the canvas.

Text topic

The theme of the text can be customized and adjusted in the settings pop-up window. The default color is #000000 in bold, and the font size defaults to 10mm.



Click "Black | 10mm" in the font column to pop up the font settings pop-up window, where you can customize the default font style.



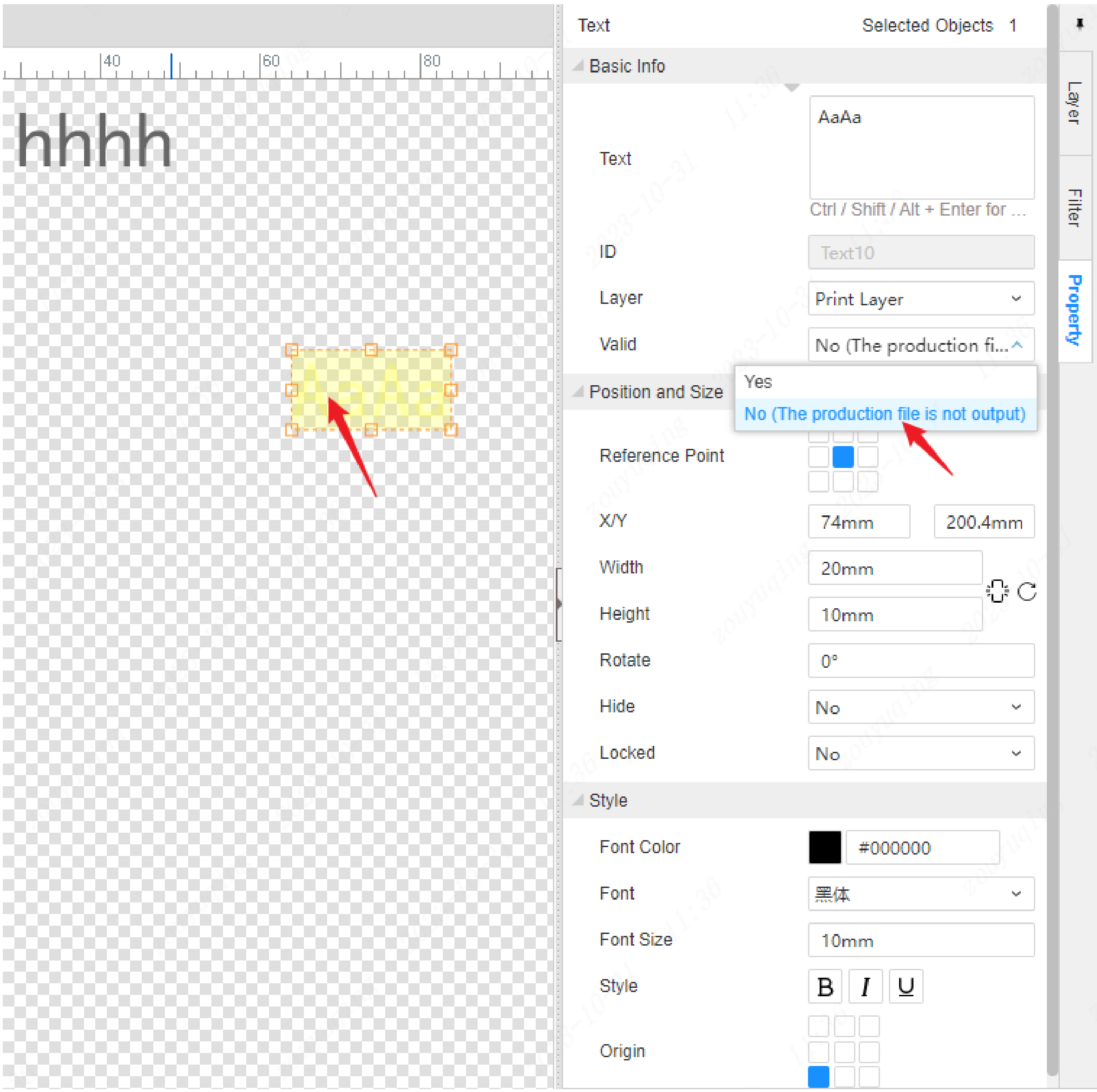
The "Restore Defaults" button in the pop-up window of the theme settings can reset the default theme style settings of the font.

Note: After changing the theme settings, you should operate the "Apply" button to make the changes take effect. A short prompt will appear in the center of the canvas: Configuration saved successfully.

Rules and Agreements

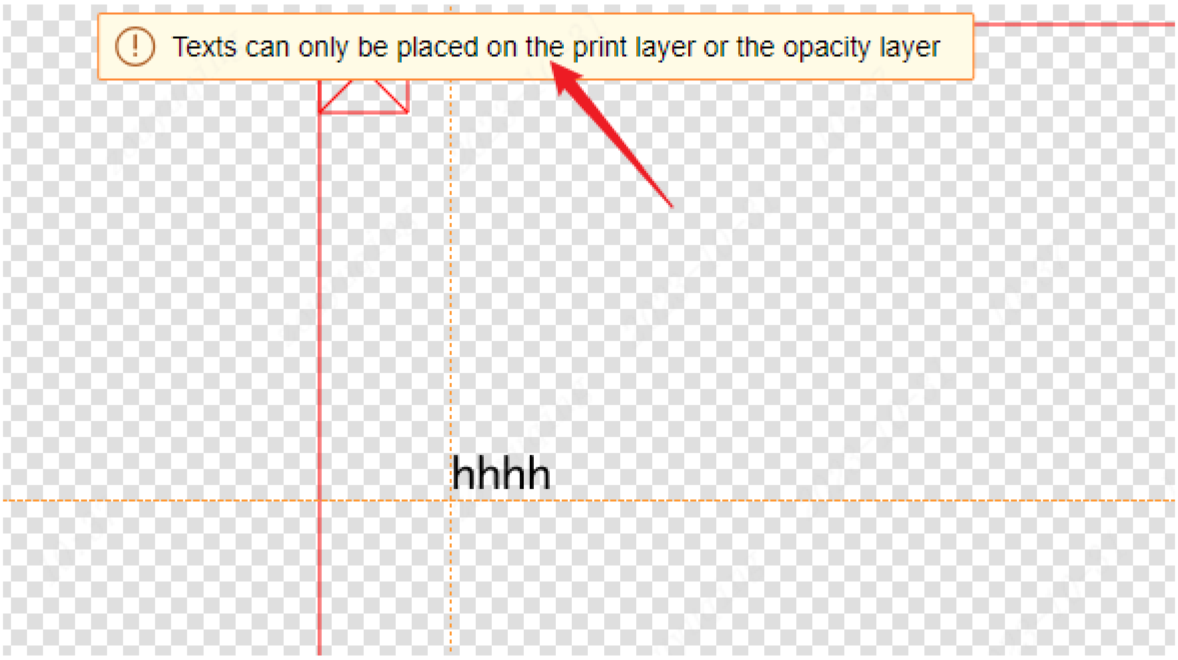
Remark text

If it needs to be used only as a note text, you can set the text property "effective" to no, or put it outside the red material boundary of the panel, and the exported production file will not be exported at that time.



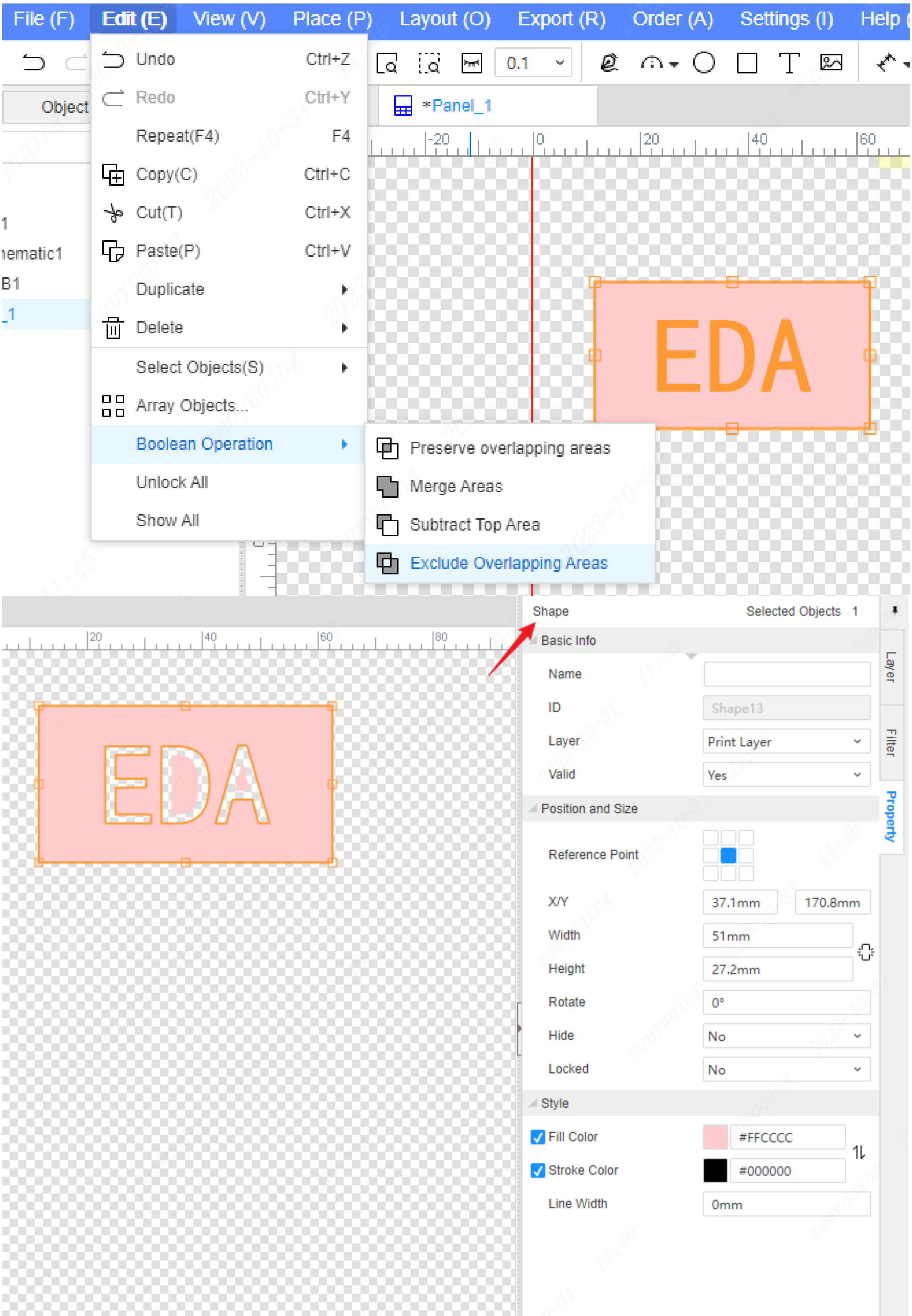
Layer where the text is located

Text can only be placed in the "Print Layer" or "Transparent Control Layer". When the current editing layer is another layer, placing text will automatically switch the current editing layer to the print layer by default, and the short prompt text in the center of the canvas can only be placed in the print layer or transparent control layer.



Text-boolean operation

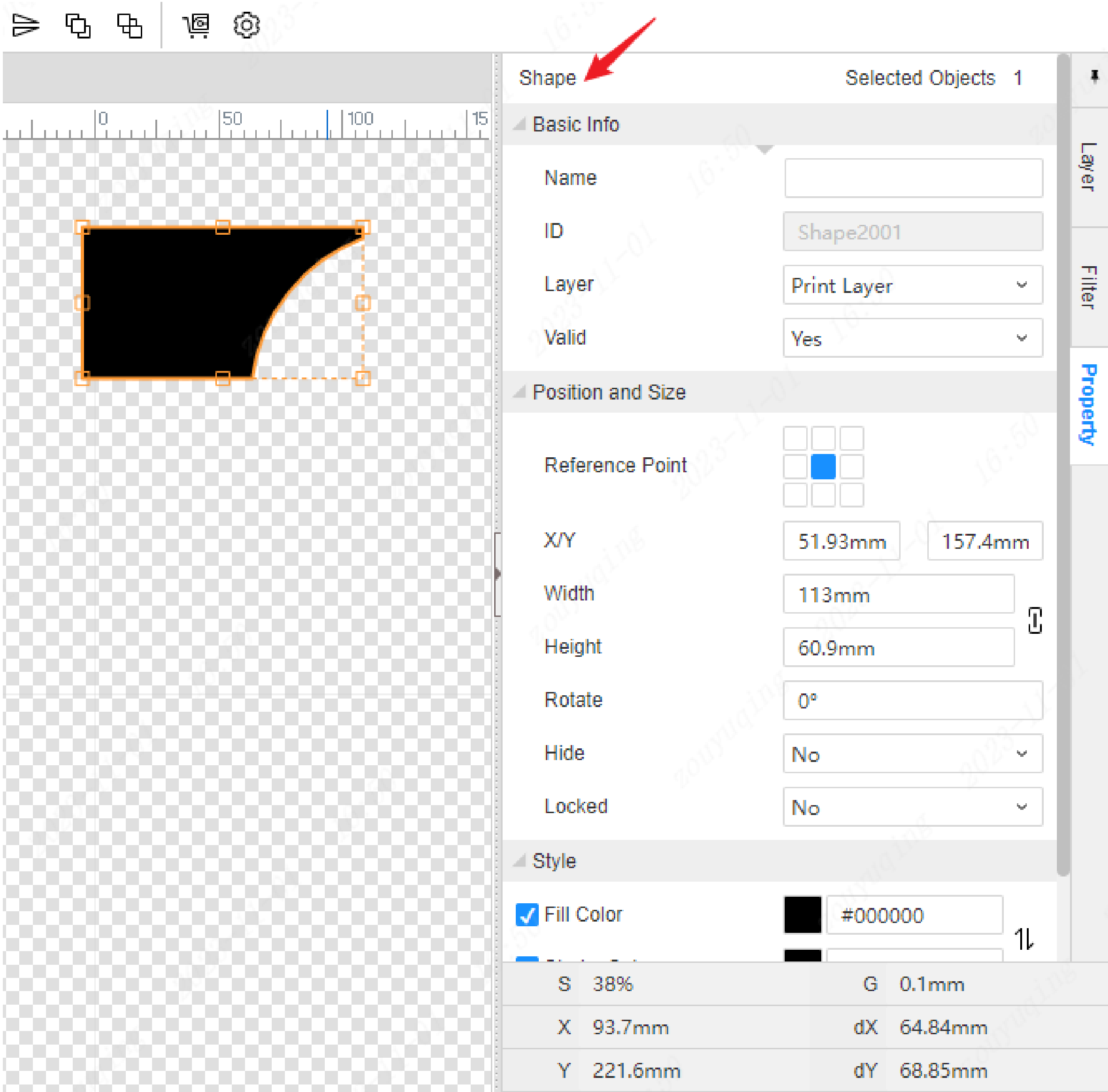
Perform Boolean operations on text and other elements, and after the operation, the attributes become shapes, which can further edit the contour path.



Shape

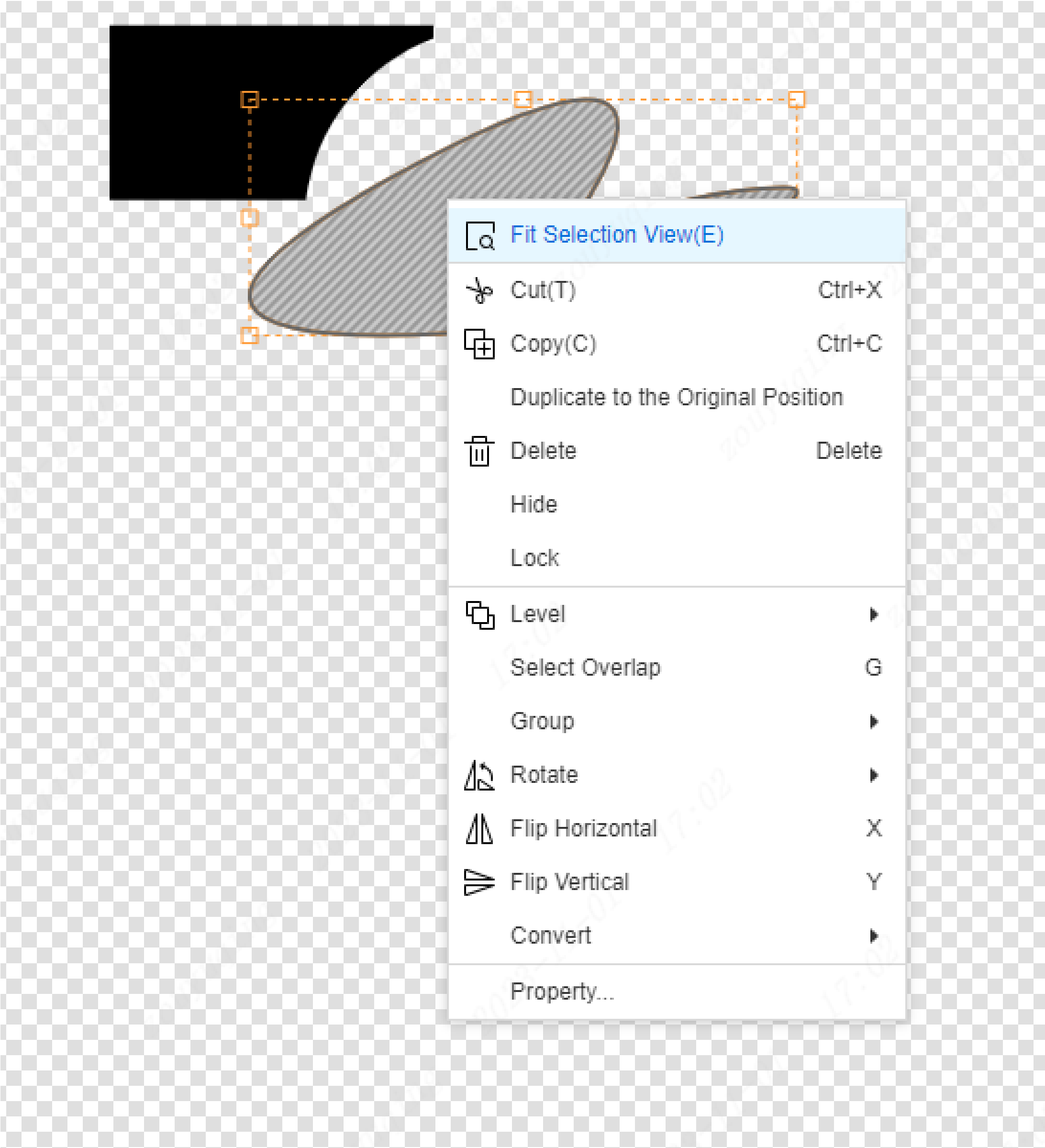
Shape

When a primitive is right-converted, or an operation such as intersection is performed, a primitive called a shape will be generated, which belongs to a polygon.



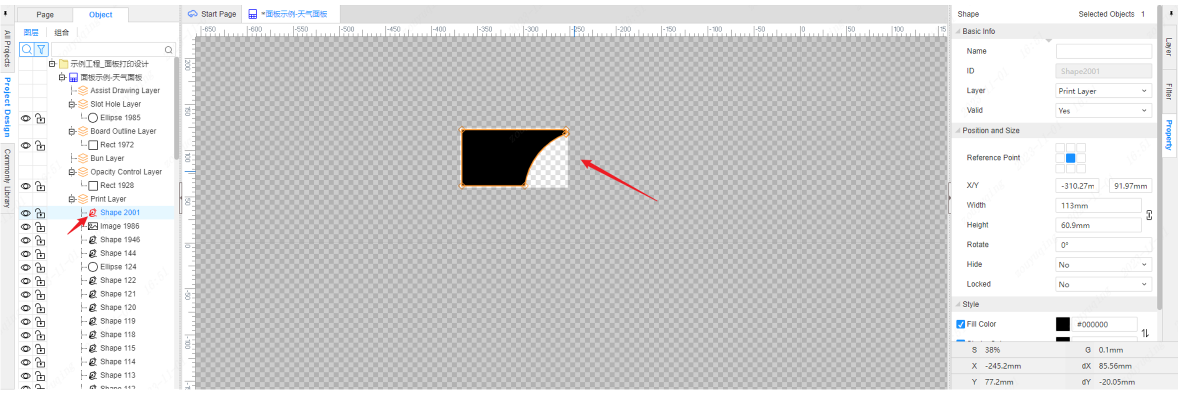
Right-click menu

To edit the relevant properties of a shape on the canvas, you can select the shape and right-click the menu operation.



Shape Mask

The first click on the shape will select all (8-point control mode).
Double-clicking will enter the shape mask and switch to the edit contour state, as shown in the control point below:

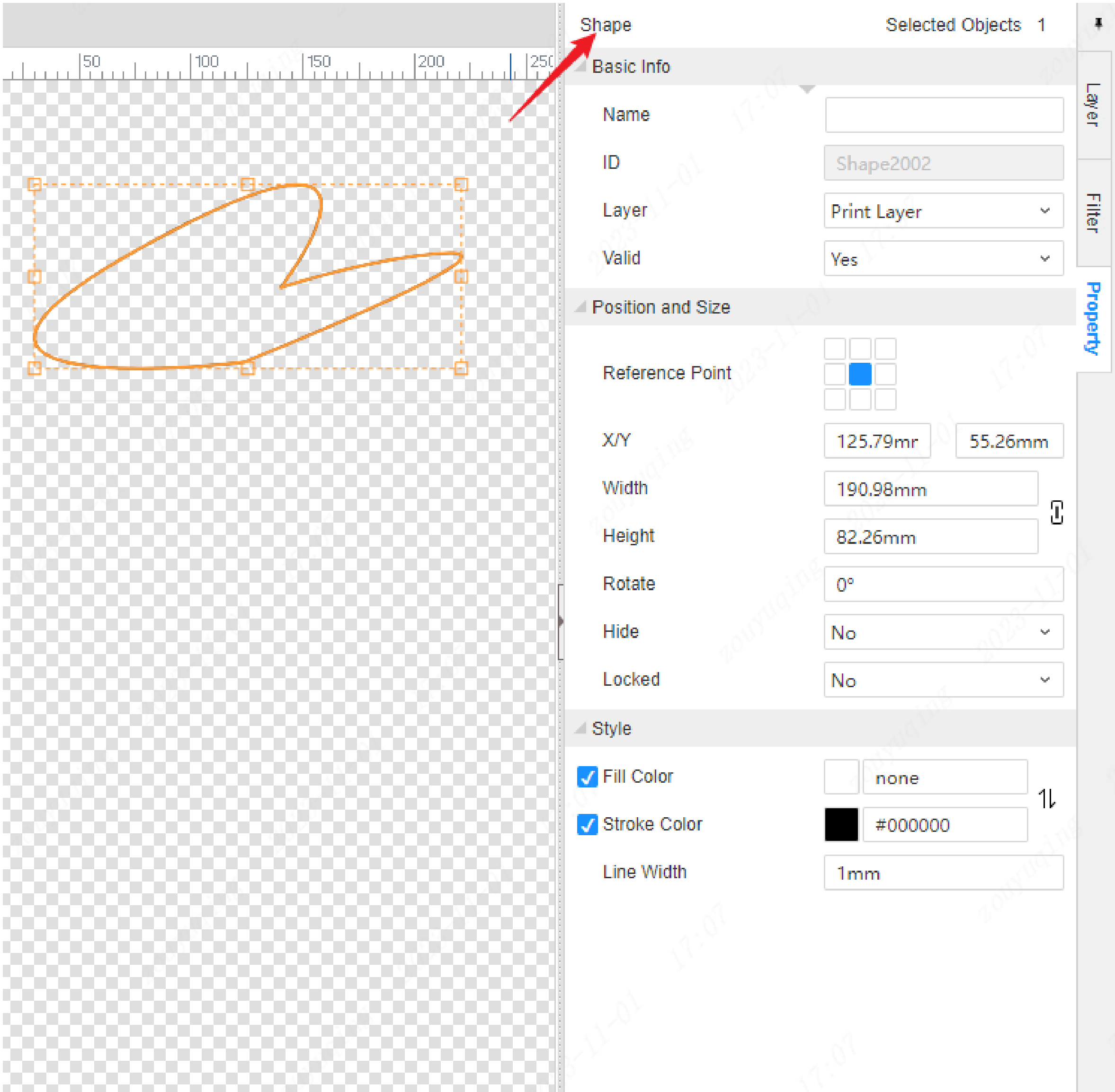


The shape can be easily adjusted through this operation control point.

Double-click the canvas blank or press Esc to exit the shape mask.

In the object tree on the left, the shape icon is displayed in red to indicate that the current canvas is in mask mode for that shape.

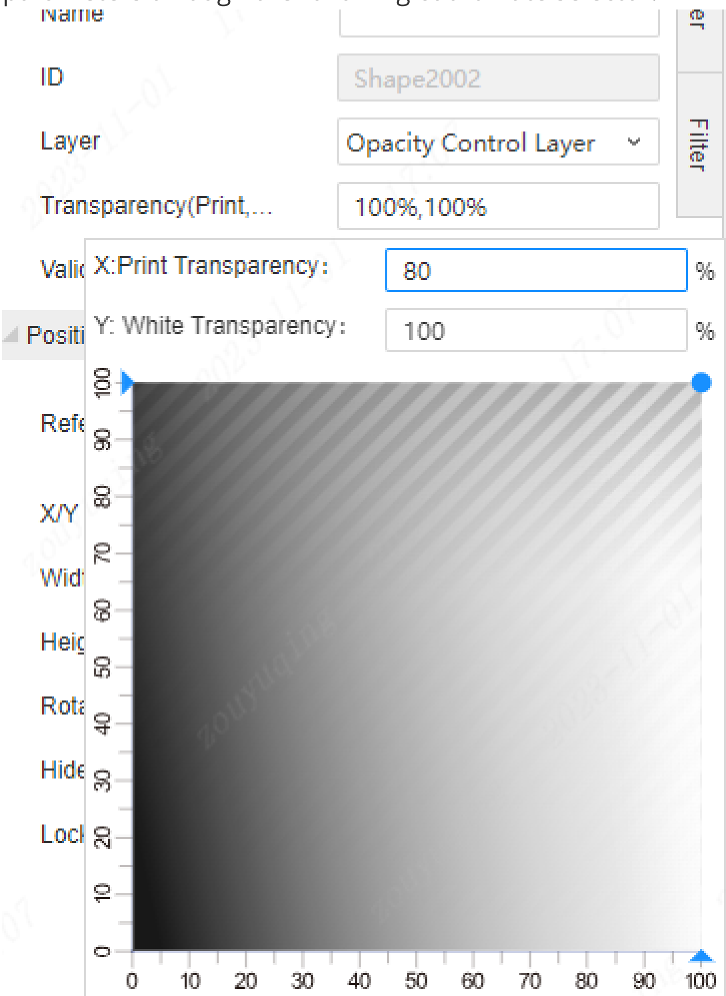
Right Property



Basic Info

- **Name** : The name of the drawing, default is empty.
- **ID** : The exclusive ID of the drawing, which cannot be modified.
- **Layer** : The graphic can be switched to the desired layer to complete the corresponding function. For example, changing to a frame layer is cutting the outline line. Changing to a print layer will become a printable graphic. If you want to set the same graphic to a different layer, you can right-click - repeat to the original position, and then change the properties of one of them to another layer.
- **Transparency(Print,White)** : used to set the printing layer and the white layer transparent, you can directly enter the specific value, you can also quickly set the relevant transparent

parameters through the following coordinate selector.



- **Valid** : You can choose whether this graphic is effective and in the design and production. Select **Yes** to produce the corresponding graphic. Select **No** side will not.

Position and Size

- **Reference point** : Set the middle point as the reference point, and adjust the concentric circle.
- **X/Y** : The X and Y axis coordinate values at the reference point of the graphic on the design drawing.
- **Width, Height** : The shape's axis aligns with the width and height of the bounding rectangle.
- **Rotate** : Set the rotation angle of the drawing.
- **Hide** : Sets whether the drawing is hidden or not on the canvas.
- **Locked** : Sets whether the drawing is locked on the canvas or not.

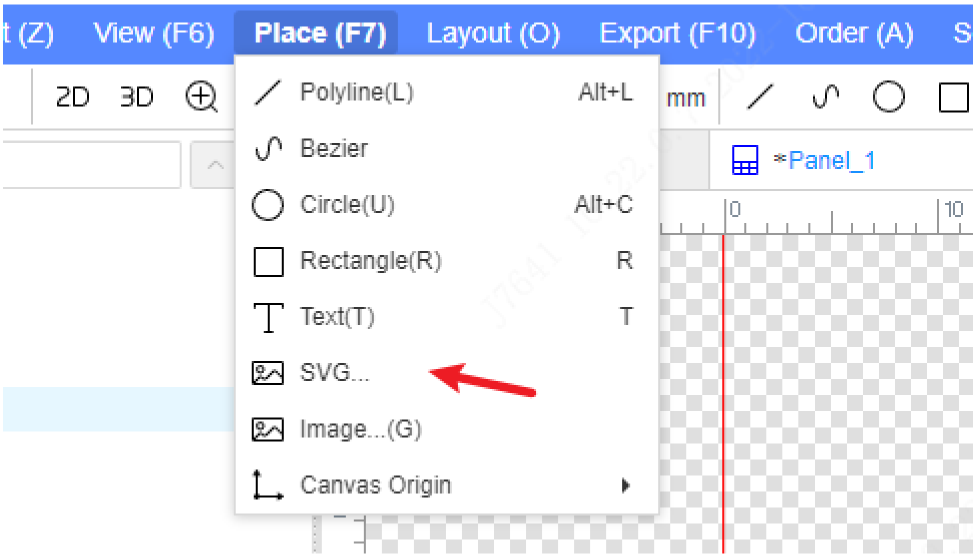
Style

- **Fill Color** : Set the fill color of the graphic, check whether to apply the fill color to the graphic, if not checked, there will be no fill
- **Stroke Color** : Set the stroke color of the graphic, check whether to apply the stroke color to the graphic, if not checked, there will be no stroke
- **Line Width** : The width setting of the drawing stroke.

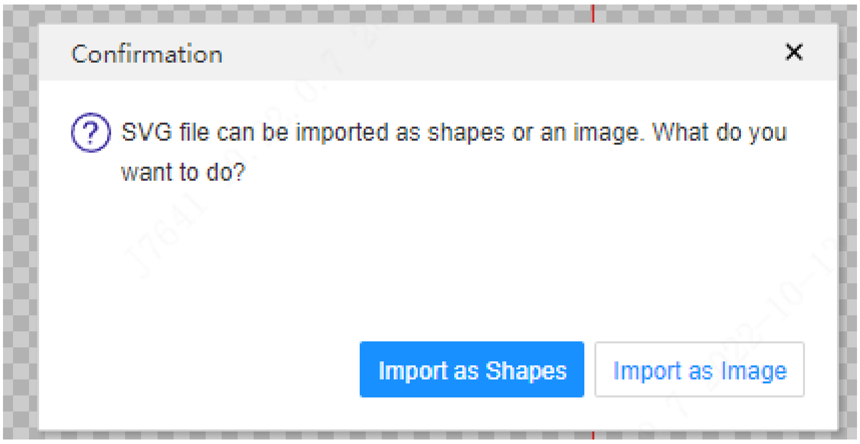
Place SVG

EasyEDA Pro supports importing SVG as a path for editing. If you need to import SVG as a picture, please select the function of placing pictures.

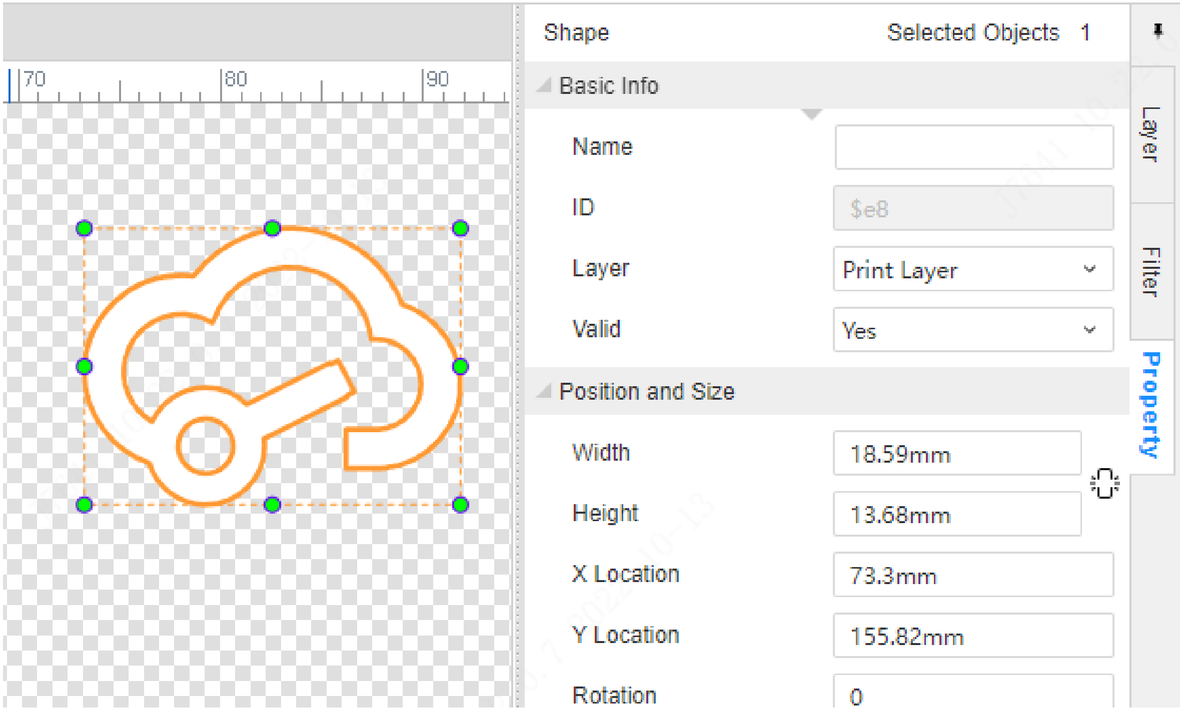
Entry: Top Menu - Place - SVG



When importing, a pop-up window will prompt you to select the import type:



After selecting an imported shape, its shape can be edited directly.



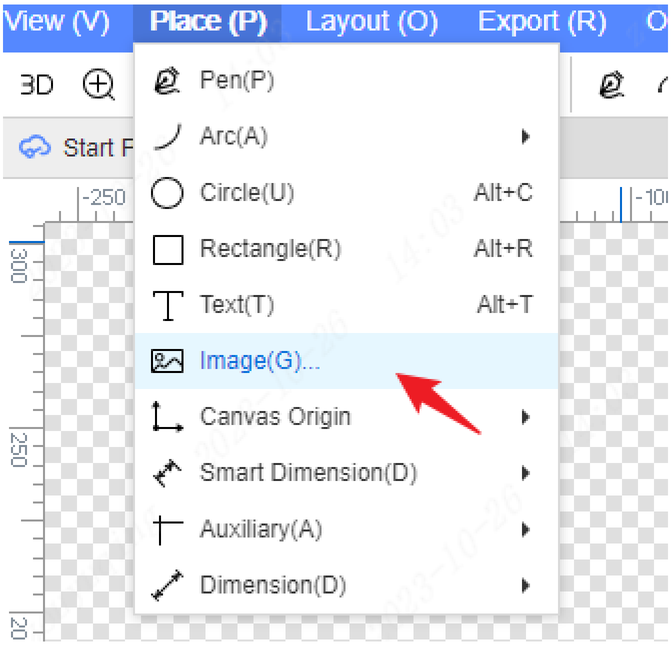
Images

Place Image

Place image operation process

- Top Menu Bar - Place - Picture

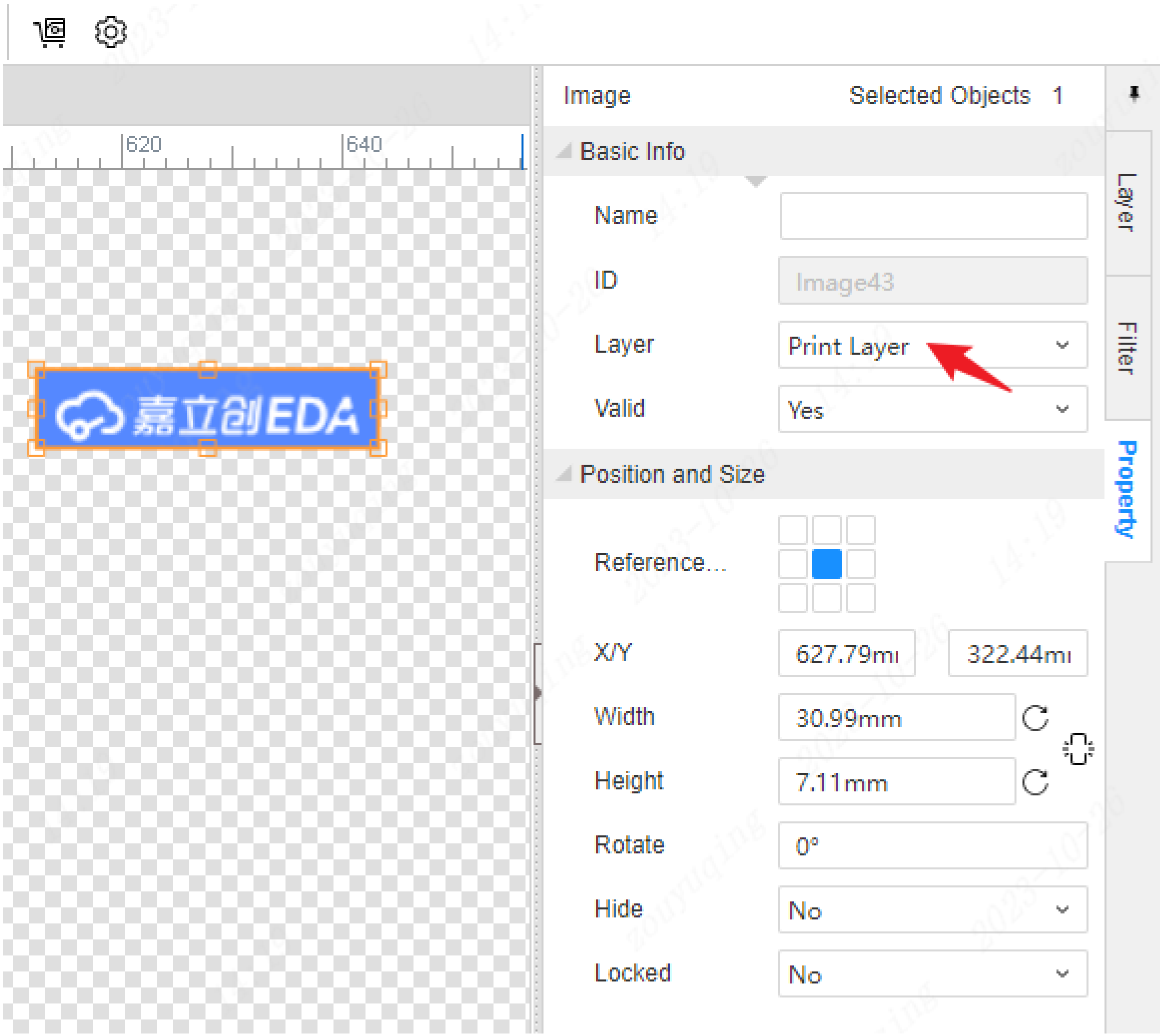
Support inserting images in panel files. Convenient for inserting various logos, etc.



Support importing multiple formats. Please limit the import size of a single file to around 1MB. Please limit the total size of multiple images to 10MB, otherwise the file may not be stored or take a long time to open and parse.

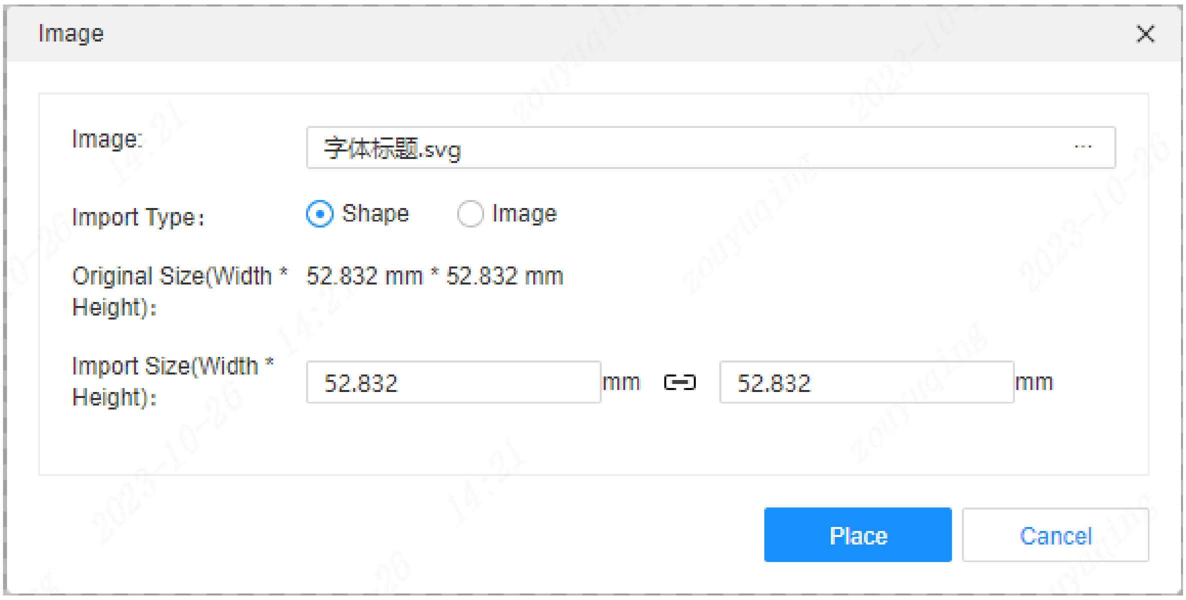
(*.jfif;*.pjpeg;*.jpeg;*.pjp;*.jpg;*.png;*.gif;*.webp;*.svg)

The image can only be set on the print layer, not on other layers. If you need to adjust the transparency of the image, you can place a rectangular transparent area inside the image to achieve the effect of image transparency by adjusting the transparency of the rectangle.



Import SVG supports setting size and import type

Operate to import the picture, select the SVG file, and the following pop-up window will pop up.



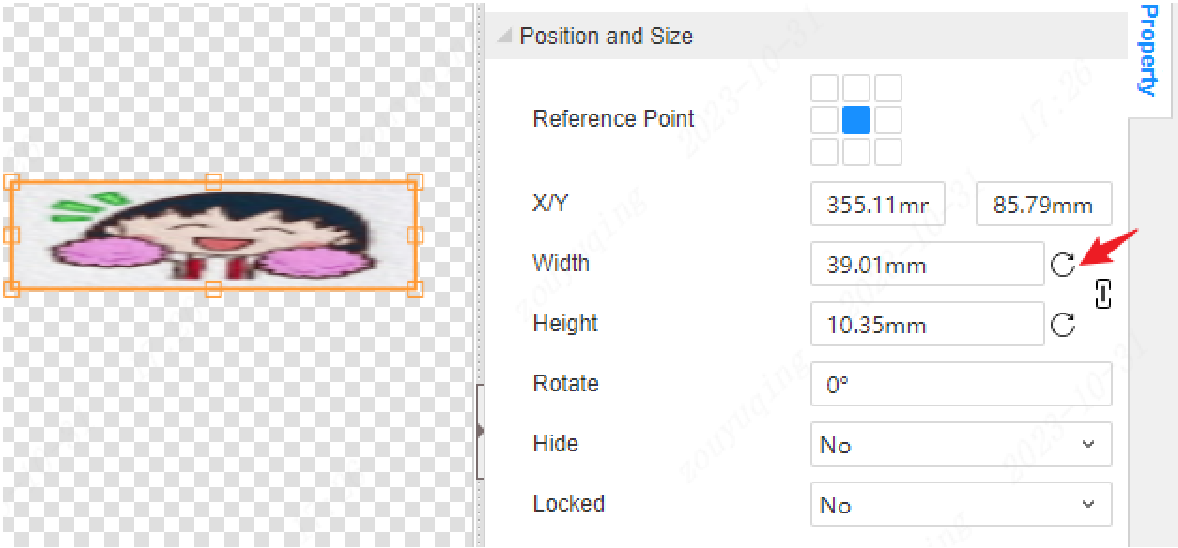
Display the original file name. After clicking, a select file pop-up window will be displayed, allowing you to re-select the file to be imported.
Import type The shape is selected by default and can be switched to an image.
Display the original size of the selected file in mm.
The imported dimensions are the same as the original dimensions by default, in mm. The aspect ratio is kept by default, that is, if the width value is adjusted, the height value will change synchronously according to the original aspect ratio. Click the keep aspect ratio setting icon to

switch to not keep the aspect ratio, that is, if the width value is adjusted, the height value will remain.

Note: When the original size cannot be obtained, the original size line will not be displayed, and the imported size will display the size to be placed according to the original rules.
If the imported file is not an SVG file, there is no import type option line.

Restore the original scale of the image

The "Restore Original Scale" button is provided behind the width and height text box of the picture. After clicking the Restore button, the unclicked row data will remain unchanged and the clicked row data will be recalculated according to the original scale of the picture.



It can quickly restore the image to the original proportion and reduce deformation.

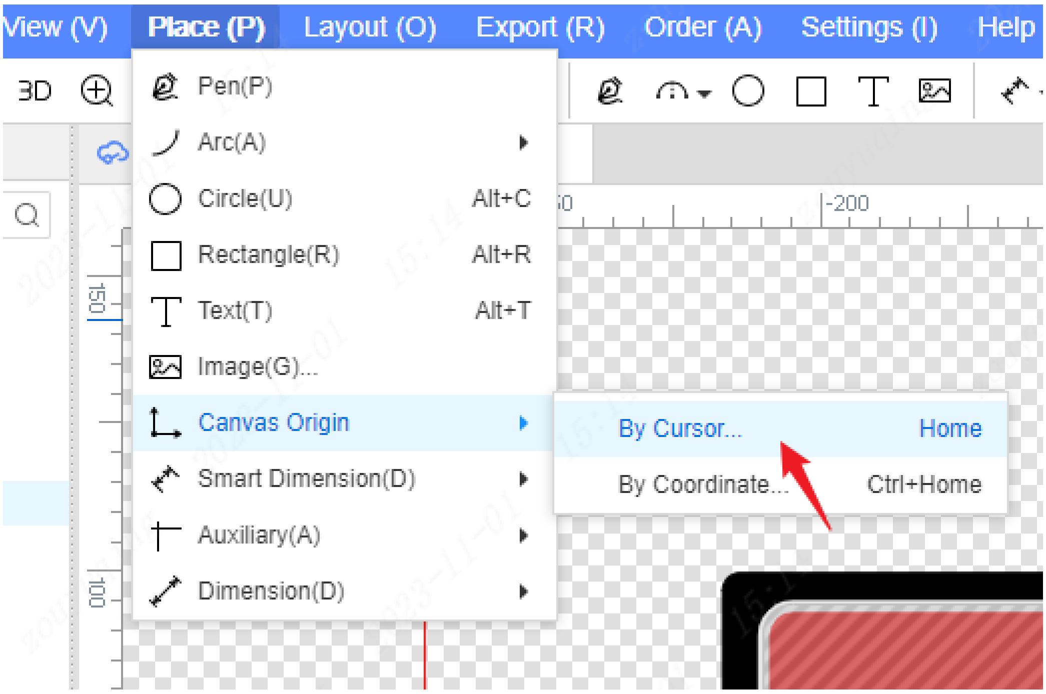
Place Canvas Origin

Canvas Origin

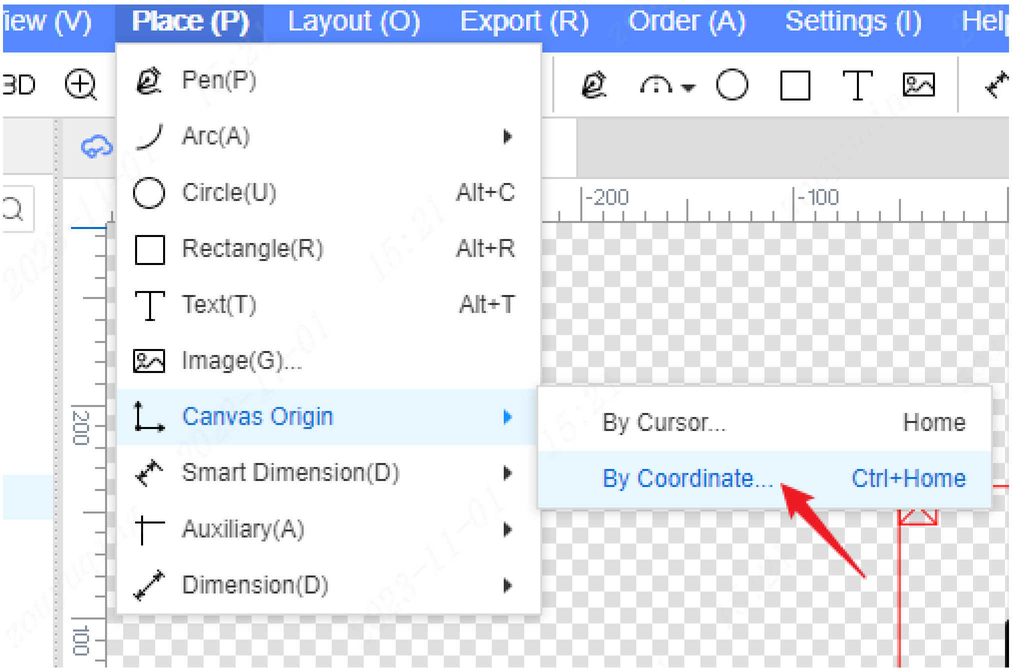
During panel design, the created new panel position is drawn with the lower left corner of the material boundary as the origin. If it does not meet the requirements, the origin of the canvas needs to be modified, and the function of placing the origin of the canvas can be used.

Function entrance

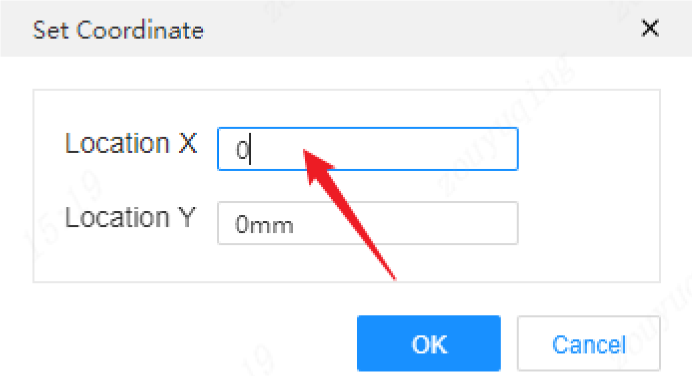
- Top Menu - Place - Canvas Origin - By Cursor...
- Global shortcut: Home



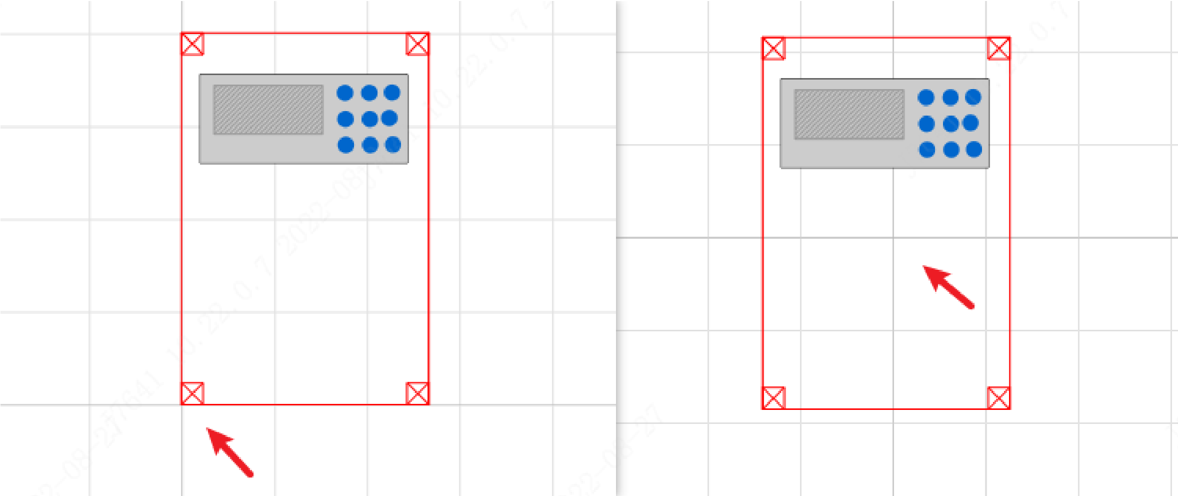
- Top Menu - Place - Canvas Origin - By Coordinate Point...
- Global shortcut： Ctrl + Home



Set the canvas origin from the coordinate point, pop up the Set Coordinates pop-up window, edit the XY coordinate value and click OK to set it as the canvas origin.



The effect after modifying the origin of the canvas.



Smart Dimension

Smart Size

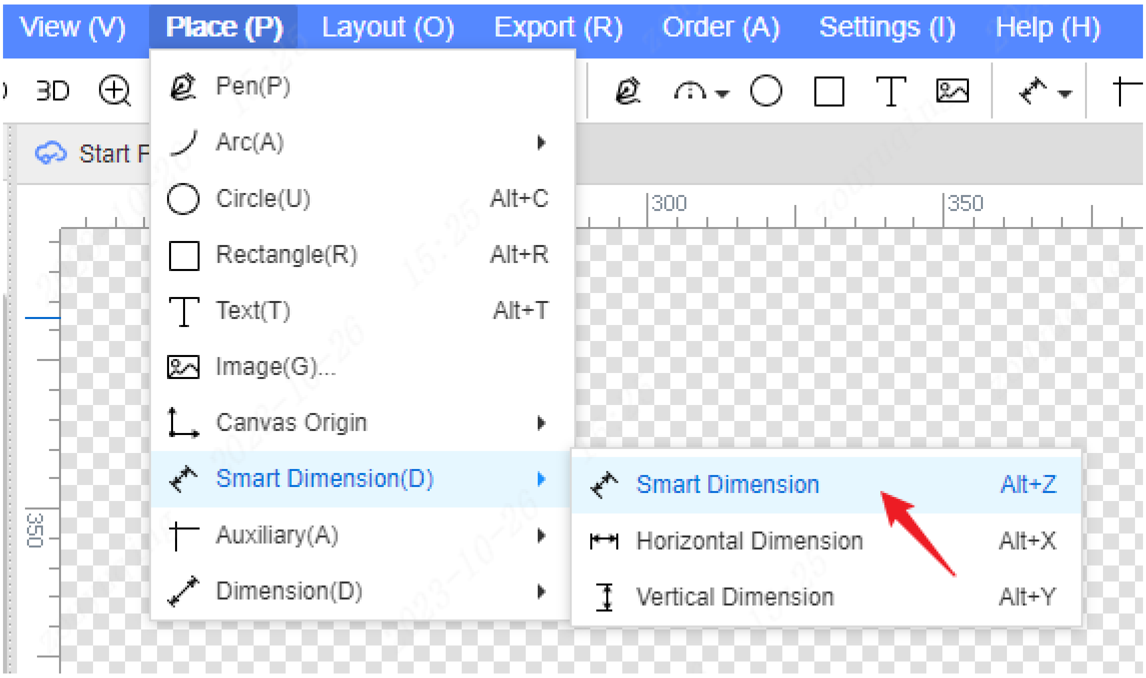
The distance between panel elements can be constrained or annotated by intelligent dimensions, which are temporarily displayed on the canvas as auxiliary elements. The intelligent dimensions are not effective in actual production.
The intelligent dimension of constraint type can modify the constraint value on the canvas, and always maintains the spacing value between two elements. The intelligent dimension of annotation type measures the spacing value between elements in real time, and the annotated elements can move freely, while the corresponding annotation value changes synchronously.

Function entry

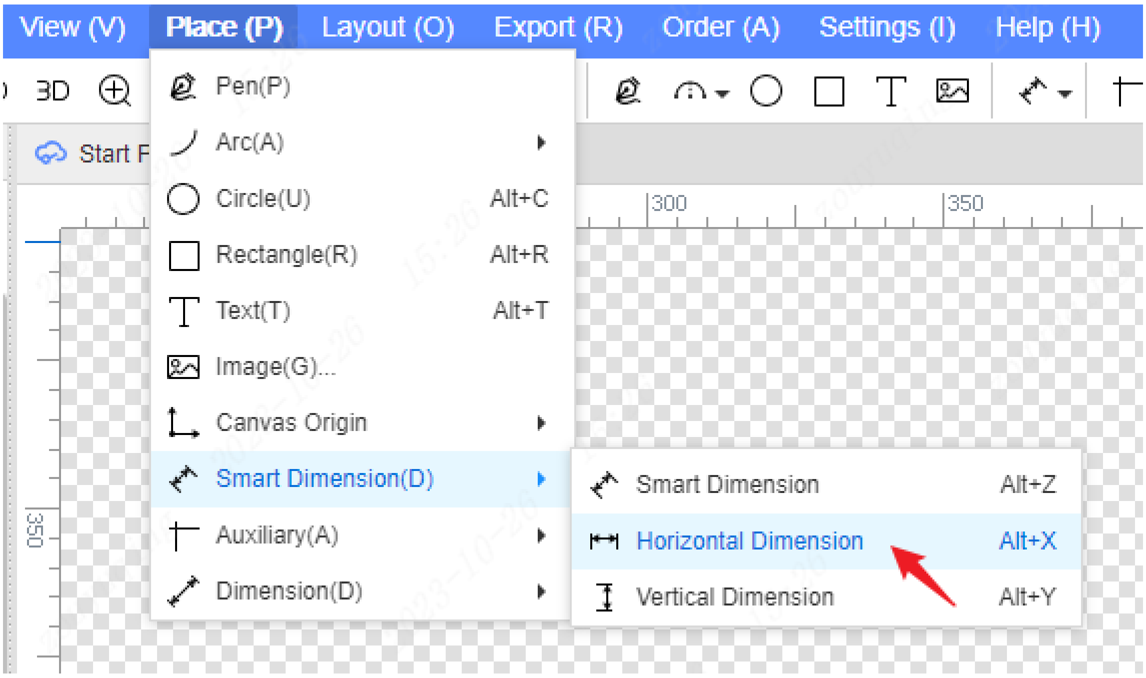
Top menu

Provides 3 modes of drawing intelligent dimensions, intelligent dimensions can draw dimensions at any angle, while horizontal dimensions will only draw intelligent dimensions in the horizontal direction, and similarly vertical dimensions will only draw intelligent dimensions in the vertical direction

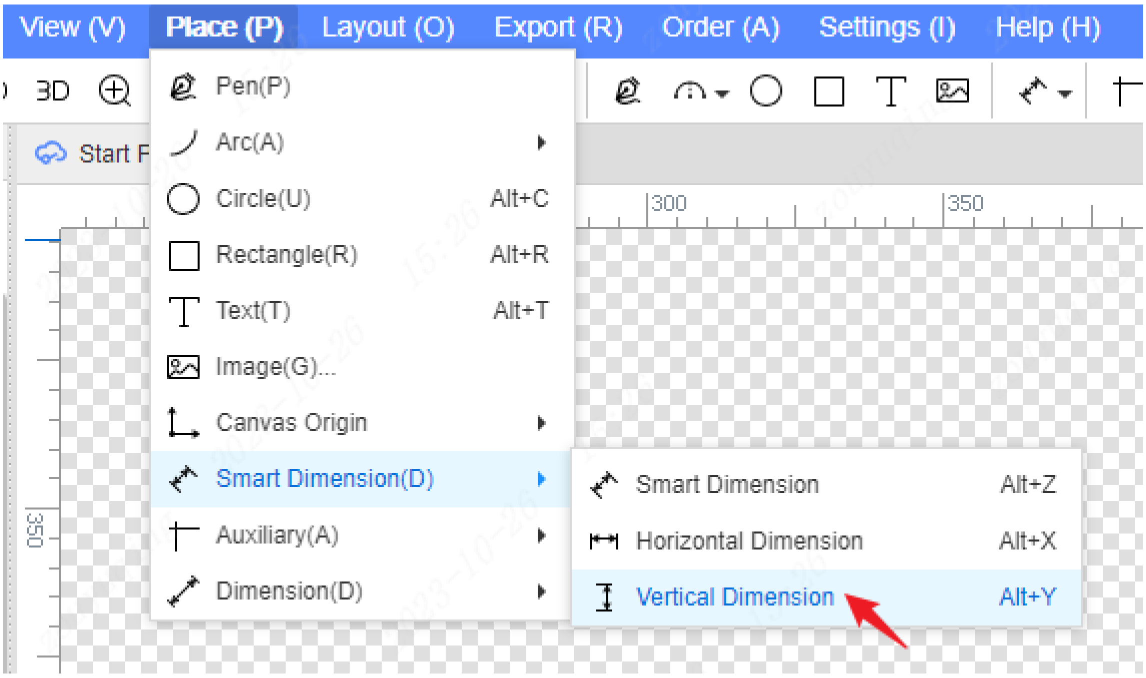
- Top Menu - Placement - Smart Size - Smart Size
- shortcut: `Alt` + `Z`



- Top Menu - Placement - Smart Size - Horizontal Size
- shortcut: Alt + X

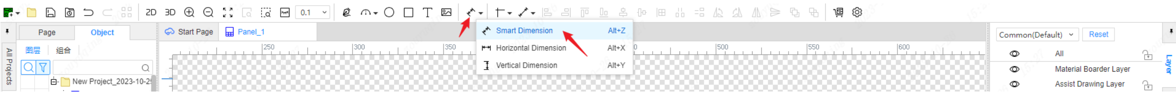


- Top Menu - Placement - Smart Size - Vertical Size
- shortcut: Alt + Y



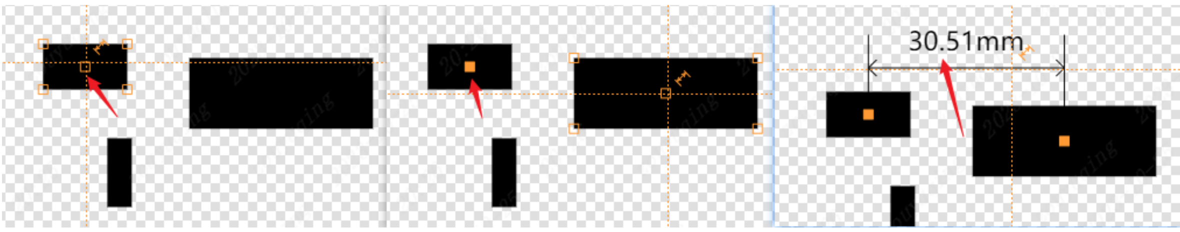
Top toolbar

- Click the smart size icon directly, or expand the drop-down option to switch to select the drawing mode

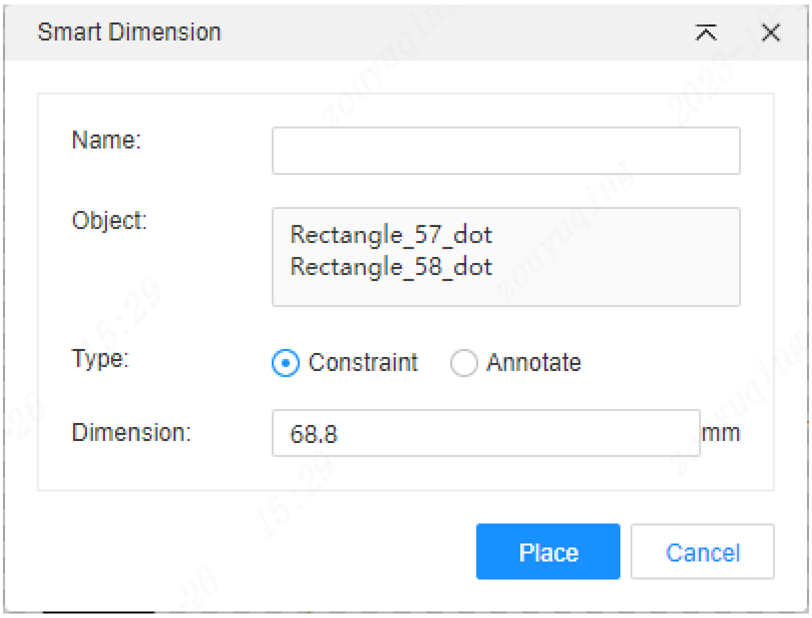


Operation steps

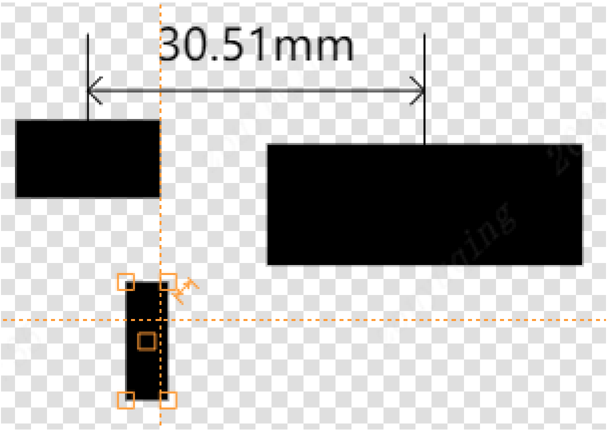
- Enter the intelligent size drawing mode from the function entrance operation, then move to the graphic element to select the measurement point. After completing the measurement point selection of the two graphic elements, the size between the two points will be displayed, and the size direction and position can be adjusted by moving.



- After adjusting the size, position, and direction, click the left mouse button to display the intelligent size pop-up window.
  In the pop-up window, you can edit the name of the intelligent size, display the information of the intelligent size associated object, select the intelligent size type as constraint or annotation, the size value of the constraint type can be edited, and the size value of the annotation type cannot be edited.



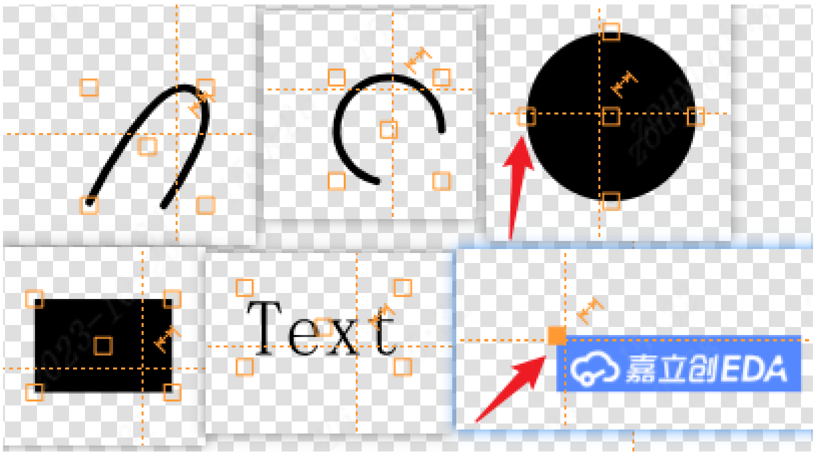
- After selecting the intelligent size type and confirming the size value, click the Place button in the pop-up window to complete the drawing of the intelligent size and enter the state of the next intelligent size to be drawn.



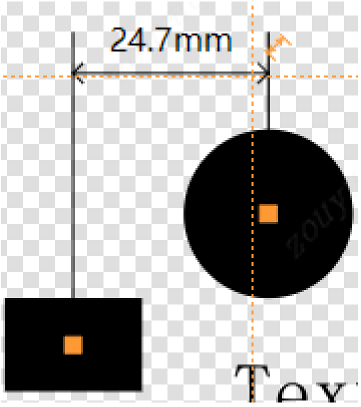
Rules and Agreements

Optional measuring point

- Five optional measurement points are provided for the types of primitives such as shape, ellipse arc, ellipse, rectangle, text, and image. The optional measurement points are hollow squares when selected, and solid square fills when selected.

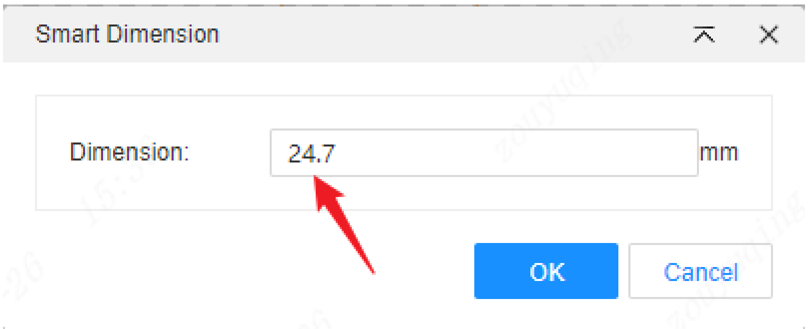


- Currently, only distance constraints or annotations between elements are supported. Only one measurement point can be selected on one element. After selecting one measurement point on the element, the remaining four will be hidden. After selecting one measurement point on each element, the intelligent size will display the actual measured distance between two points.

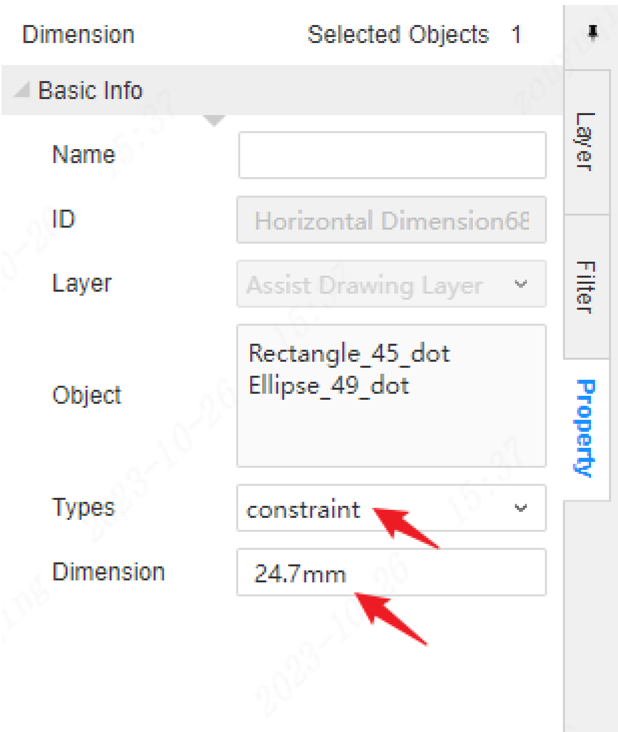


Constraints - Smart Dimensions

The intelligent size of the constraint type can be modified by double-clicking the value on the canvas, and the pop-up window for modifying the size value will pop up. After modifying the value, click the OK button in the pop-up window to complete the constraint value modification.



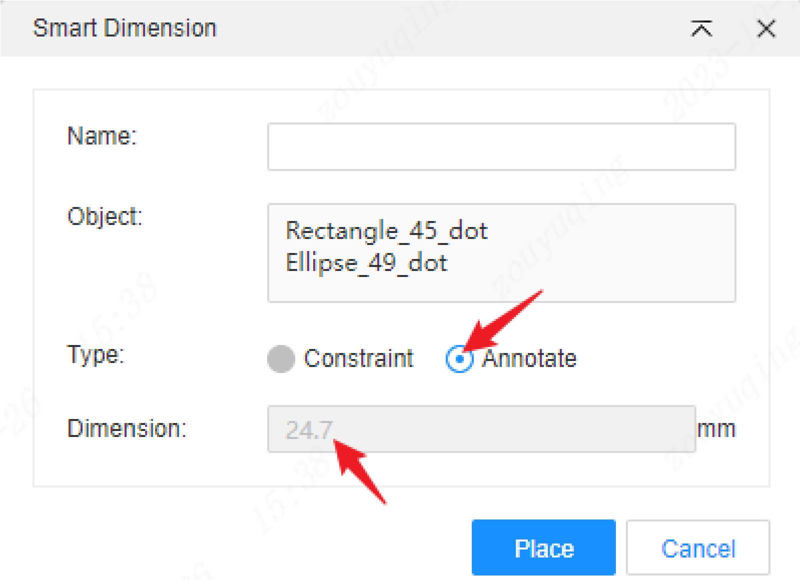
- Select Smart Size, expand the property panel on the right, and you can also adjust the size of the constraint value.

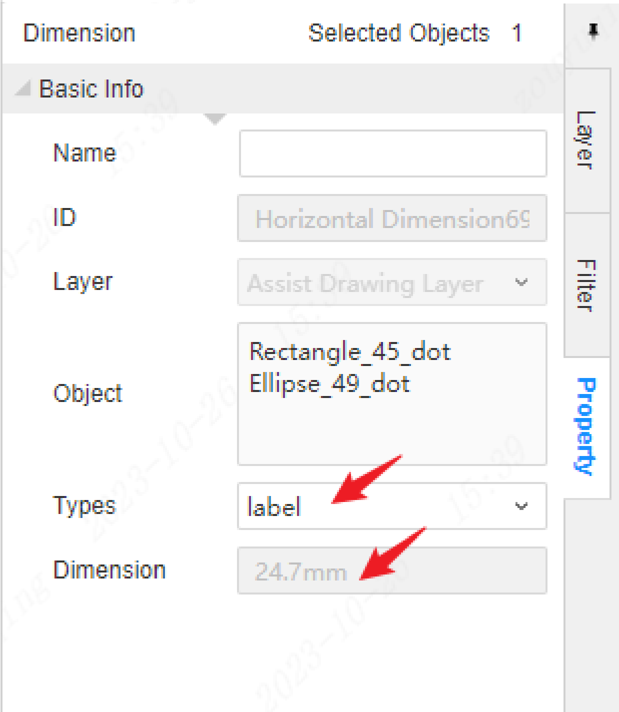


- The spacing value between the two constrained points is fixed. To satisfy the constraint relationship, moving one of the constraint objects will also move to the other constraint object through the constraint intelligent size.
! [Figure 9] (../../_images/place-smart-dimension _20231025_204905 png)

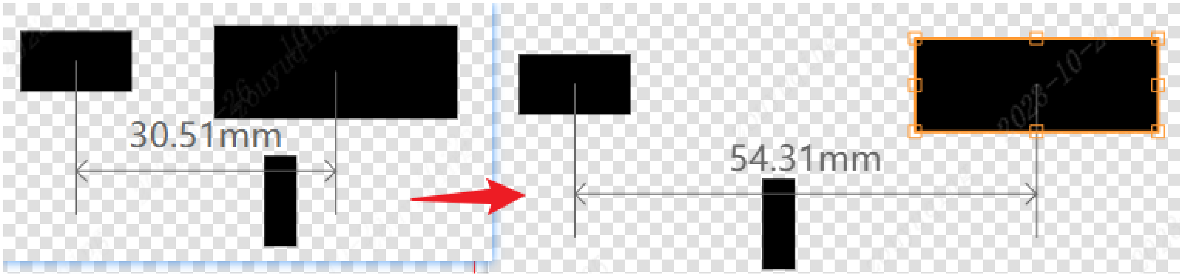
Annotation - Smart Dimensions

- The size value of the intelligent size of the annotation type is always measured in real time and only displayed as a display. It is grey out and cannot be edited in the initial creation intelligent size pop-up window and the property panel on the right.



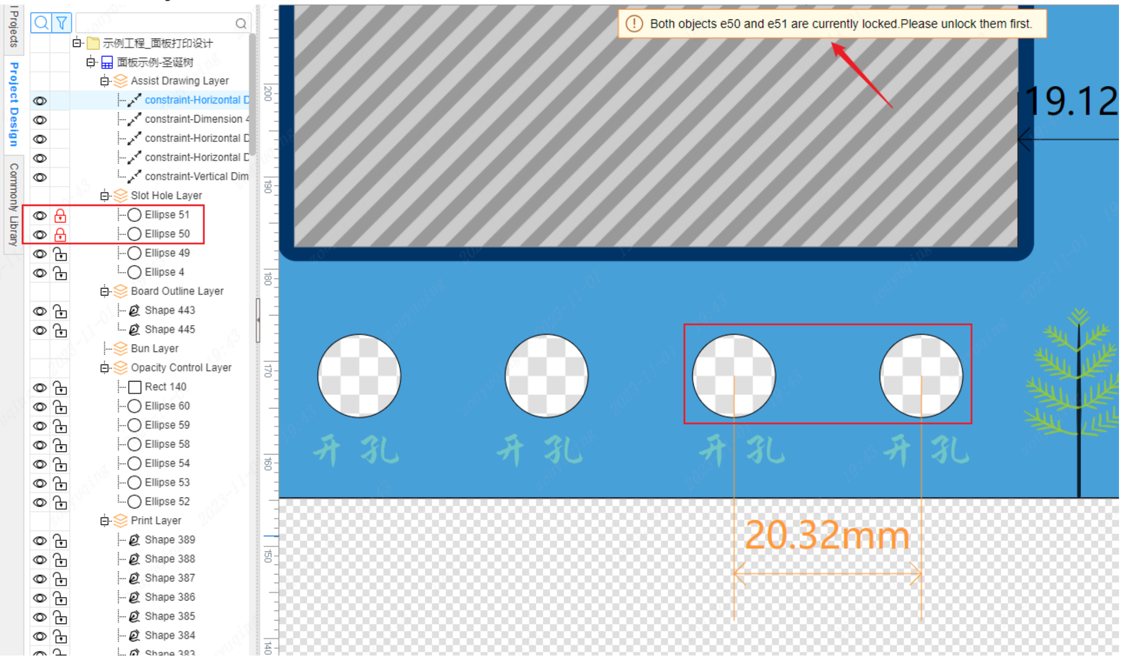


- The spacing value between the two marked points is adjustable. Moving one of the marked objects will keep the coordinate value of the other marked object, and the intelligent size value of the marked object will be updated in real time.

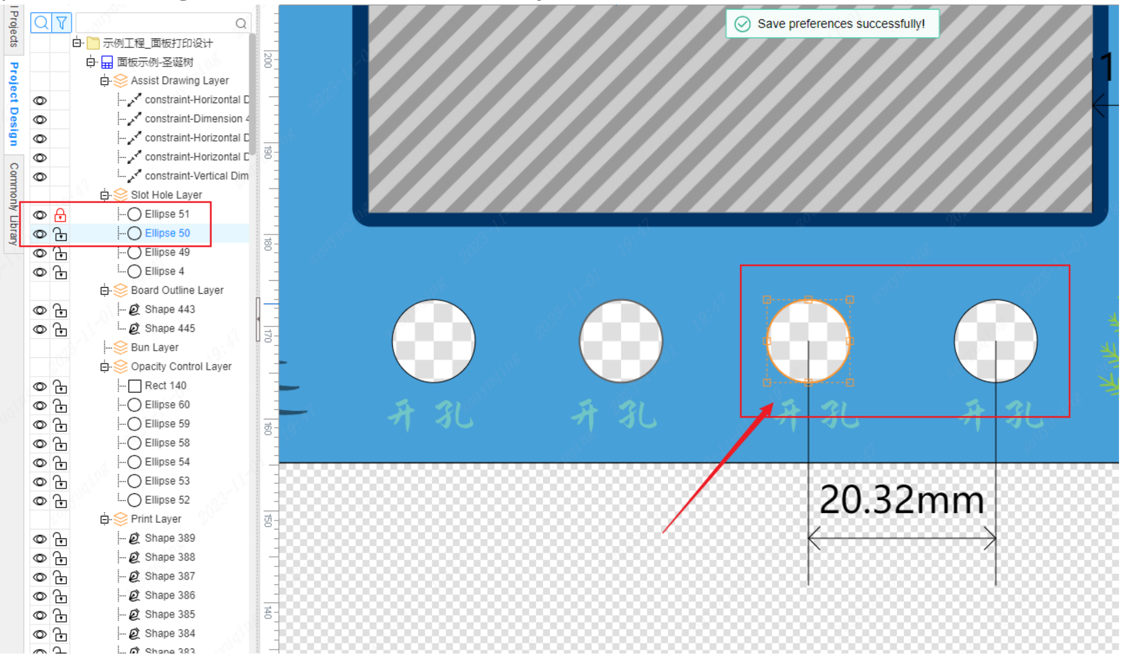


Constraint associated object locking

- When both associated objects of the constraint intelligent size are locked, double-click the constraint value on the canvas, and a short prompt will appear that both constraint objects are locked, please unlock them first. That is, the constraint condition that both constraint associated objects are locked cannot be modified.

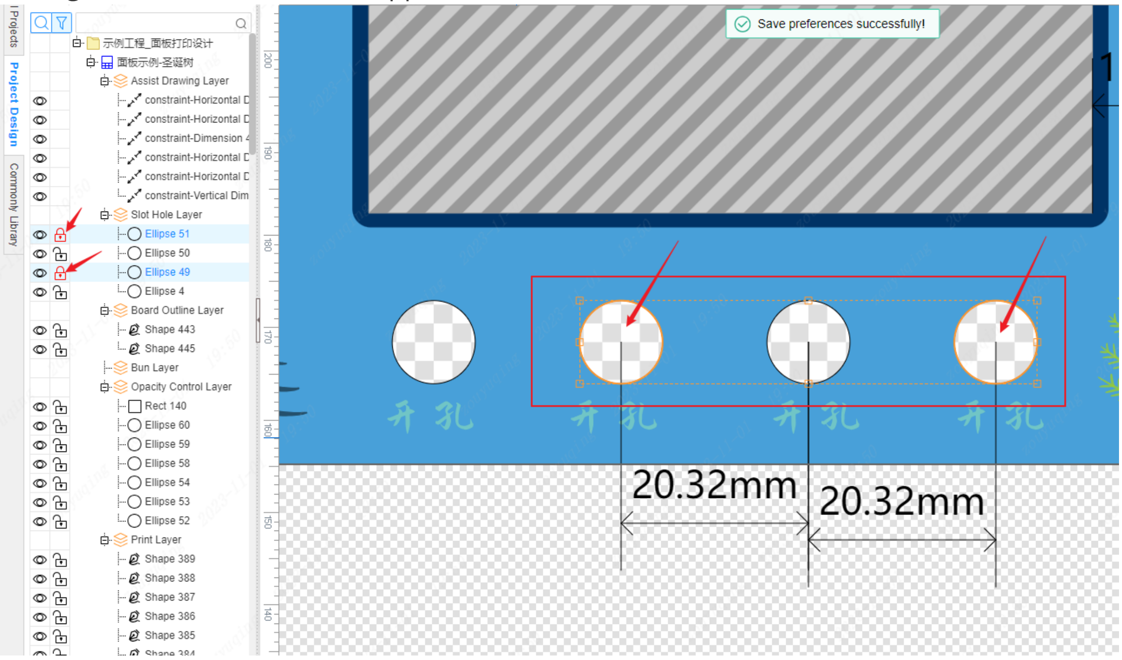


- Only one of the associated objects of the constrained intelligent size is locked. Double-clicking the constraint value on the canvas or modifying the constraint value in the property panel on the right will move the unlocked object to meet the new constraint condition.

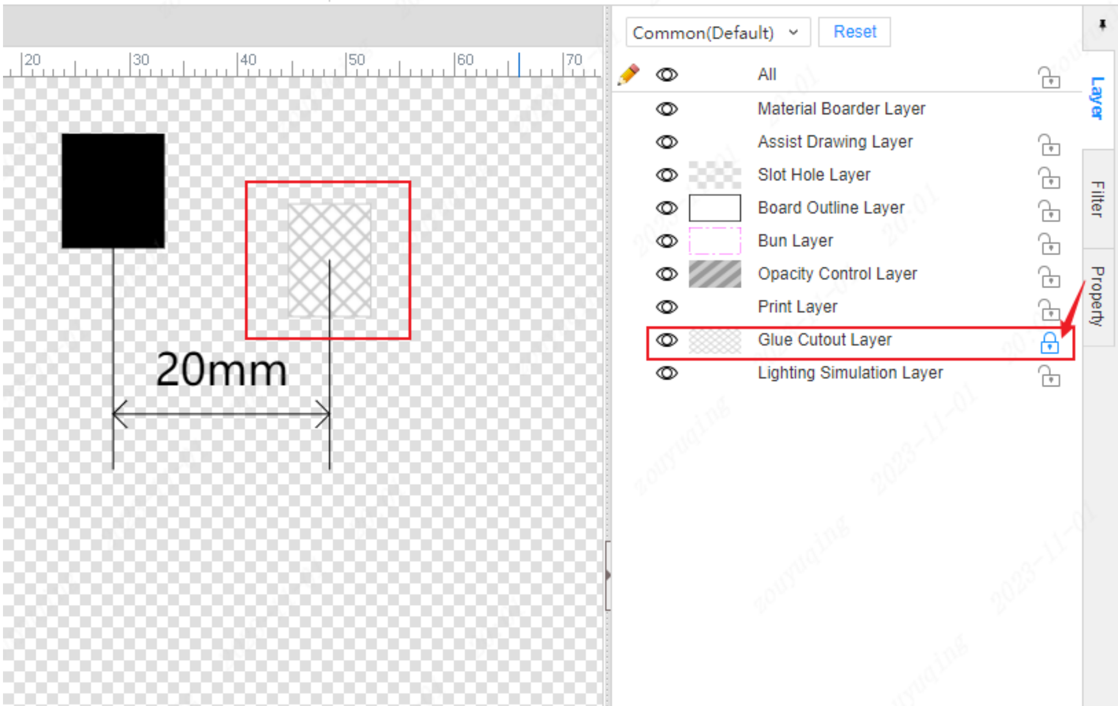


- There are constraints on size and locked objects between multiple objects, and the same applies to the above rules.

  For example, in the scenario shown in the figure below, the constraint values of the two intelligent dimensions do not support modification.

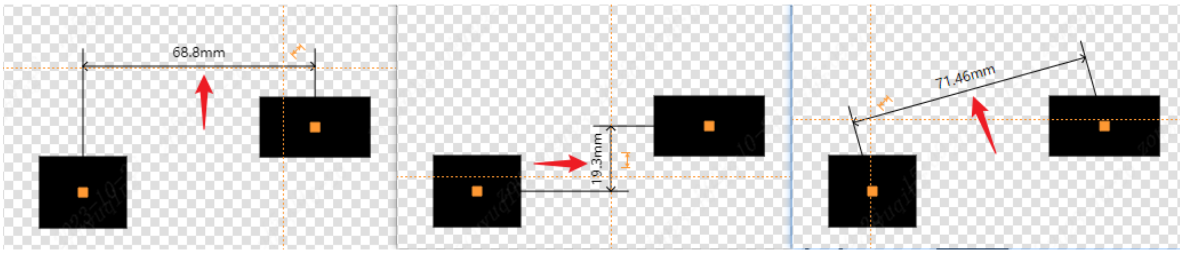


- The spacing between two pairs of images in the combination cannot be adjusted by constraint size, only the combination can be kept moving as a whole. Double-click the constraint value of the constraint associated object in the same combination, and the canvas short prompt will indicate that this change will result in over-definition or no solution, and modification is not supported.
- The right layer setting is not selectable, and the object can be moved to the non-selectable layer by constraining the size.

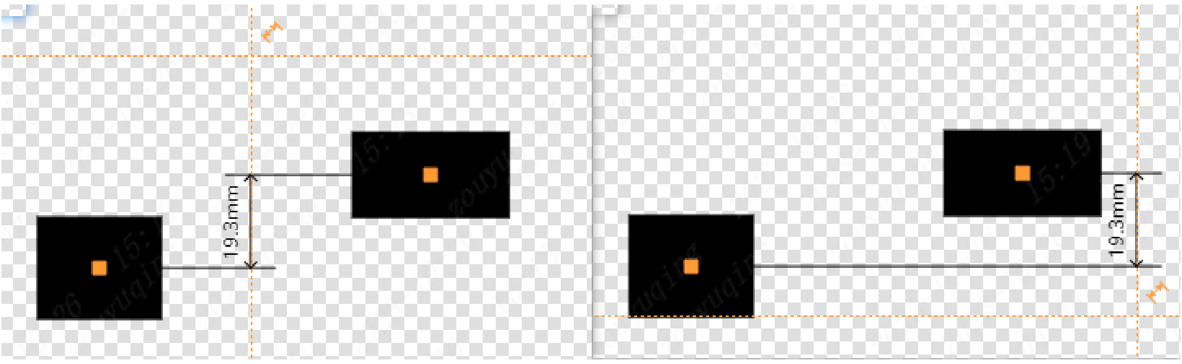


Size type

- There are three types of intelligent dimensions: horizontal dimension, vertical dimension, and dimension. Horizontal dimension measures the spacing in the horizontal direction, vertical dimension measures the spacing in the vertical direction, and dimension measures the spacing in any other angle direction except the horizontal and vertical directions.

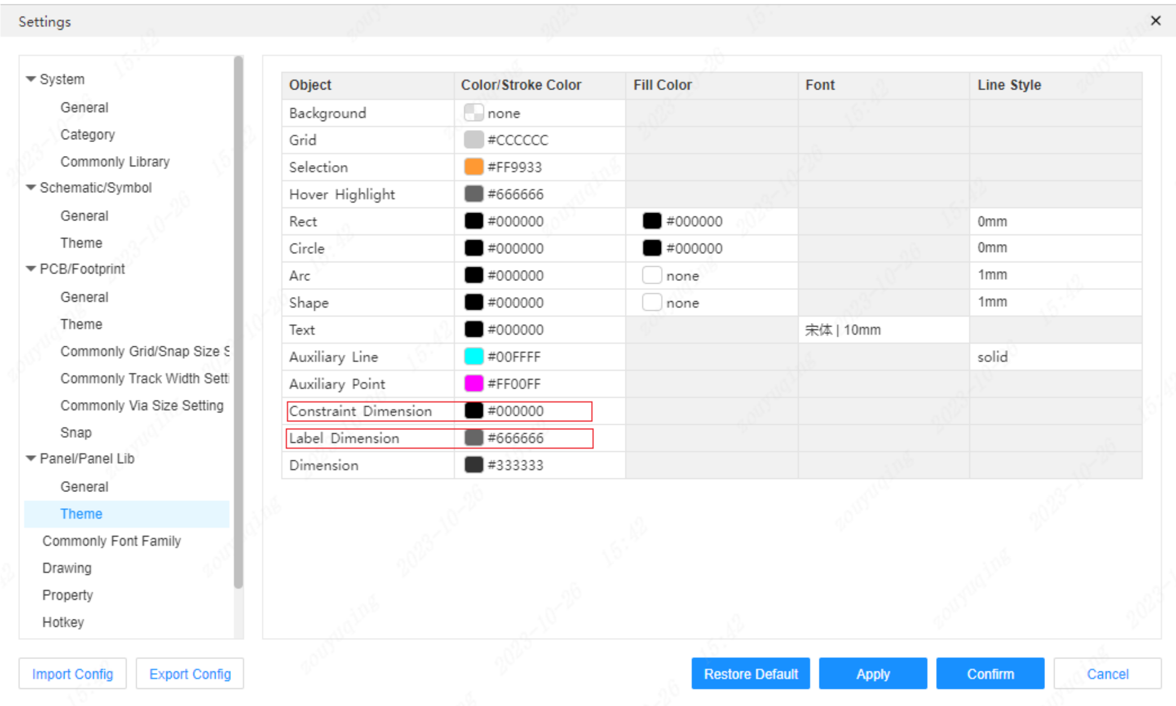


- The size type can be locked and unlocked by the space bar .
  When there are three directions of spacing between the two measured elements, it defaults to the unlocked state during the measurement process through the intelligent size mode. Pressing the space bar once will lock it to the current size type, and any subsequent movement will not switch the measurement direction. Pressing the space bar again will unlock it.



Theme settings

- Settings - Panel/Panel Library - Theme - Constraint Size
- Settings - Panel/Panel Library - Theme - Dimension



The default constraint size is different from the annotated size theme color and supports custom adjustment.

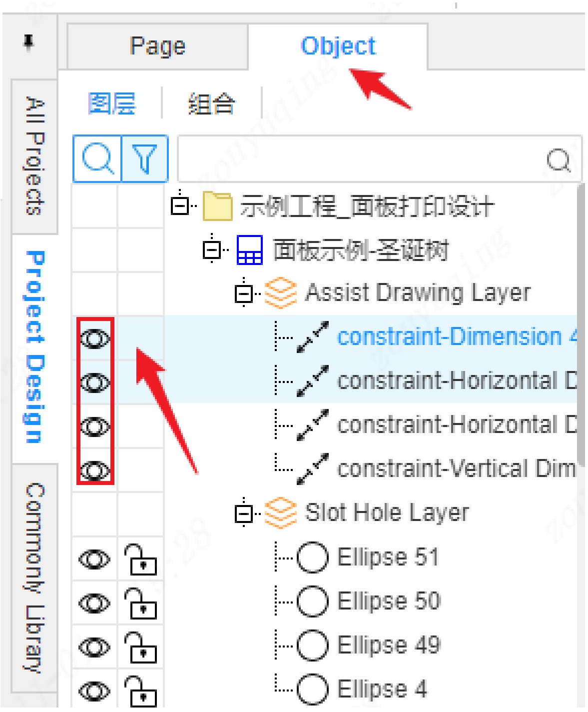
Right-click menu

The right-click menu of the intelligent size has adaptive selection, deletion, hiding, and attribute menu items, that is, selecting the intelligent size on the canvas can perform these operations.

Considering the intelligent size as a special auxiliary element, it does not support operations such as cutting, copying, and locking.

Left Panel - Object

In the left object panel, only the hidden display setting is provided, and no lock unlock setting is provided like other elements.



Auxiliary

support

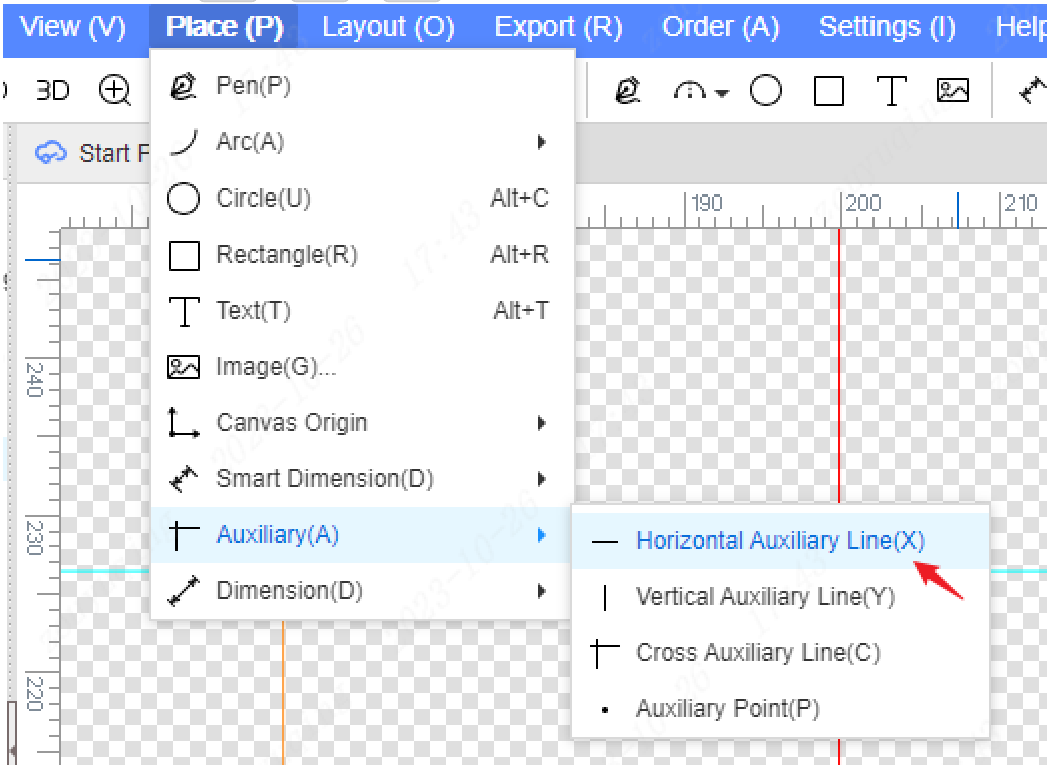
Auxiliary lines and auxiliary points can be used to assist other ordinary elements in alignment and adsorption , and only exist temporarily on the canvas. The elements of the auxiliary drawing layer in actual production are not effective.

Function entry

Top menu

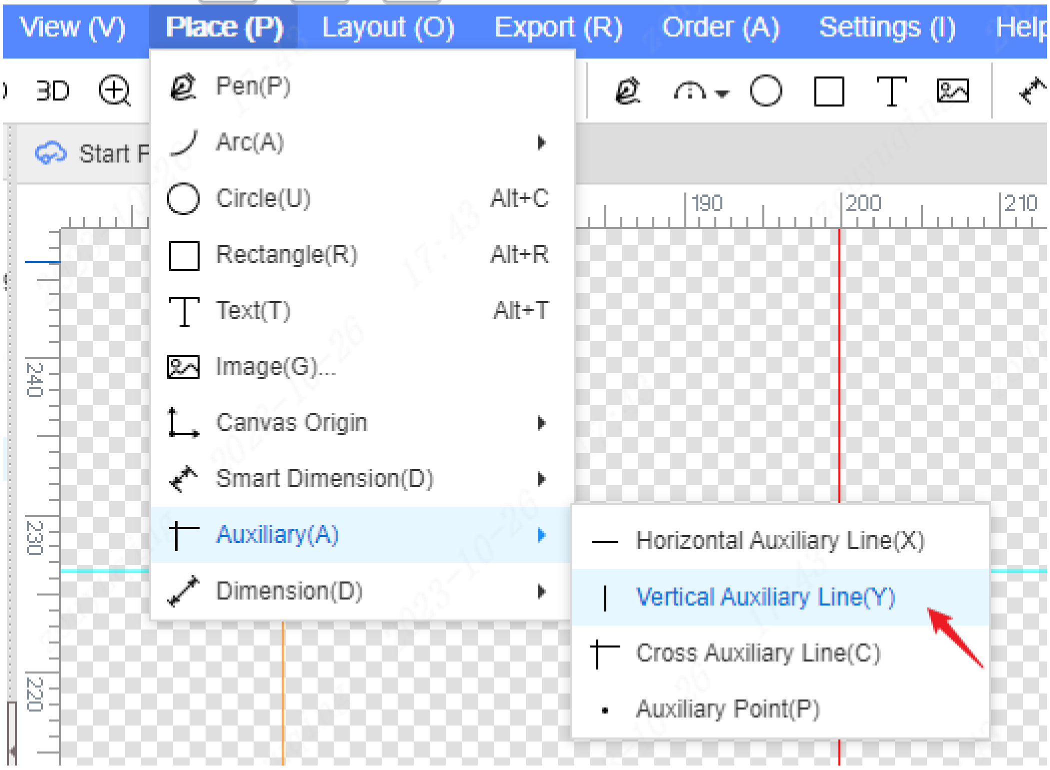
The drawing of auxiliary lines provides three menu options, among which the cross auxiliary line is to draw one horizontal auxiliary line and one vertical auxiliary line at the same time, which are independent of each other after being placed on the canvas.

- Top Menu - Place - Auxiliary - Horizontal Auxiliary Line
- menu shortcut: P + A + X

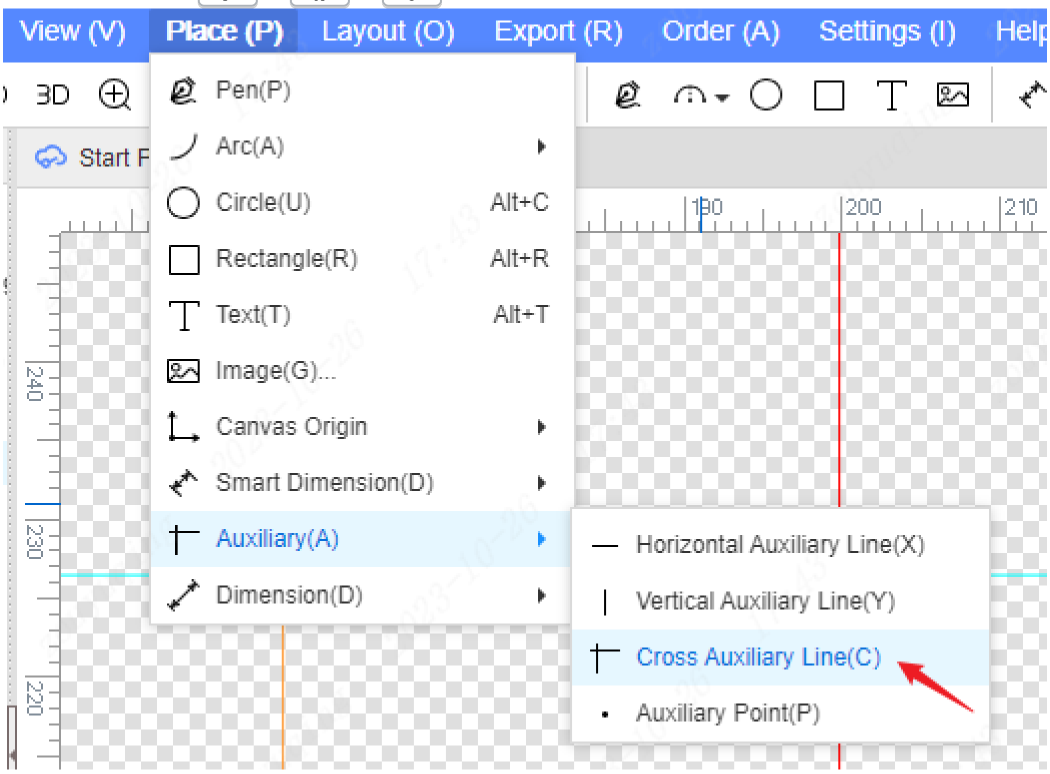


- Top Menu - Place - Auxiliary - Vertical Auxiliary Line

- menu shortcut: P + A + Y

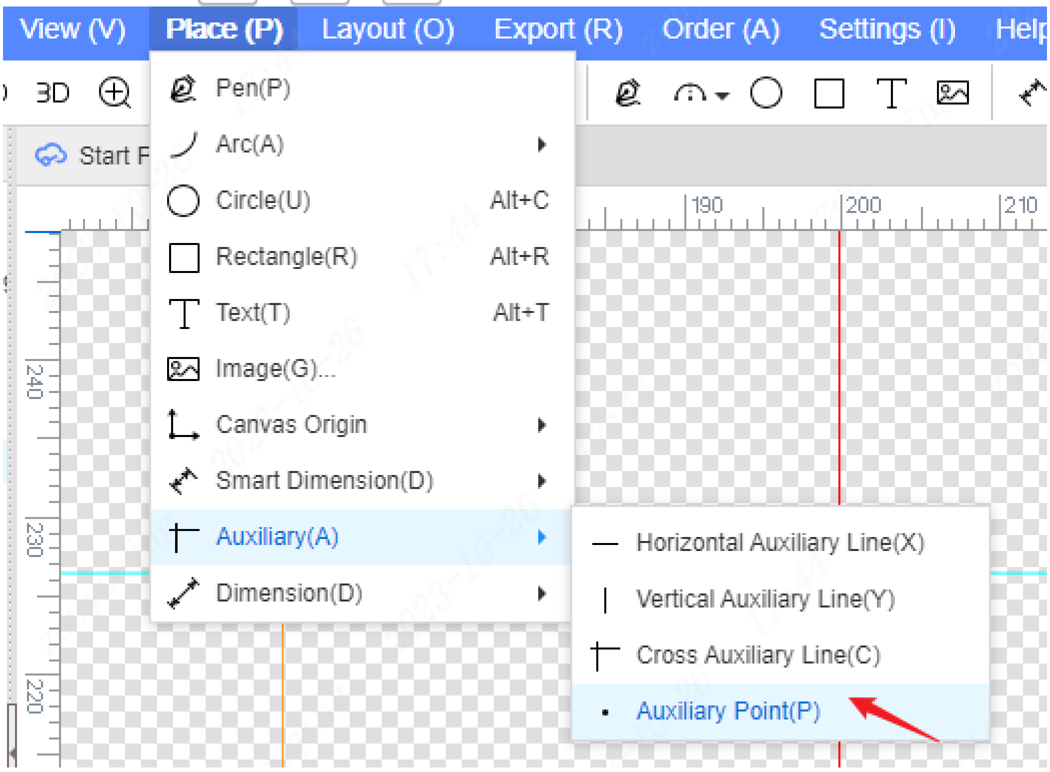


- Top Menu - Placement - Auxiliary - Cross Auxiliary Line
- menu shortcut: P + A + C



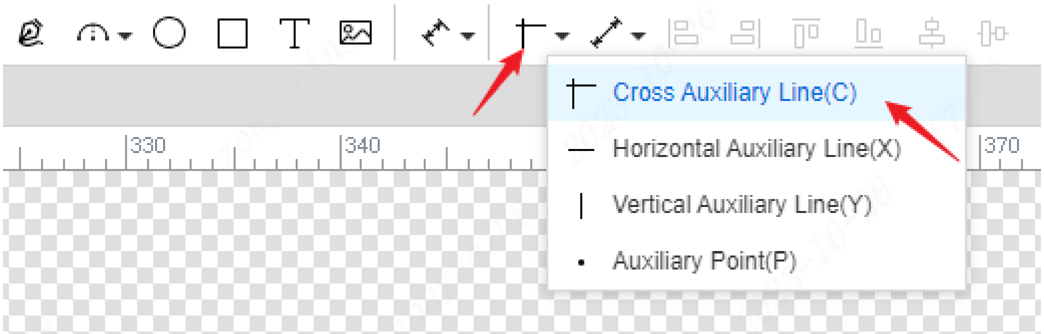
- Top Menu-Place-Auxiliary-Auxiliary Point

- menu shortcut: P + A + P



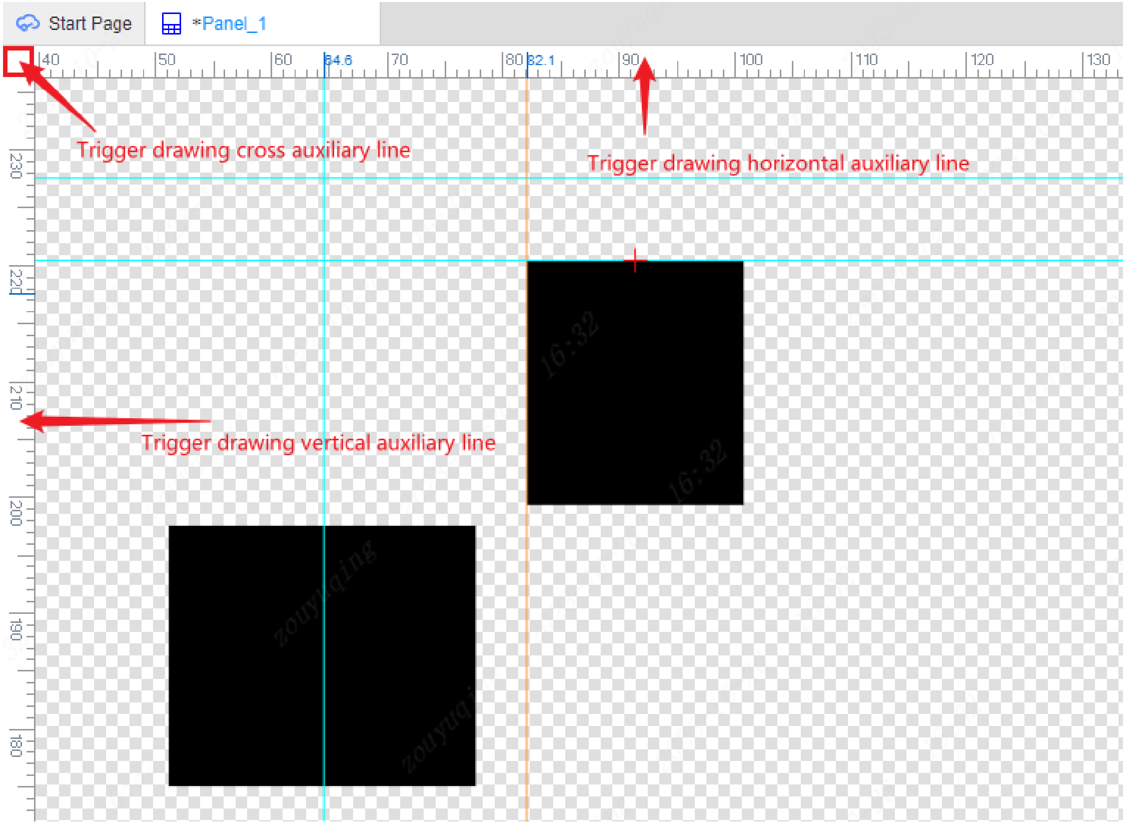
Top toolbar

- Directly click on the auxiliary icon in the top toolbar. The default is to draw a cross auxiliary line. You can also expand the drop-down options to switch to place other types of auxiliary elements.



Ruler

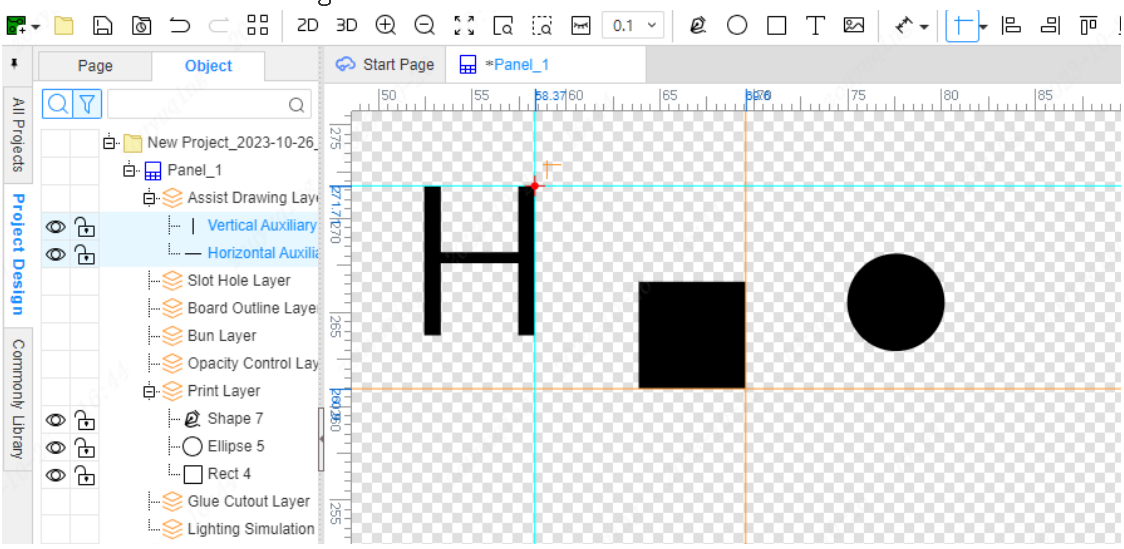
- The ruler area on the upper and left edges of the panel canvas editing area can also quickly trigger the drawing of auxiliary lines. After clicking the left mouse button, the cursor will follow and display the corresponding type of auxiliary line in the pending position.



- You can also hold down the left button in the ruler area and drag it out. After releasing the left button, display the corresponding type of auxiliary line, which is in the waiting position.

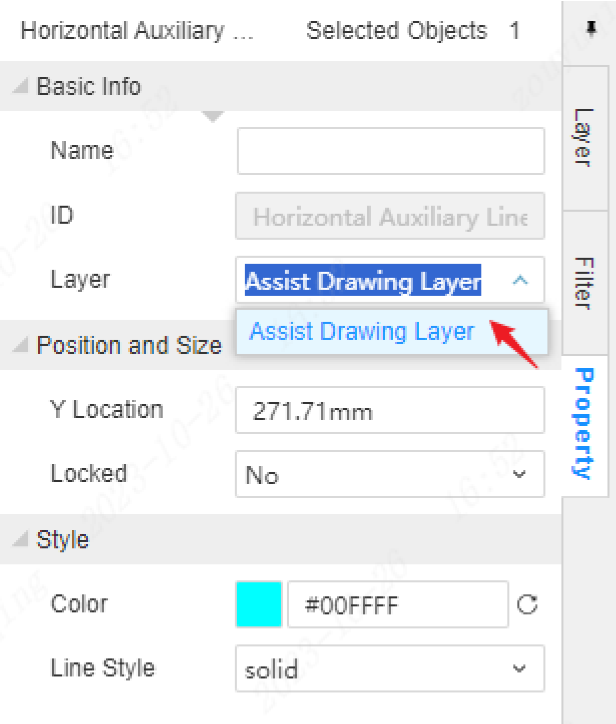
Operation steps

- Enter the drawing mode of auxiliary line or auxiliary point from the function entrance operation. The corresponding type of icon will be displayed in the upper right corner of the cursor, and the corresponding type of auxiliary element will also be adsorbed on the cursor. Move the left mouse button on the canvas to complete the placement, and the right mouse button will exit the drawing state.

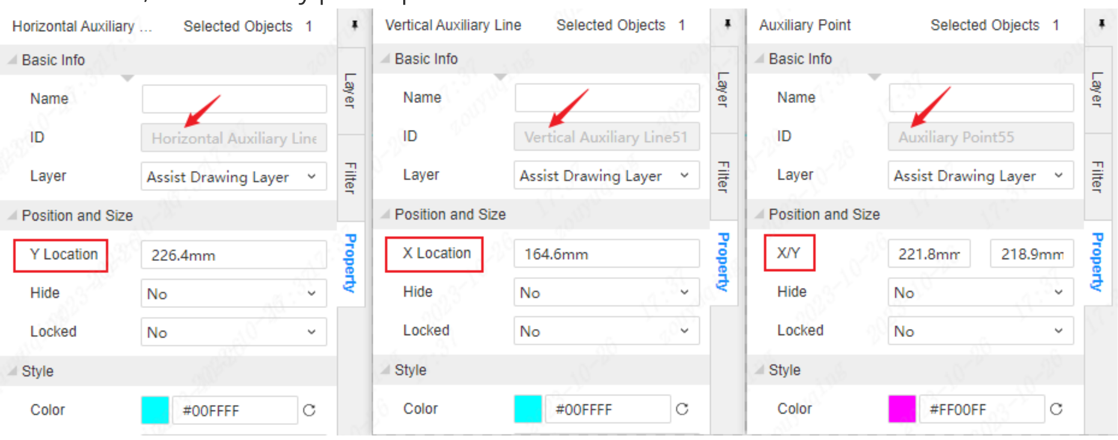


Rules

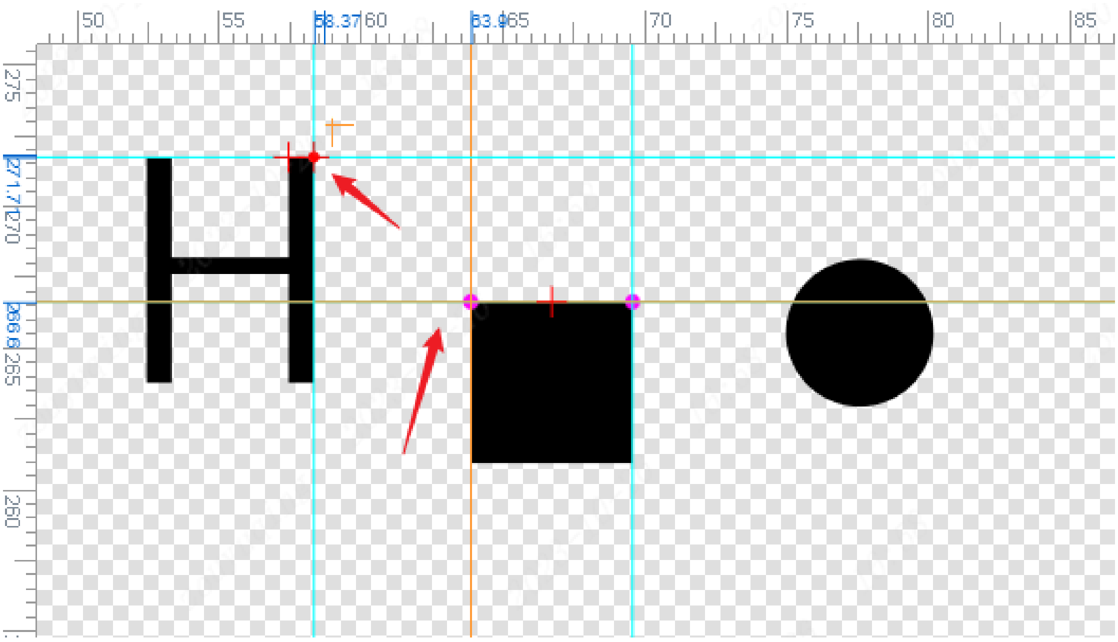
- After the auxiliary lines and points are drawn, they are saved in the auxiliary drawing layer and cannot be switched to other layers.



- Horizontal auxiliary lines only provide Y coordinates, vertical auxiliary lines only provide X coordinates, and auxiliary points provide XY coordinates

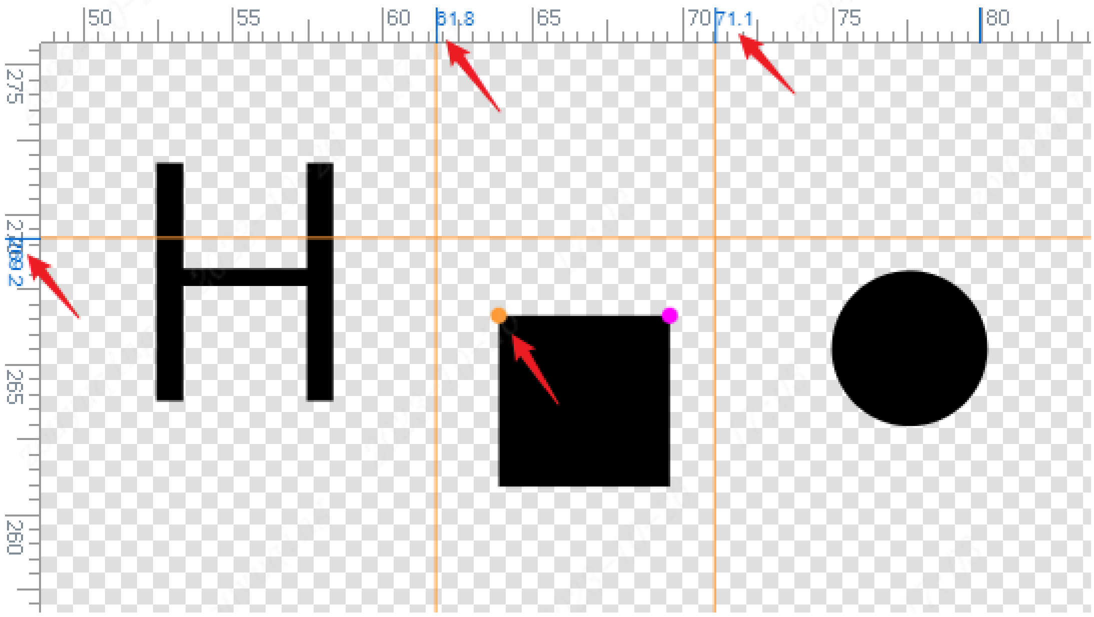


- Auxiliary lines and auxiliary points are placed continuously by default. The selection of the previous placed auxiliary object will be kept and the next auxiliary object will be placed. Right-clicking the mouse or pressing Esc can exit the auxiliary drawing mode.

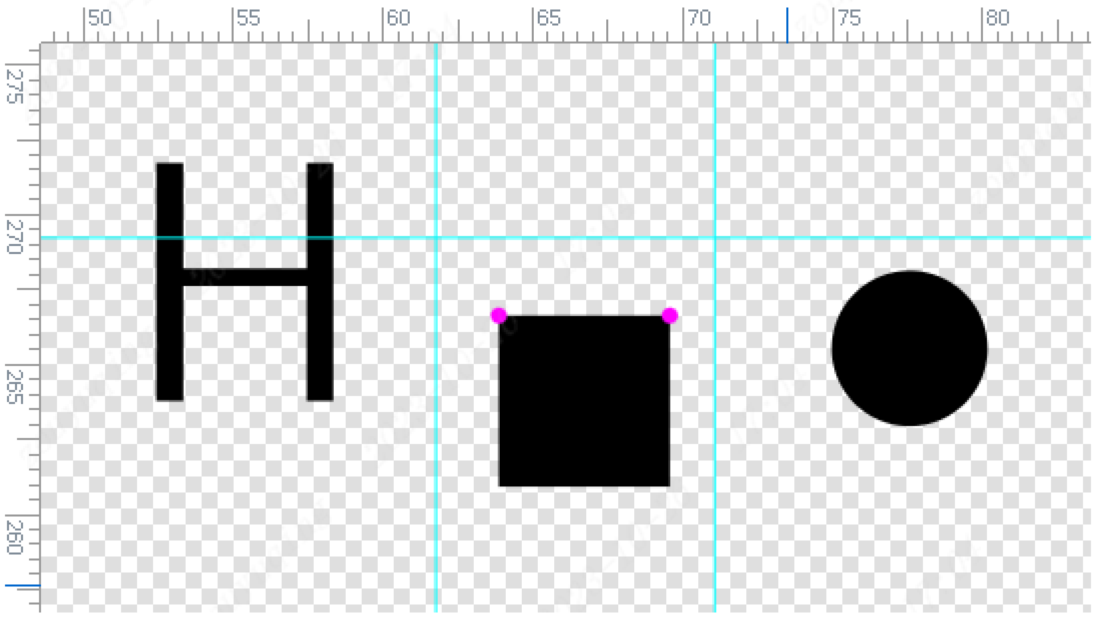


- During the process of auxiliary primitive movement to be placed, the corresponding position in the ruler area will display its corresponding coordinate value.

Note: When the auxiliary point is selected, the corresponding coordinate value will not be displayed at the corresponding position in the ruler area.

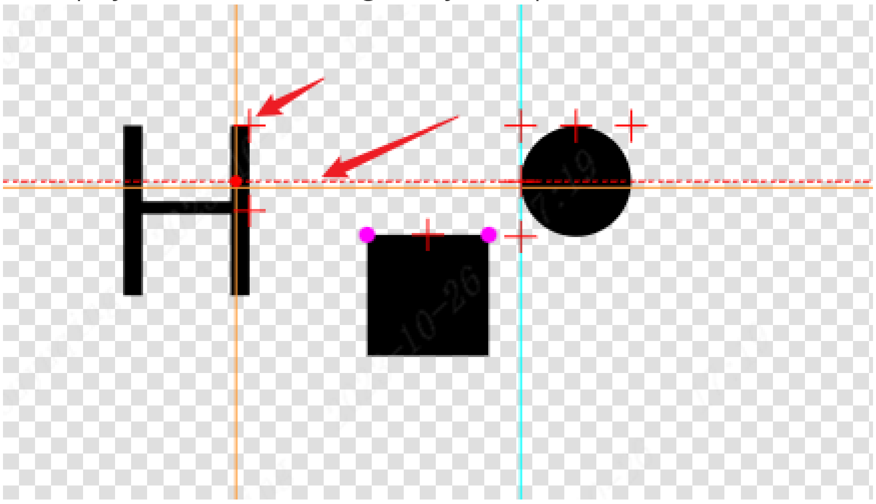


- When the auxiliary object is unselected, the corresponding coordinate value is not displayed at the corresponding position in the ruler area

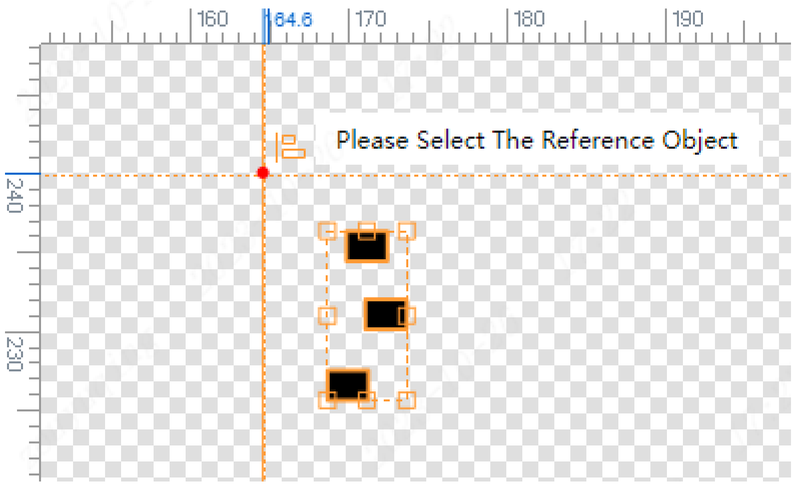


Adsorption, alignment

- Auxiliary lines and auxiliary points both have adsorption functions, which are convenient for other ordinary graphics to align and layout. After dragging the graphic element to touch the auxiliary line and auxiliary point, the adsorption points and auxiliary lines in a cross style will be displayed, which can be aligned by adsorption.

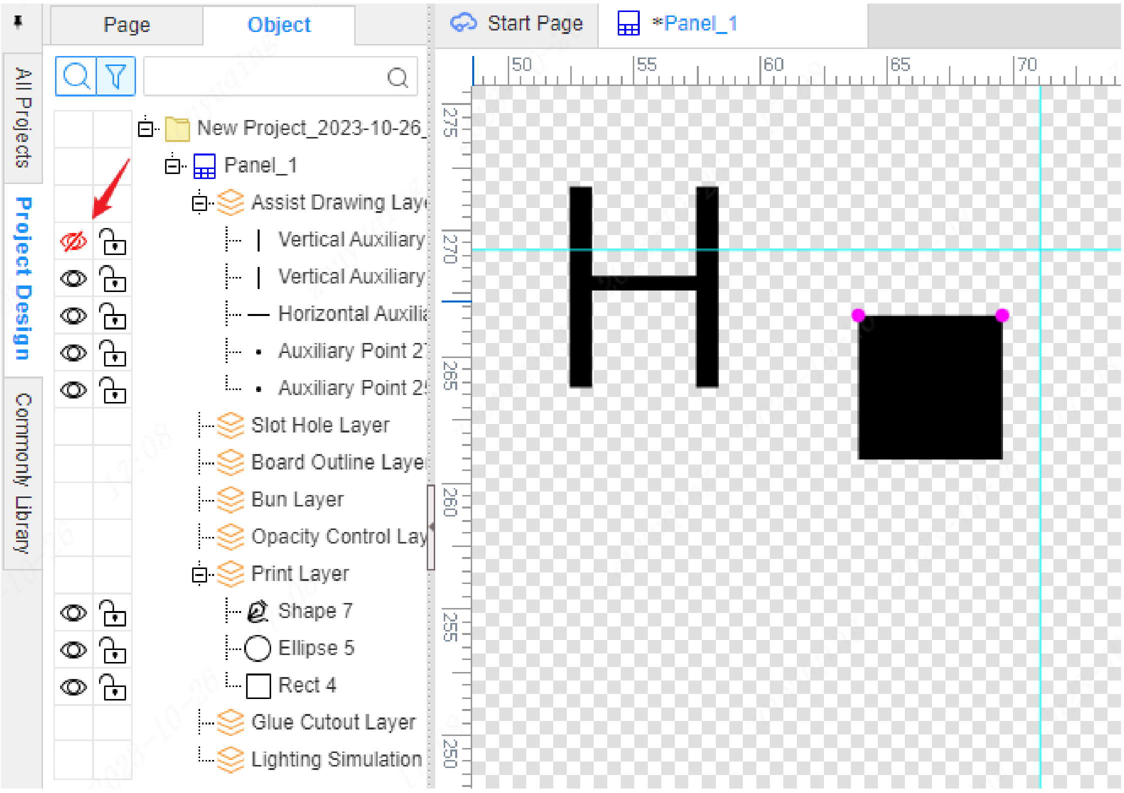


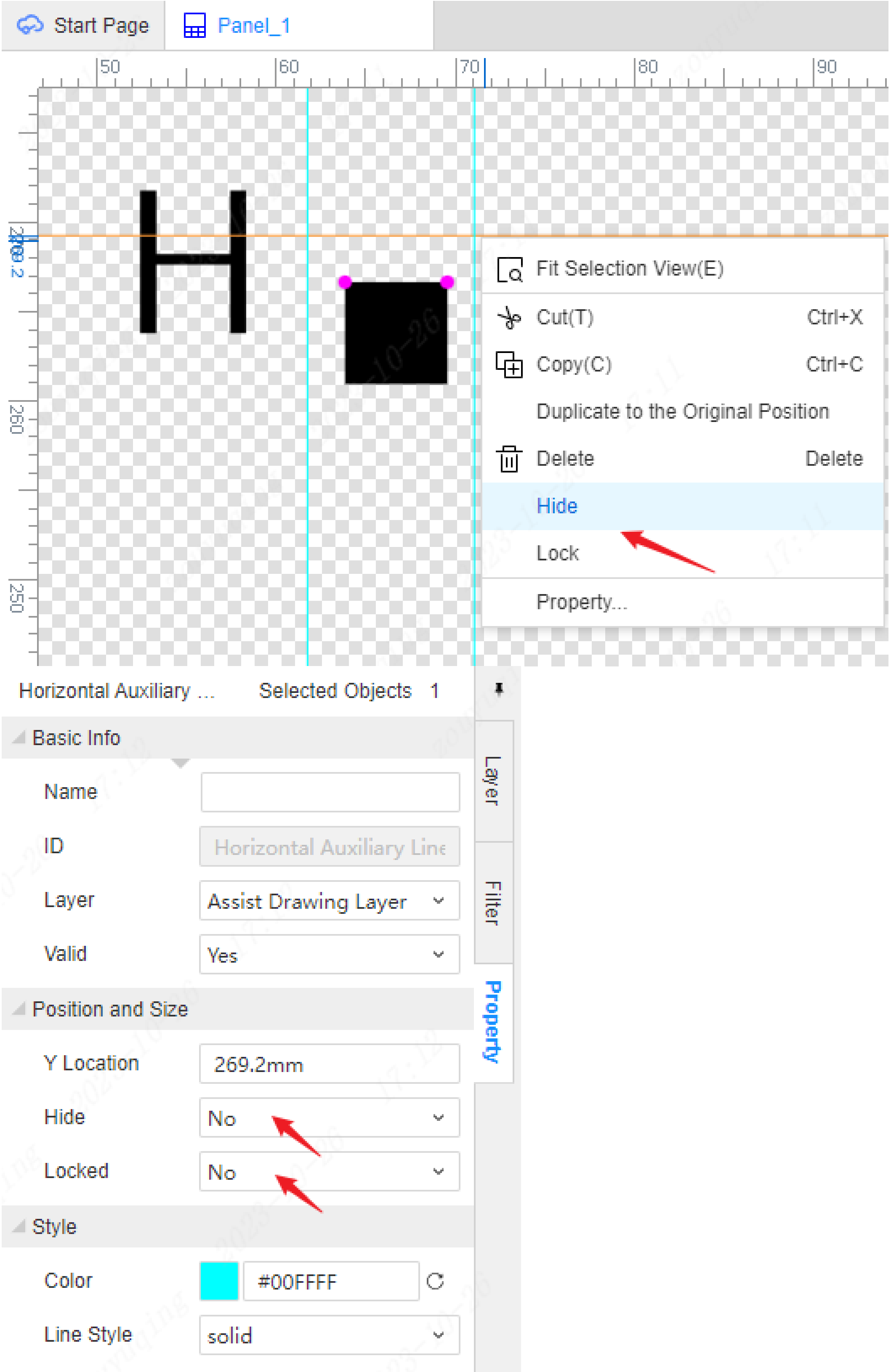
- Support for selecting reference objects aligned with auxiliary objects



Lock unlock & show hide

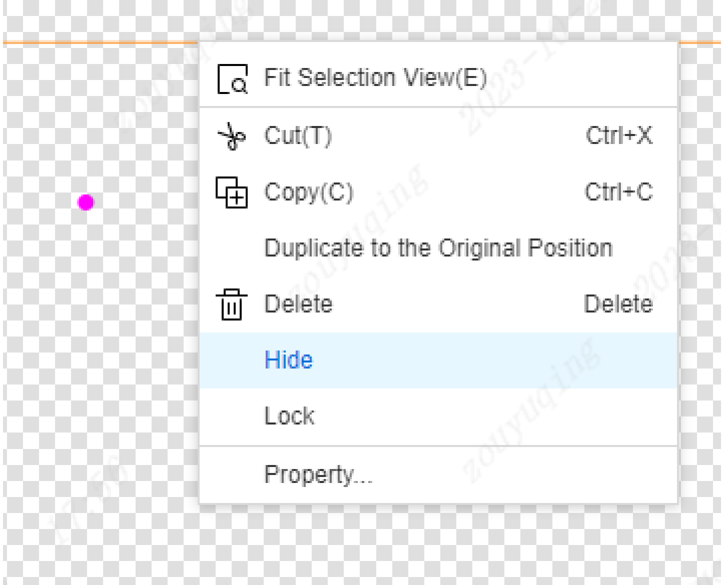
- The left panel of the auxiliary object, the object tree, the right-click menu, and the right property panel, can all be set to lock and unlock and show and hide. The three settings are kept in real-time linkage.





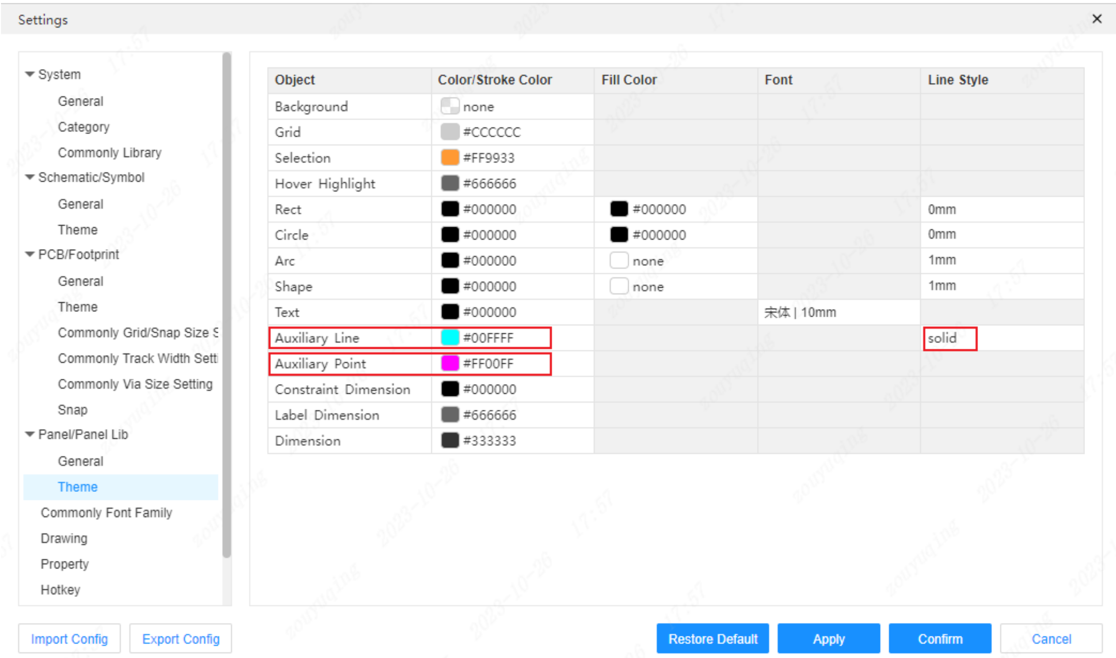
Rotate, flip

- There are two types of auxiliary lines: horizontal and vertical, neither of which supports rotating and flipping with ordinary elements. Elements that do not provide the rotation and flipping menu in the right-click menu cannot be flipped and rotated together with other ordinary elements as a whole. Unsupported elements will be filtered and remain unchanged.



Theme settings

- Settings - Panel/Panel Library - Theme - Auxiliary
- Settings - Panel/Panel Library - Theme - Auxiliary Point



Auxiliary lines can not only be set in color, but also in line type. Auxiliary points can only be set in color.

Dimension

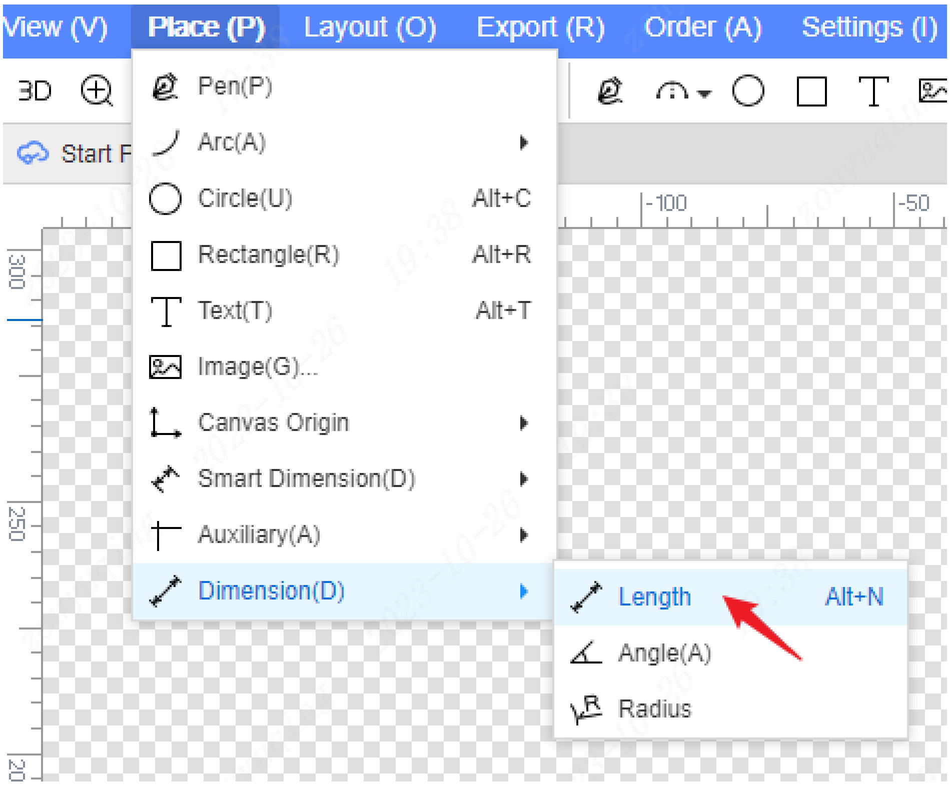
size

The panel also provides the same size drawing function as the PCB , and can choose to measure the length size or angle size or radius size, which exists as an independent auxiliary element and is not effective in actual production.

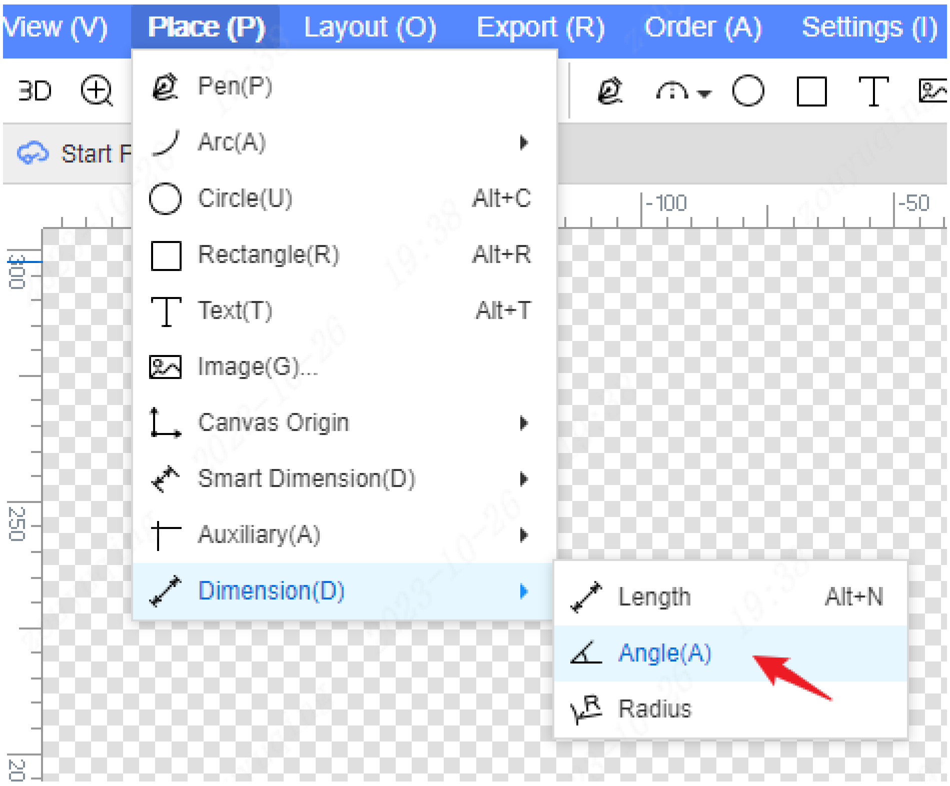
Function entry

Top menu

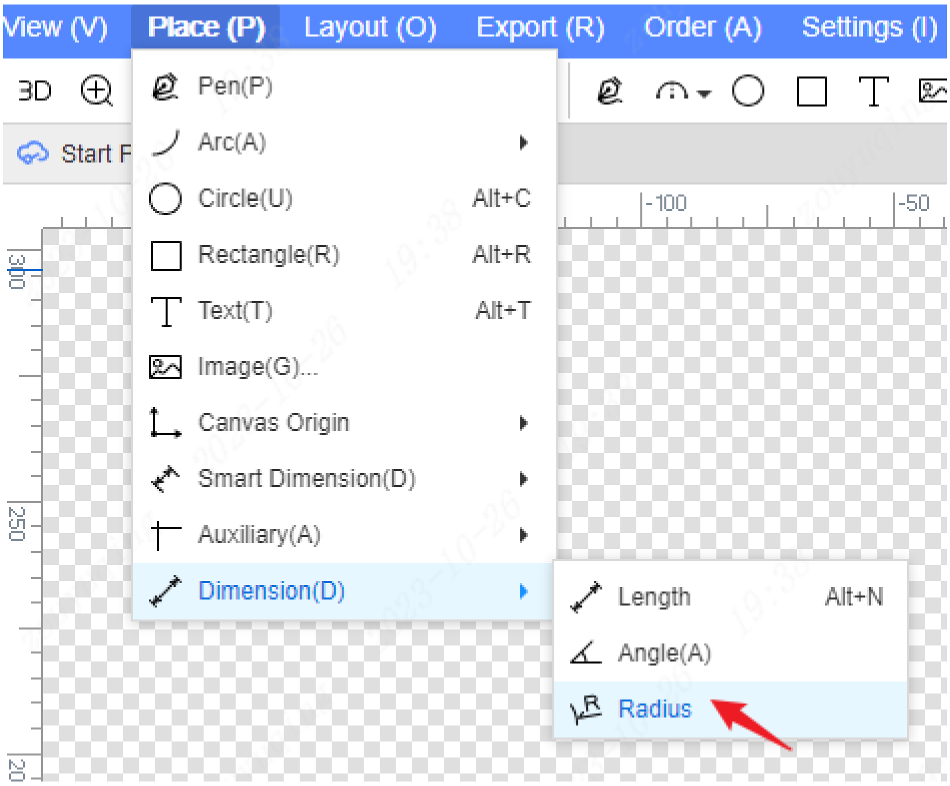
- Top Menu-Placement-Size-Length
- shortcut: `Alt` + `N`



- Top Menu-Placement-Size-Angle
- Menu shortcut: `P` + `D` + `A`

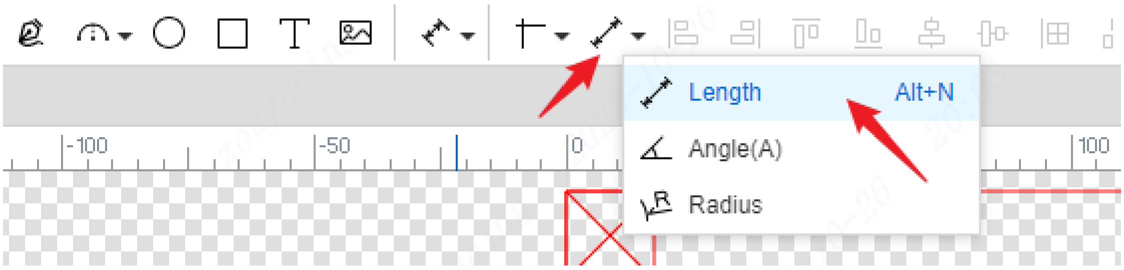


- Top Menu-Placement-Size-Radius



Top toolbar

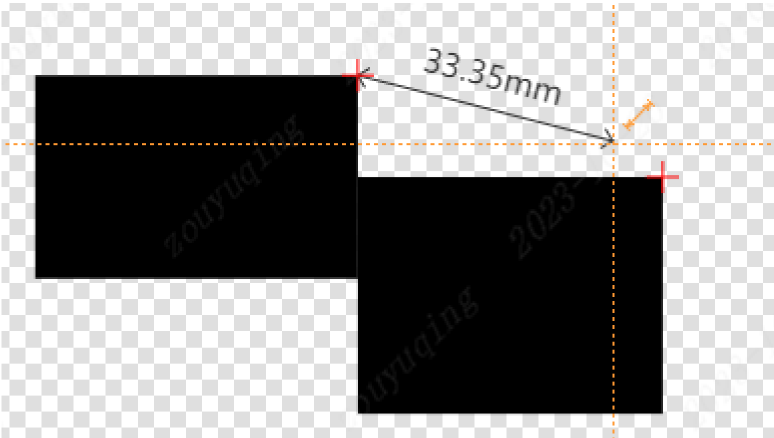
- Click the size icon directly, or expand the drop-down option to switch the selection size drawing type



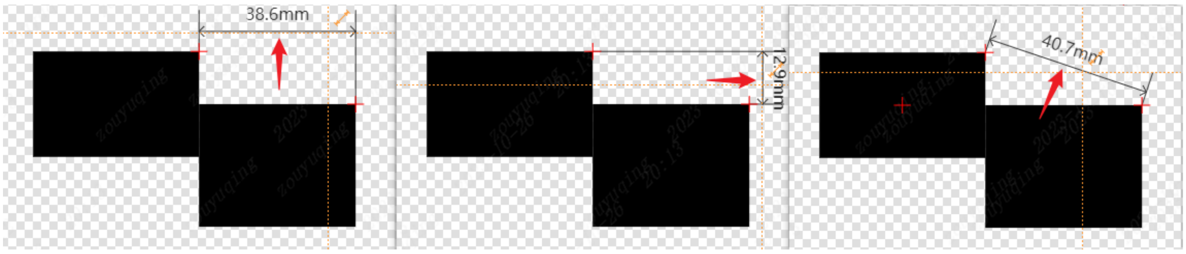
Operation steps

Length

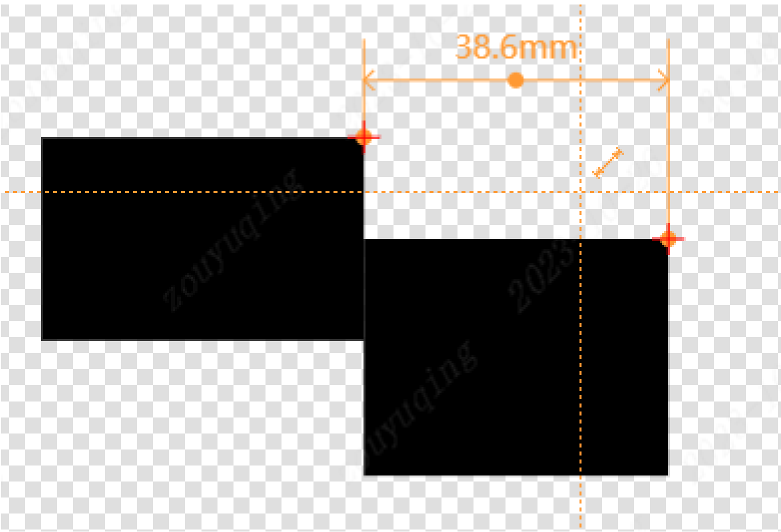
- Enter the length size drawing mode from the function entry operation, and the left mouse button on the canvas can determine the starting measurement point at any position. Move the cursor to display the length size to be determined



- Click the left mouse button again anywhere on the canvas to determine the end point of the length dimension. Display the size between two points, and moving the cursor will switch to measure the horizontal, vertical, or straight-line distance between the two points.

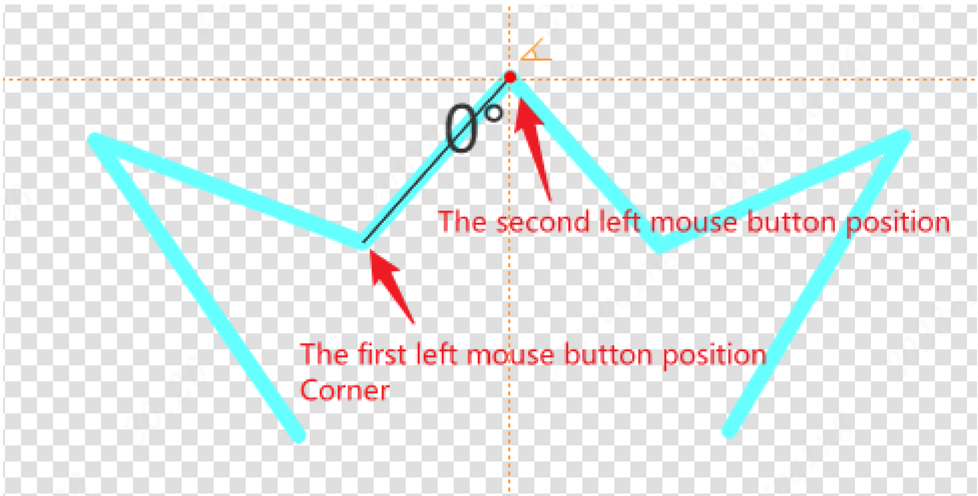


After determining the measurement direction and style position of the length dimension, the left mouse button can complete the drawing of this length dimension and enter the drawing of the next length dimension.

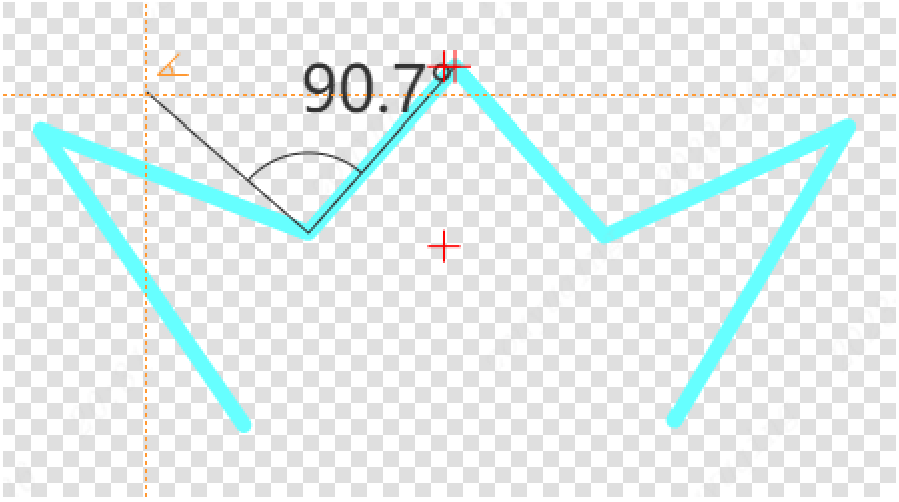


Angle

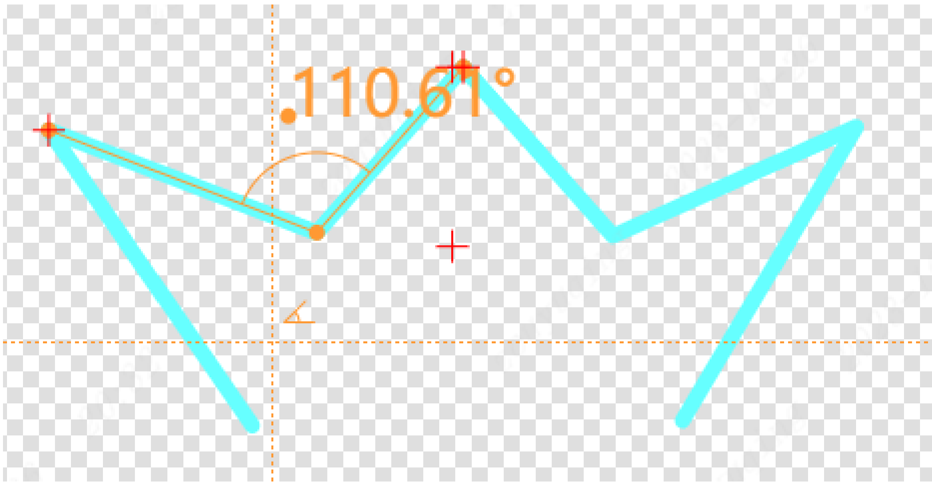
- Enter the angle size drawing mode from the function entrance operation. The first left mouse button on the canvas can determine the position of the corner . Move the cursor to display the corner length to be determined, and continue the second left mouse button to determine the corner length.



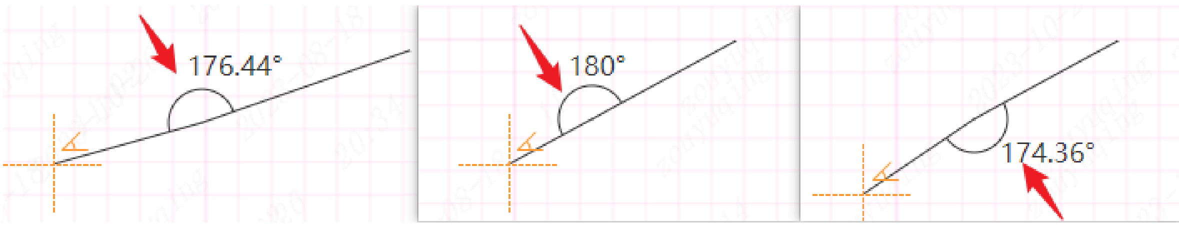
- Move the mouse to display the angle size to be determined, and the angle value will be updated in real time as the mouse moves.



- After continuing with the third left mouse button, you can determine the position of the other corner, complete the angle size drawing, and enter the next angle size drawing.

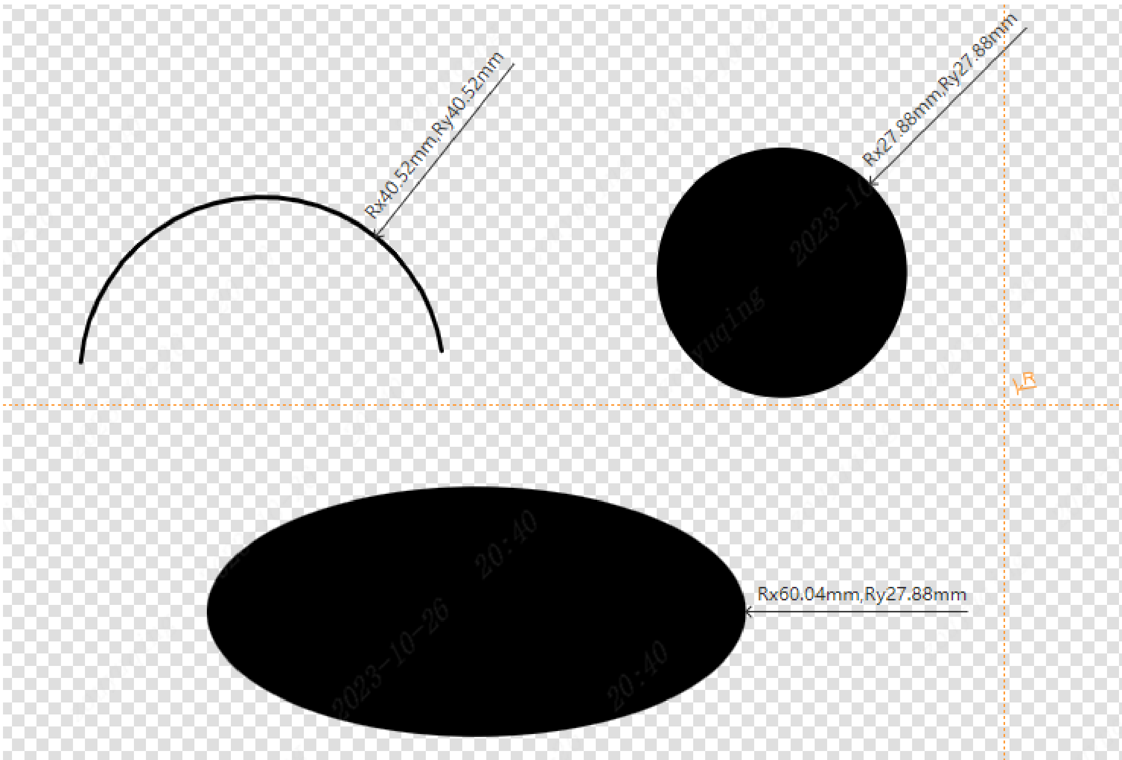


- The maximum angle is 180 °, if it exceeds, the reverse angle of the measurement will be displayed

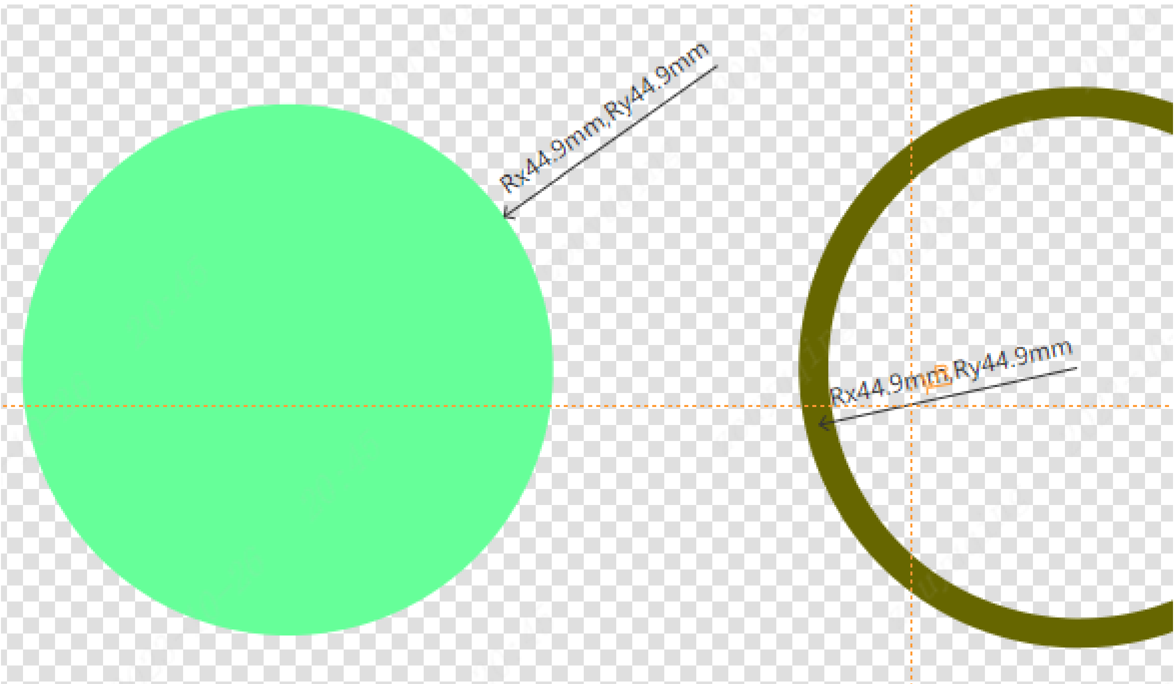


Radius

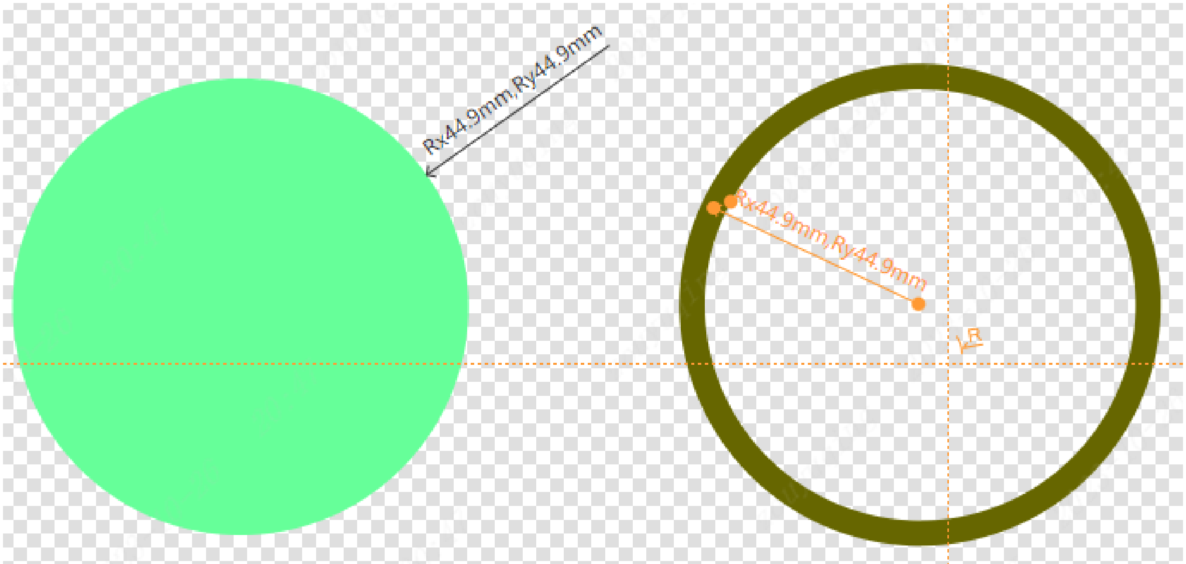
- The drawing of the radius size depends on the center of the circle, and the circles and arcs in the panel can be measured for radius.



- Enter the radius size drawing mode from the function entrance operation, move the cursor to the unfilled circle or the filled circle, and display the actual measured radius size after the left mouse button



- You can move the radius size arbitrarily on the circumference. After adjusting to the appropriate position, continue to left-click the mouse to complete the drawing of this radius size and enter the drawing of the next radius size.

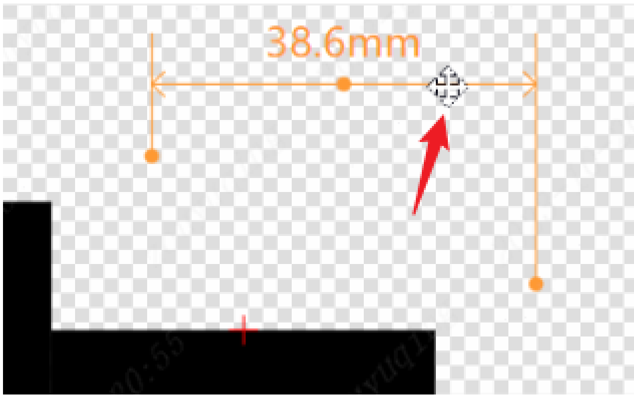


- The length and radius of a perfect circle are consistent, while the length and radius of an ellipse are inconsistent. Measuring the radius size will measure and display both the long and short radii of the object being measured.
- The radius dimension measurement of the arc is consistent with the measurement method of the circle.

Rules and Agreements

Whole movement

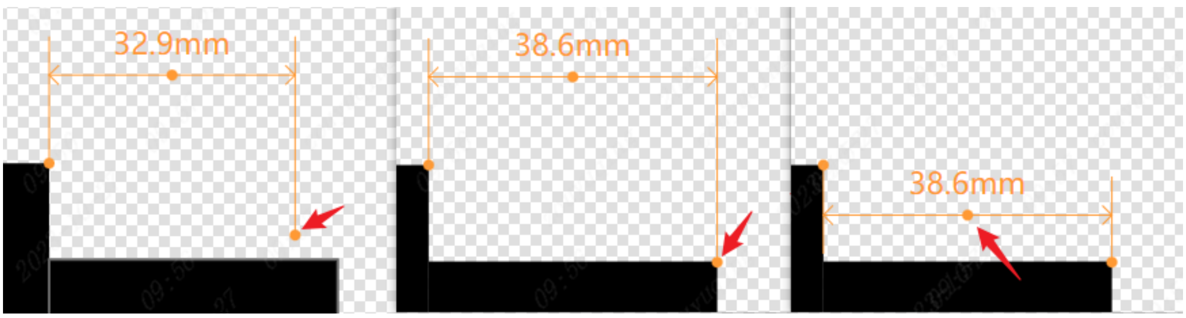
- Select the size, there are no 8-point control boxes, only the selected color highlight is displayed, the cursor moves to the position of the non-size control point, the drag cursor style is displayed, and the entire size can be moved.



- You can also drag the size element directly without selecting it to adjust the position of the size.

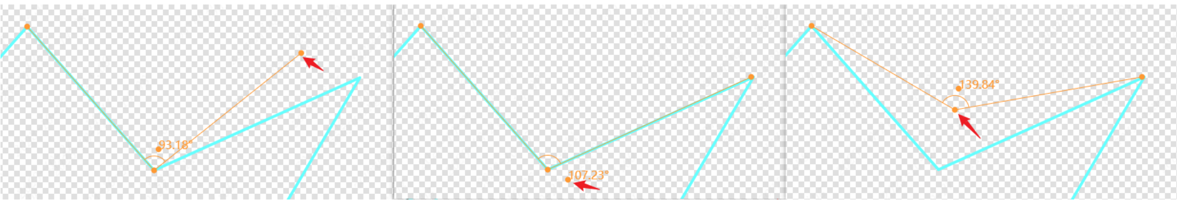
Control Point - Resize

- **Length dimension**, providing three control points, two measurement reference points for the length dimension and a numerical text control point in the middle of the dimension line



After selection, hold down the left mouse button to move the control point, you can adjust the measurement reference point of the length size and update the length value in real time. You can also adjust the position of the numerical text to change the size style without moving the entire length size.

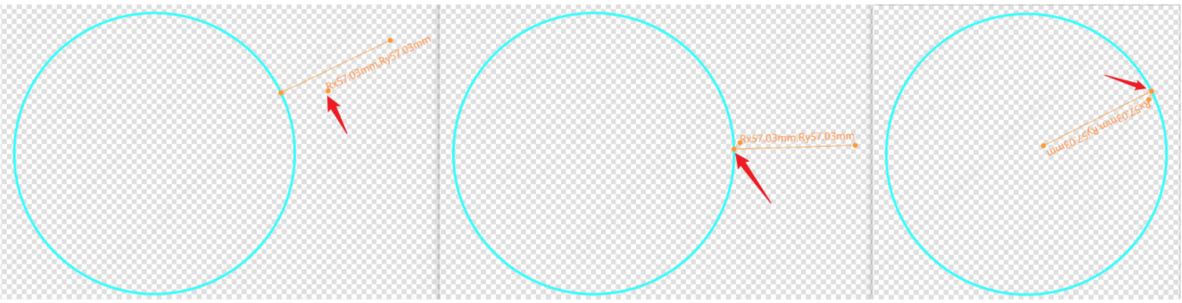
- **Angle dimensions**, providing 4 control points, namely 3 measurement reference points for angle dimensions (i.e. corner points and 2 corner edge points) and numerical text control points on the arc dimension line



After selection, hold down the left mouse button to move the control point, you can adjust the measurement reference point of the angle size and update the angle value in real time. The numerical text can also be adjusted to any position by moving the corresponding control point.

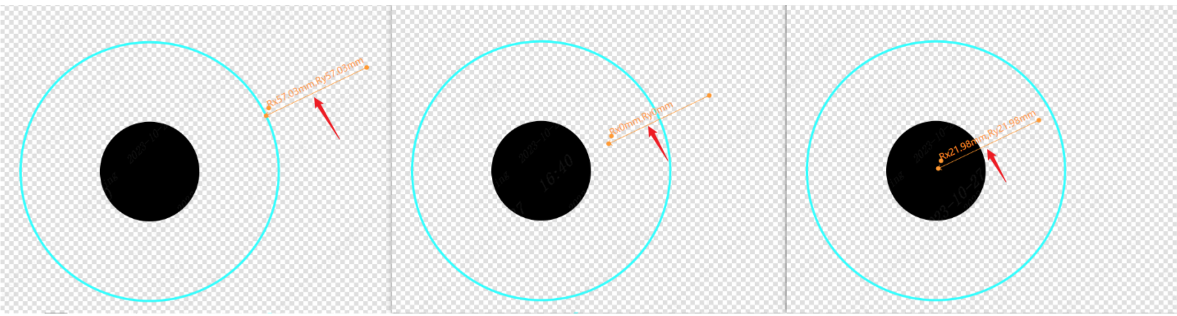
Note: Once the control point at the corner is moved, the numerical text will automatically adjust back to the arc size line.

- **Radius dimension**, providing 3 control points, namely the measurement reference point of radius dimension, the size line length control point and the numerical text control point



After selection, hold down the left mouse button to move the control point, and the control point with adjusted radius size can move freely. The position of the measurement reference point determines the actual measurement object. The numerical text can be moved freely, and the length of the size line can be adjusted.

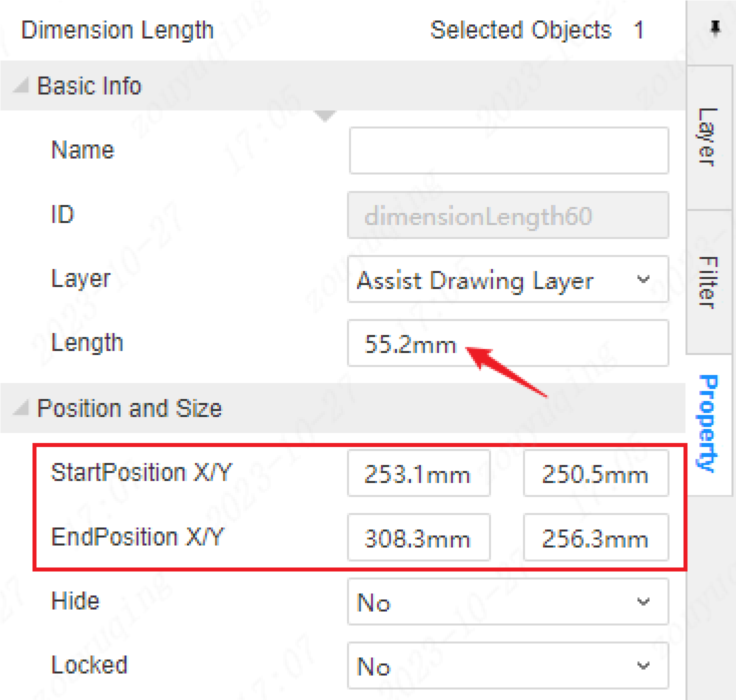
Note: Once the measurement reference point of the radius size moves away from the circumference or fills the circle area, the value is set to 0. Dragging the measurement reference point to any circumference or fills the circle will re-measure the radius.



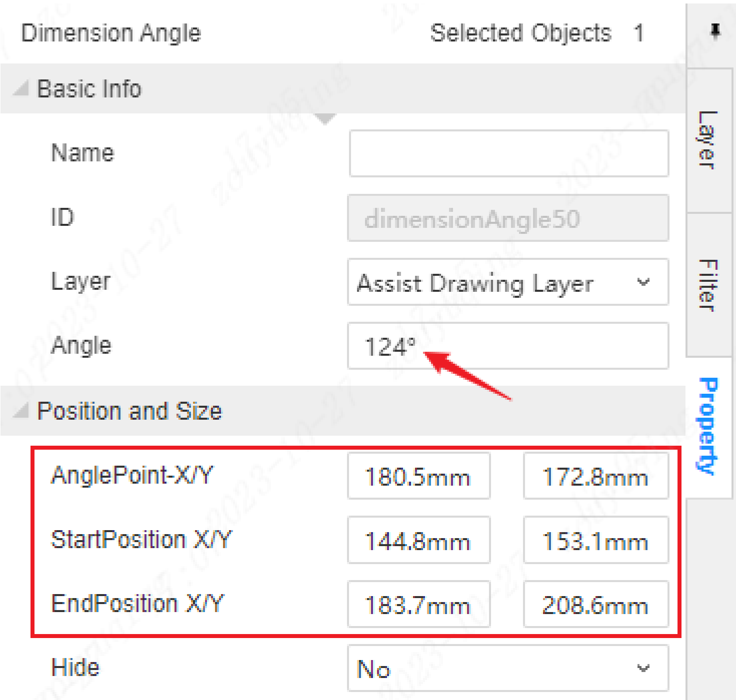
attribute

Measurement reference point coordinates & size values

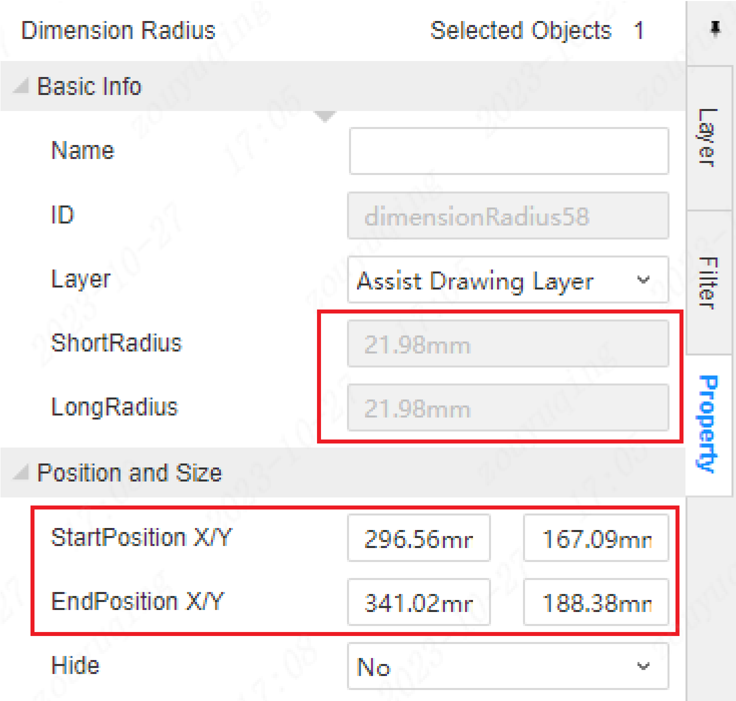
- In the attribute panel on the right side of the length dimension, not only can the coordinate values of its two measurement reference points be adjusted, but also the length dimension values can be adjusted.



- In the attribute panel on the right side of the angle dimension, not only can the coordinate values of its three measurement reference points be adjusted, but also the angle dimension values can be adjusted.

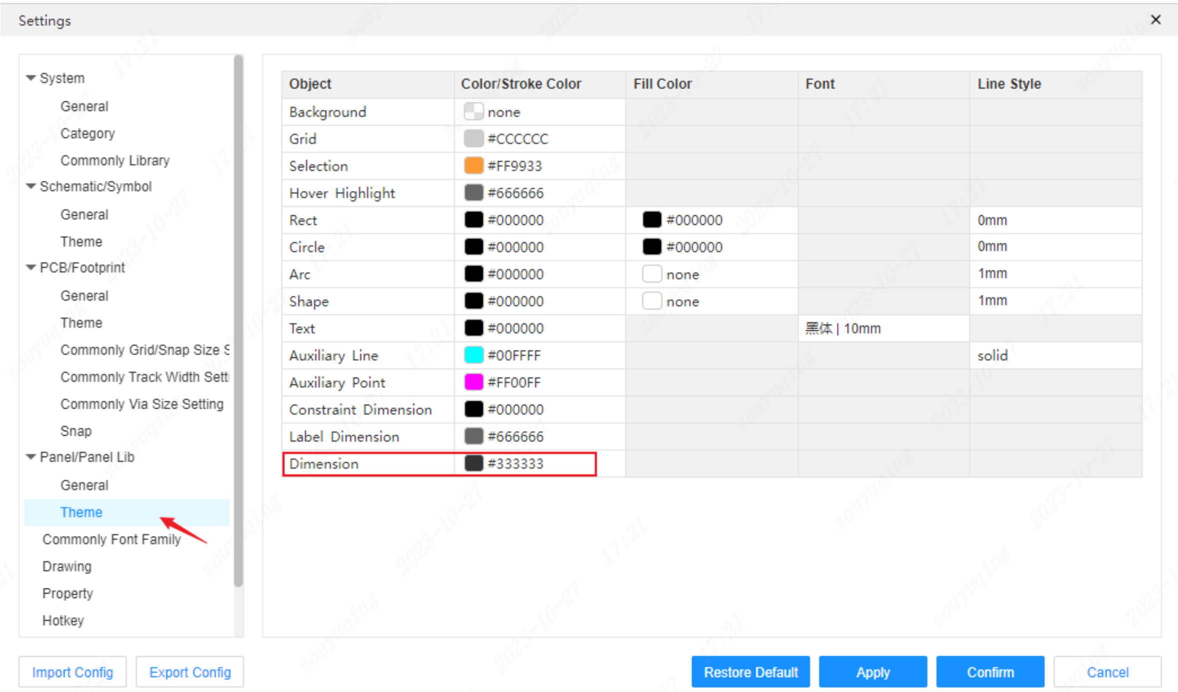


- In the property panel on the right side of the radius dimension, the starting point X/Y controls the coordinates of the measurement reference point, while adjusting the ending point X/Y will change the length of the radius dimension line. The length and length of the radius dimension values are only displayed in grey out and cannot be edited.



Theme settings

- Settings - Panel/Panel Library - Theme - Size



The default size theme color is #333333, and custom adjustments are supported.

Group

Group

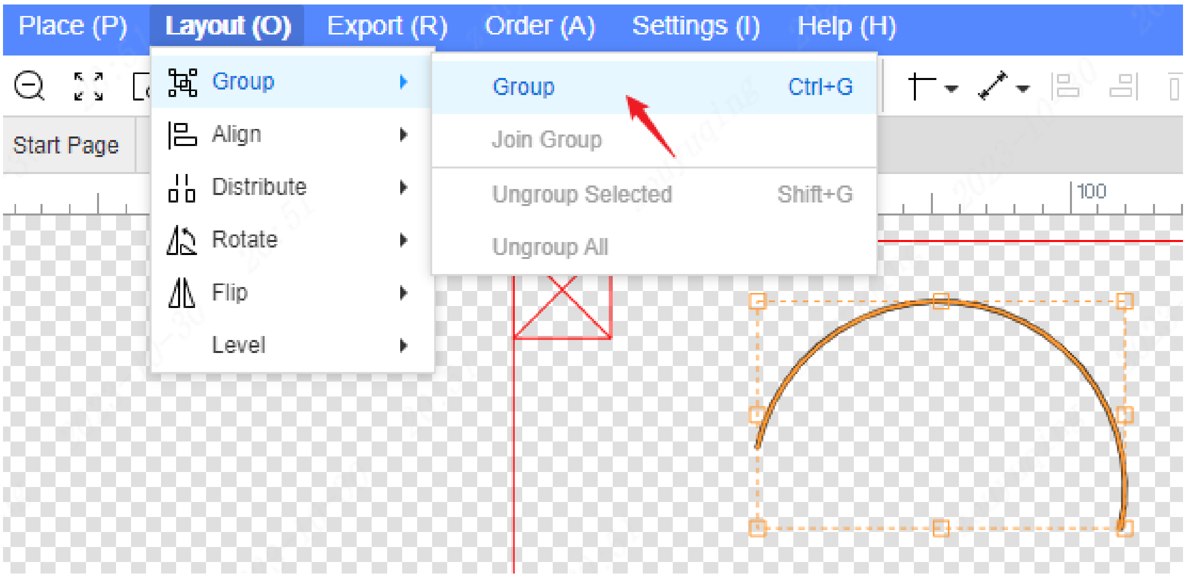
The panel supports the creation of group, which is convenient for establishing group relationships between elements. After combining multiple scattered elements, they can be moved as a whole on the canvas. Editing the elements in the group needs to enter the group mask, and only the elements in the group mask can be operated, which can effectively block the interference of other non-combined elements.

Function entry

Top menu

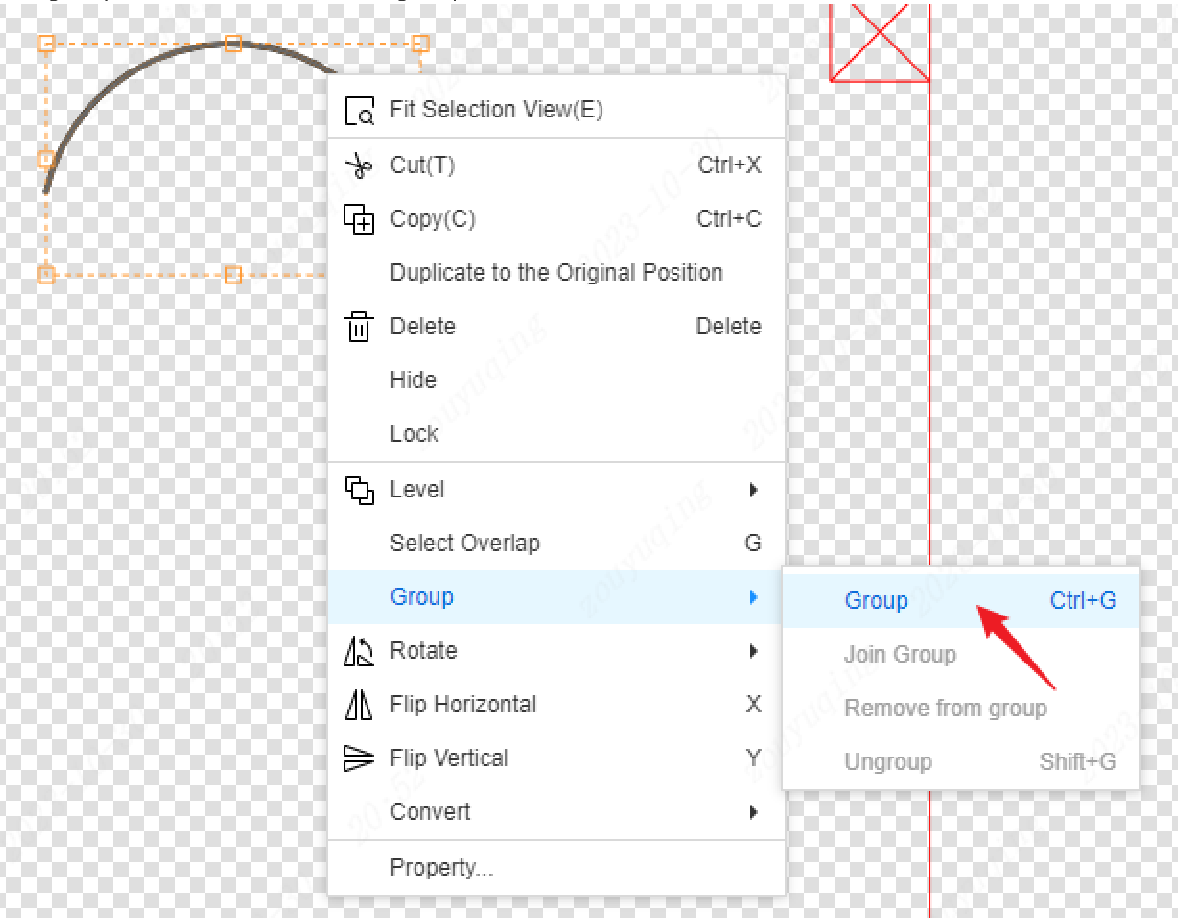
At least one object must be selected on the panel canvas to create a group.

- Select Element - Top Menu - Layout - Group - Group
- Global menu shortcut: `Ctrl`+`G`



Right click menu

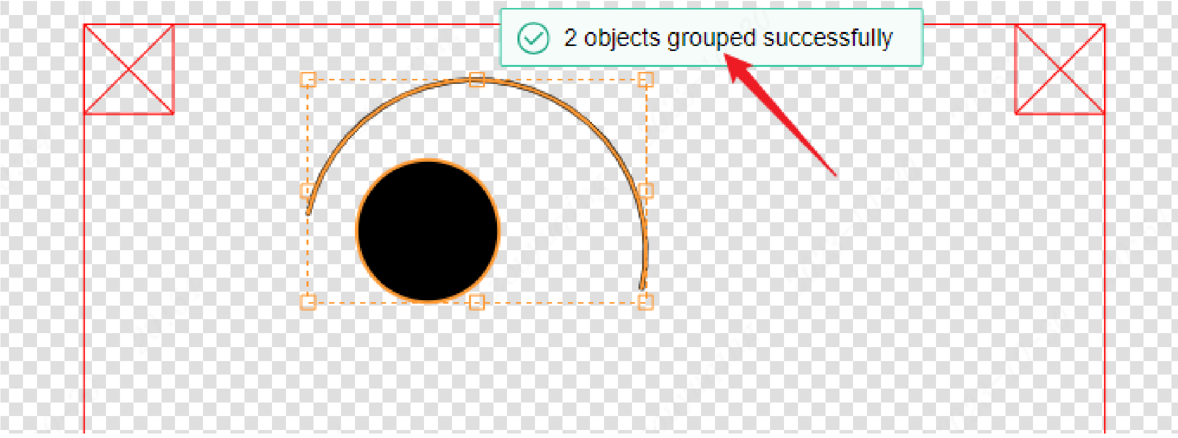
After selecting the element on the canvas, right-click the mouse to operate the submenu group of the group menu to create a new group.



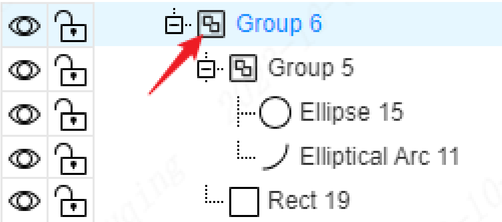
Instructions for use

Create a group

Create a new one by using the top menu or right-click menu or pressing Ctrl + G. A short prompt will appear in the center of the canvas to indicate the successful group.
The elements in the same group will remain as a whole on the canvas. Clicking on any object in the group will select the entire group.



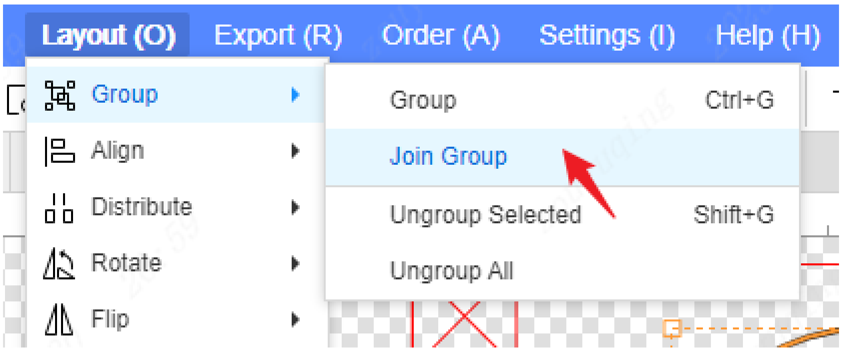
Panel supports **nested group** . When the selected object contains a group, you can continue to operate to create a new group, generating a new parent group containing the original selected subgroup.
Note: A sub-group is a sub-object under the parent group, and the elements in the sub-group maintain the original group relationship.



Join group

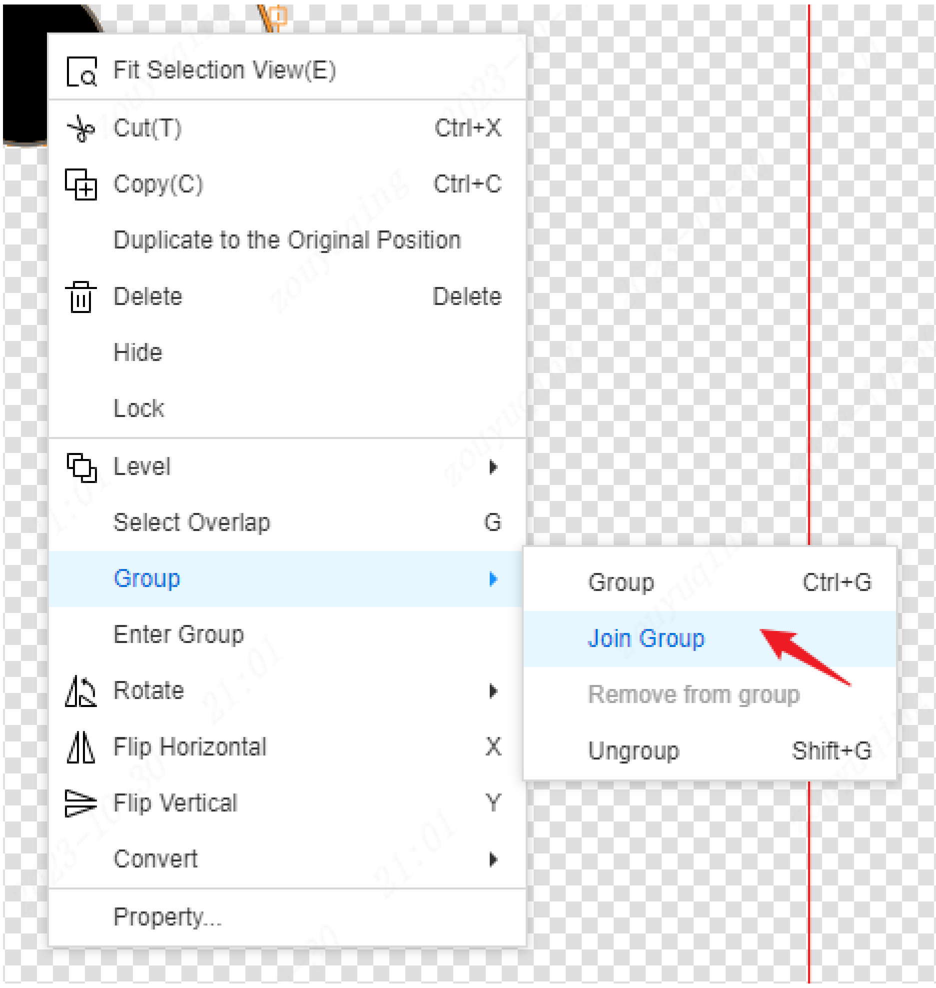
There must be a group and at least one object must be selected to operate on joining the group.

- Select Object - Top Menu - Layout - group - Join group

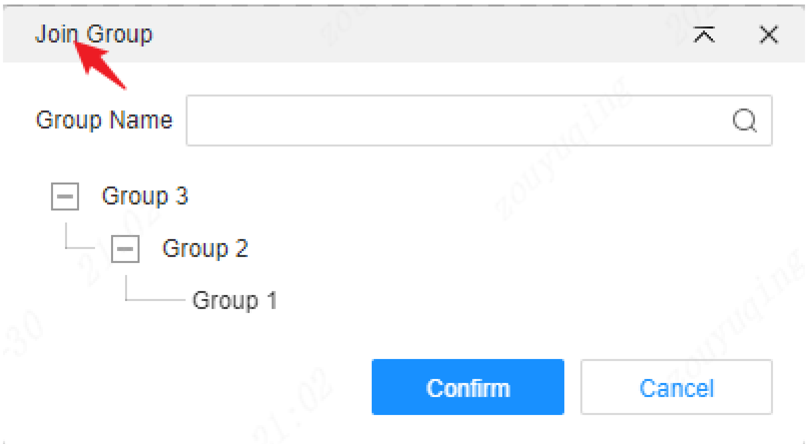


Or

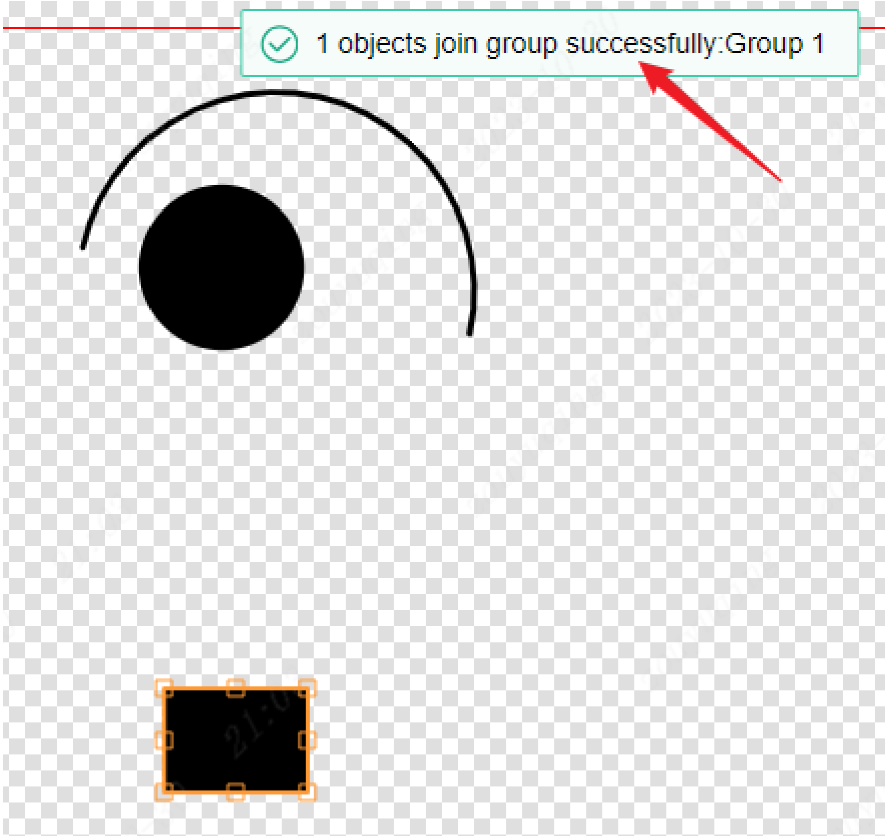
- Select Object - Right Click Menu - group - Join group



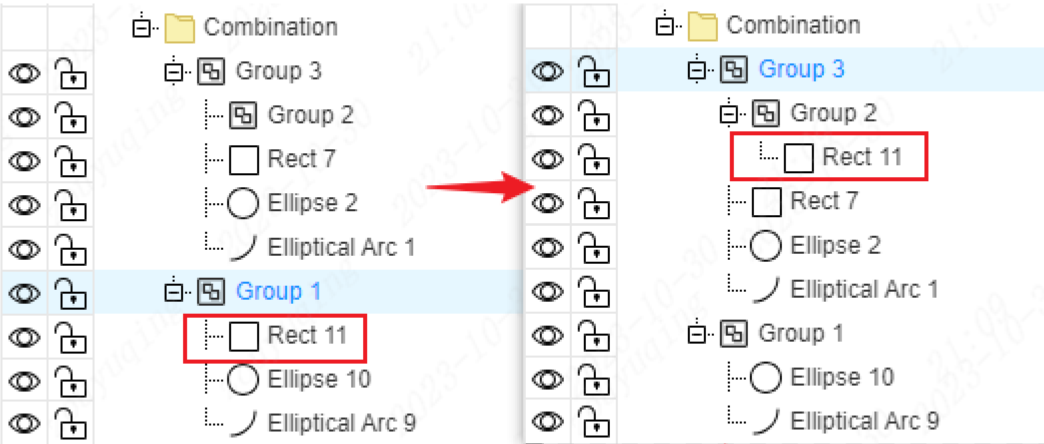
After the operation to join a group, the Add to group selection pop-up window will pop up. The pop-up window displays the names of all existing groups, and the tree structure is convenient for intuitive preview of the relationship between nested groups.



After selecting the group name to be added, click the Confirm button, and a short prompt in the center of the canvas will indicate that the group has been successfully added.



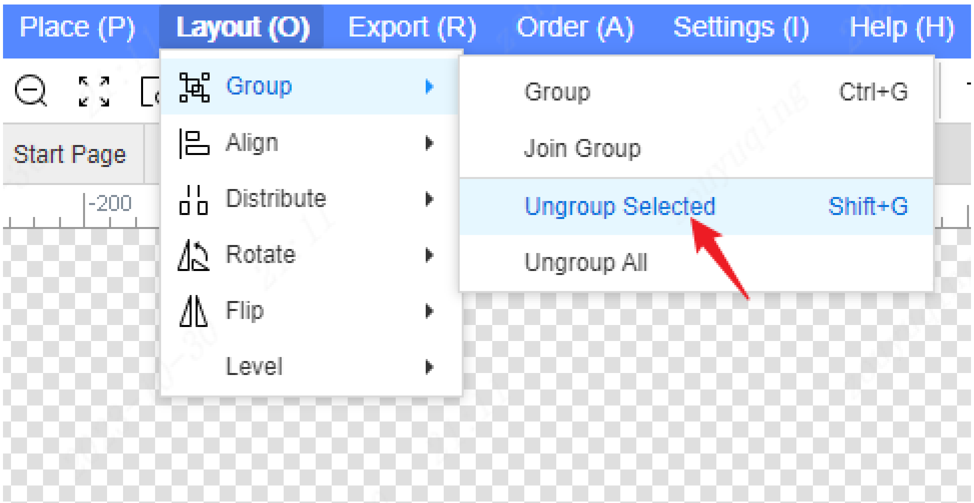
Note: Any object can be added to any group, even if it already exists in a group, it will be moved out of the group and then added to the target group (panel group allow nesting and cannot cross into a loop).



Ungroup selected

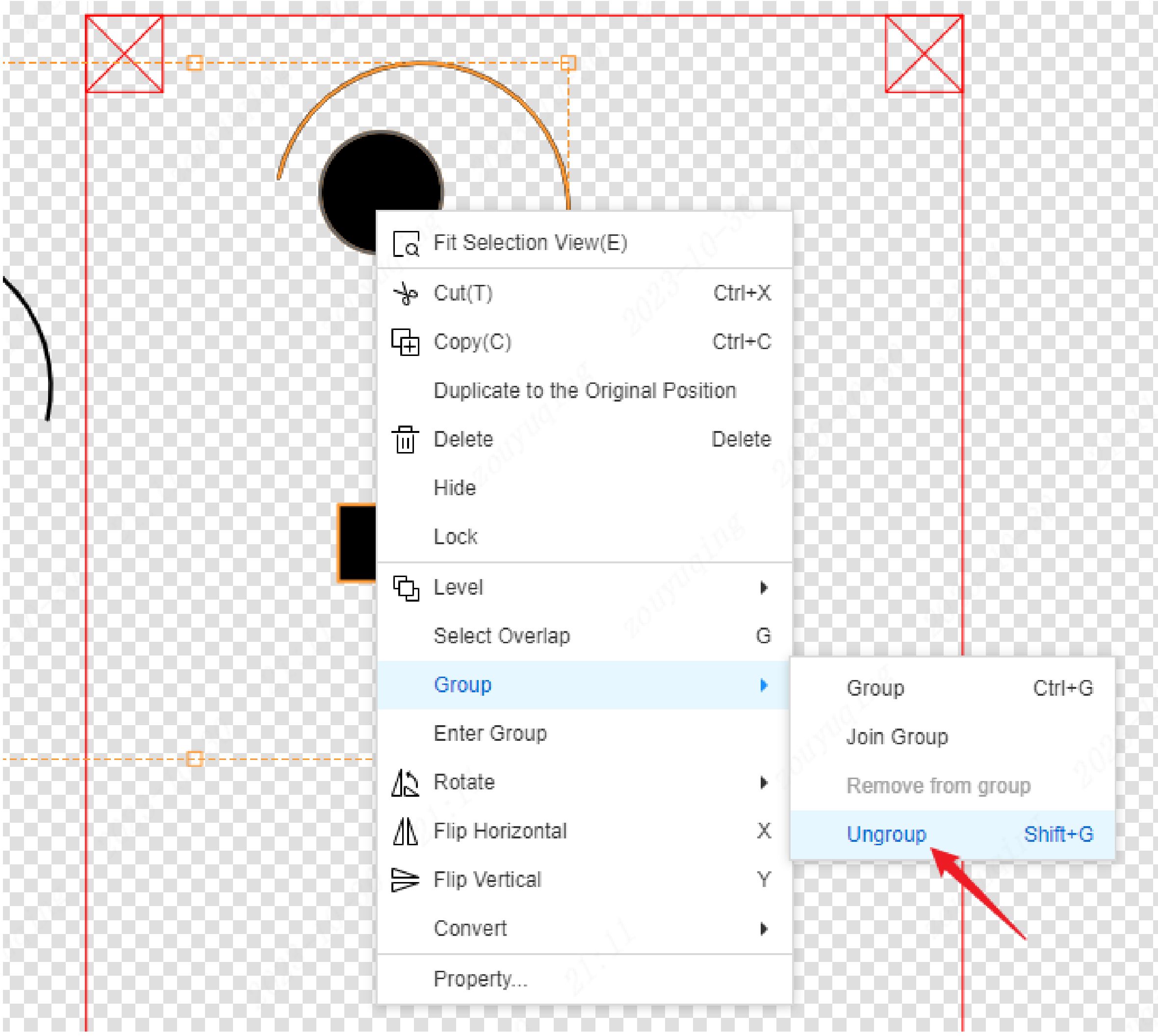
You must select when there is a group before you can operate to cancel the group.

- Select Group - Top Menu - Layout - Group - Ungroup selected
- Global shortcut: `Shift`+`G`

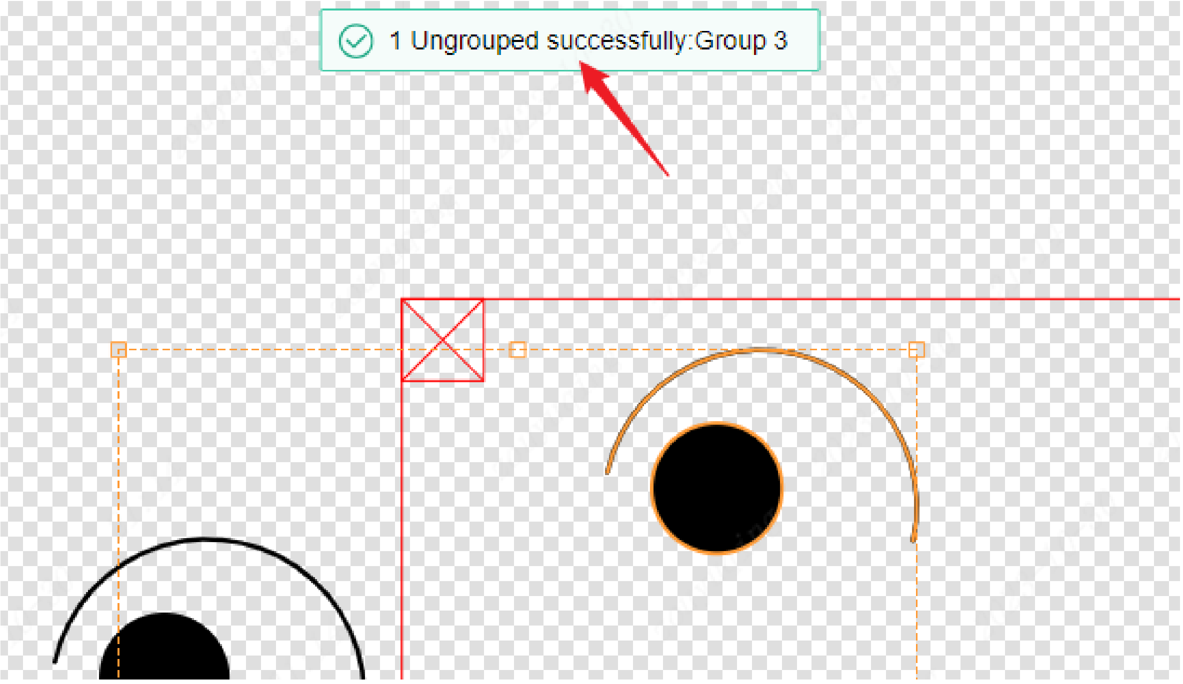


Or

- Select Group - Right Click Menu - Group - UnGroup selected

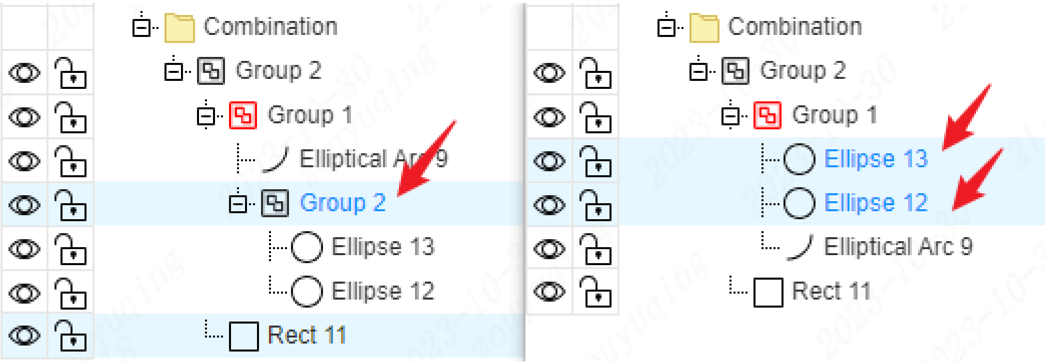


After the operation cancels the group, all the selected combinations will be canceled, and the short prompt in the center of the canvas will cancel the group successfully.



Note:

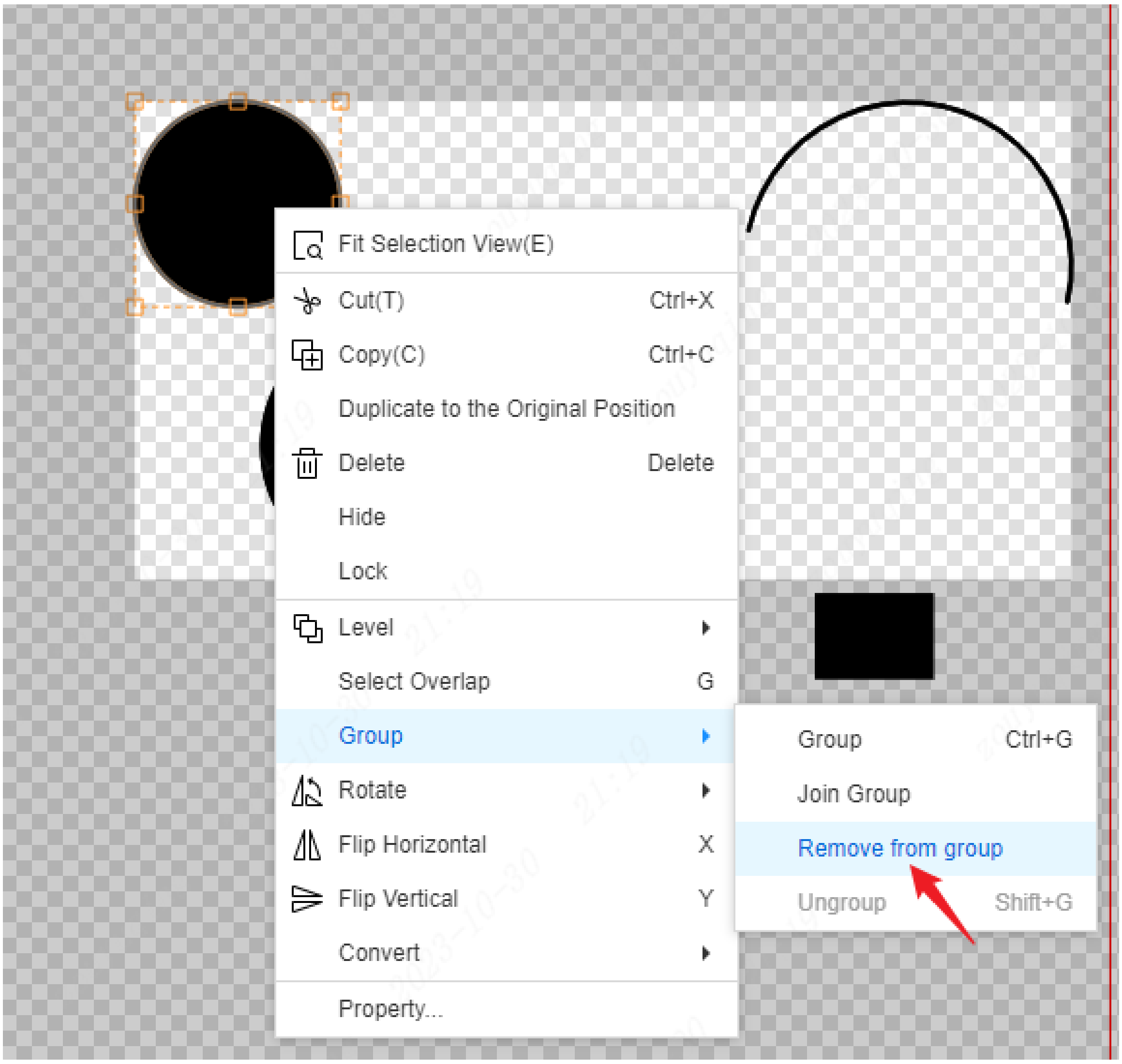
- To cancel a combination, you must select the entire combination to operate. Selecting only the objects within the combination cannot operate to cancel the combination state of the combination it is in.
- Cancel the combination status of the sub-combination, and the objects in the original sub-combination will be moved to the parent group of the sub-combination at the previous level.



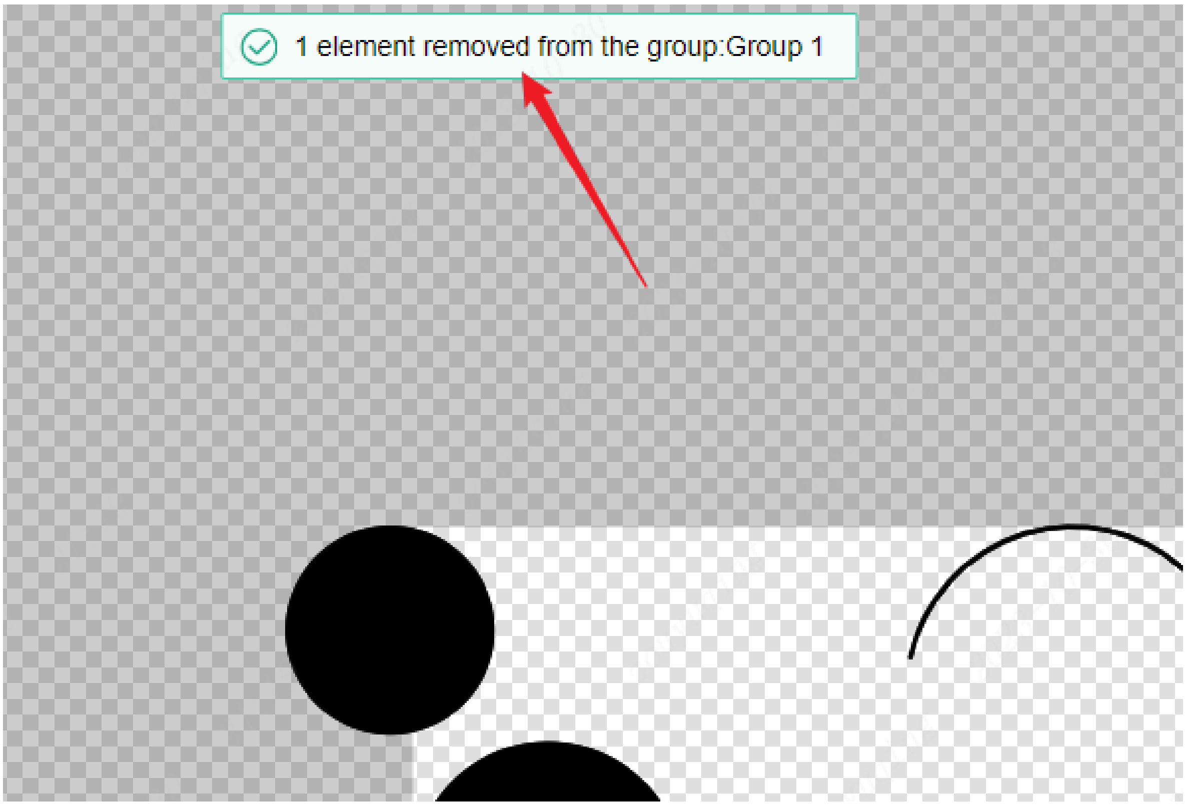
Removal from group

You must select the objects in the group before you can operate to remove them.

- Select Object - Right Click Menu - Group - Remove from group

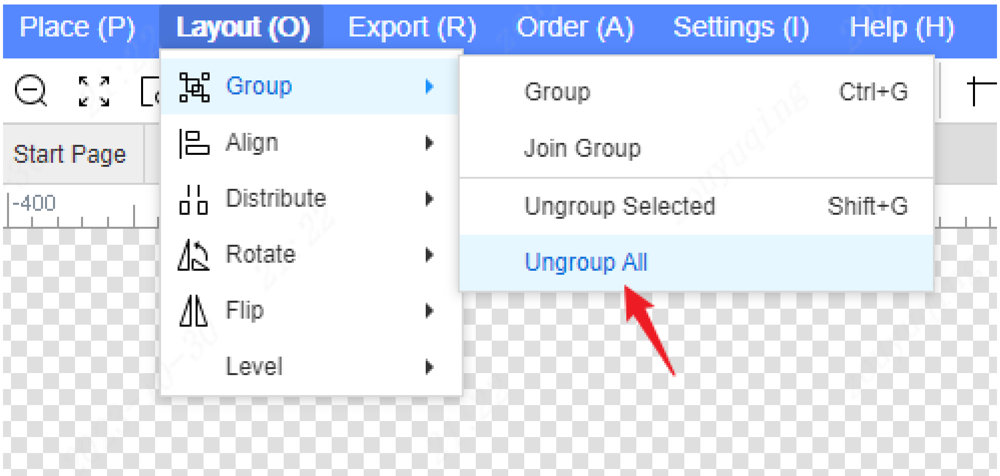


After the operation is removed from the combination, the selected object will be directly removed from all combinations, and the short prompt in the center of the canvas indicates that the object has been successfully removed from the combination.



Ungroup all

- Top Menu - Layout - Gorup - Ungroup all

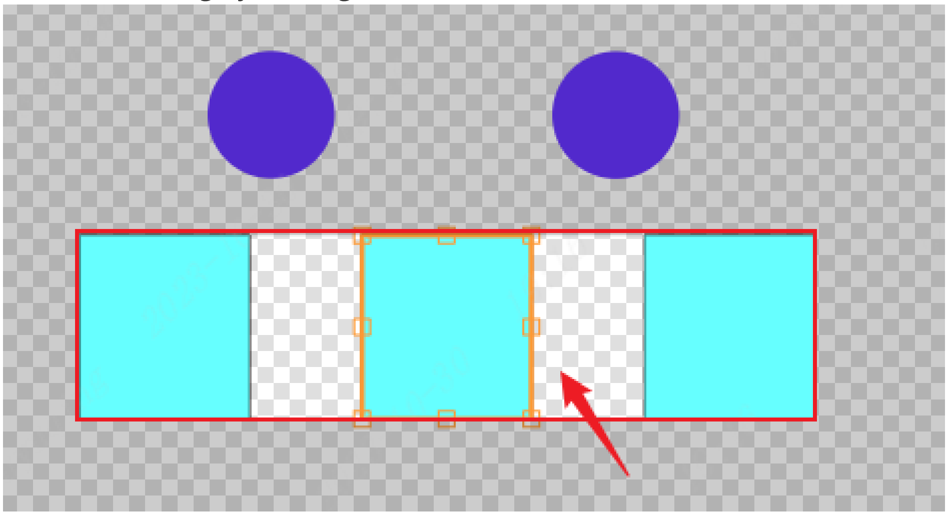


Operation Cancel All Combinations will cancel all existing combination statuses on the current panel canvas, and a short prompt in the center of the canvas will indicate that all combinations have been successfully canceled.

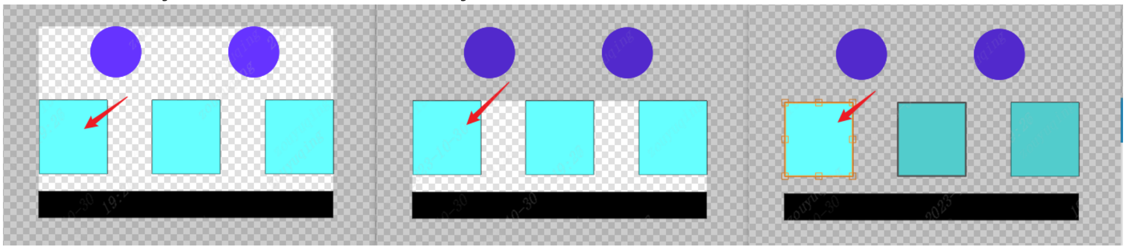
Gorup mask

In order to facilitate the operation of objects in the combination without accidentally touching other non-combination objects, the function of combination mask has been added. Double-clicking any object in the combination on the canvas can enter the combination mask. After entering the combination mask, other non-combination objects cannot be selected. You can double-click or press Esc to exit the combination mask.

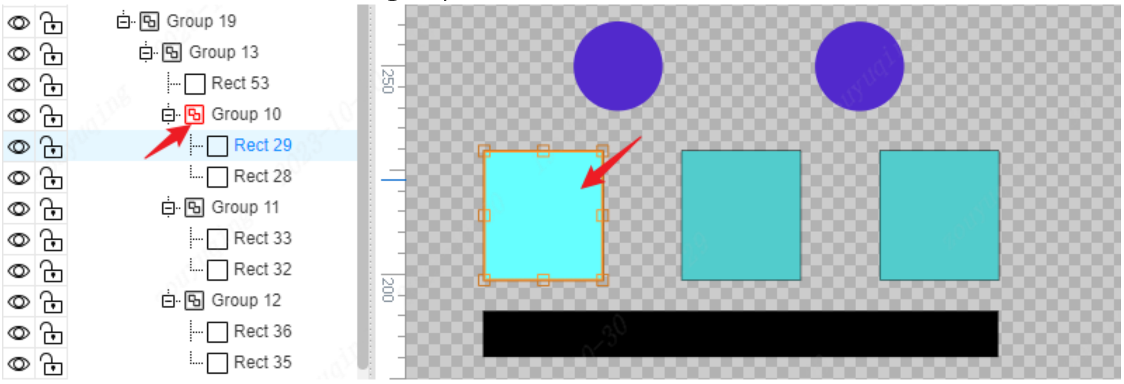
- Enter the combination mask, the normal brightness display in the visible mask area on the canvas, and the gray coverage effect outside the mask area.



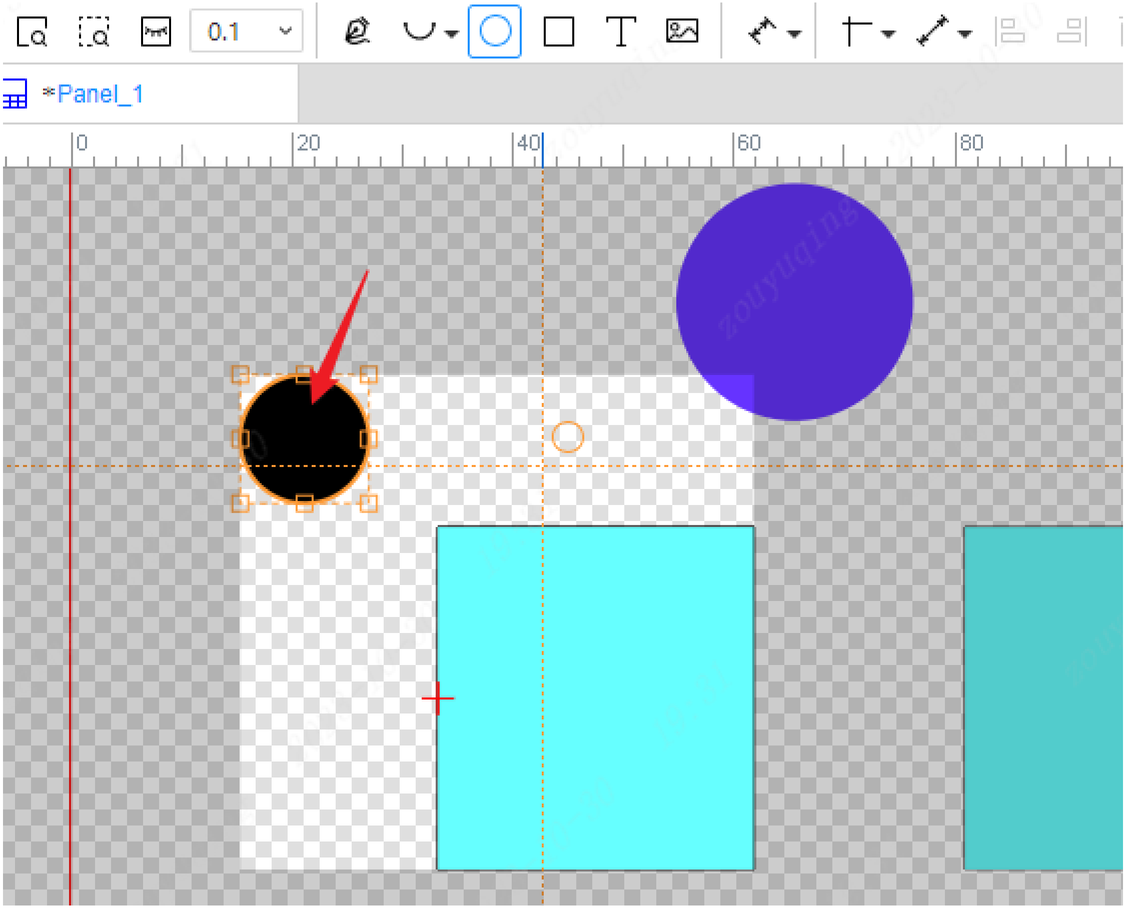
- Nested group need to be entered step by step. Every time you double-click an object in the group, switch to the next level of group mask, and only after entering the minimum group where the object is located can the object be selected on the canvas.



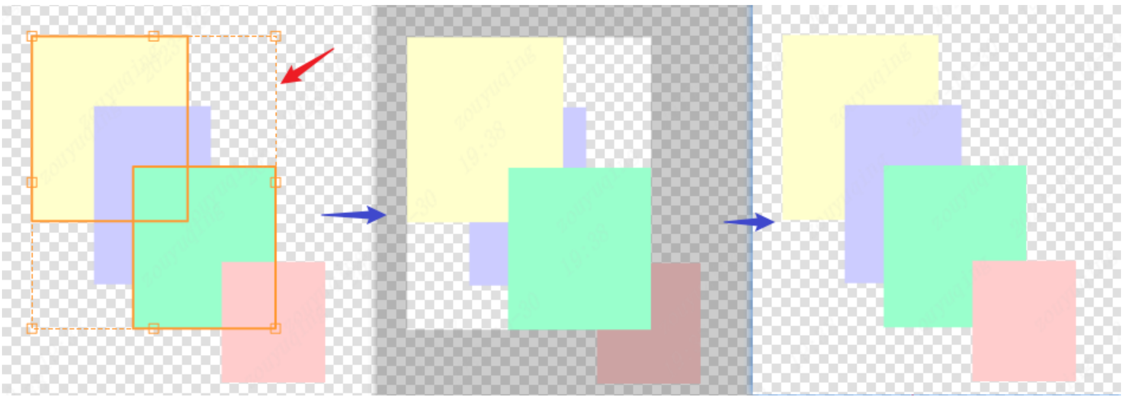
- The object tree on the left directly selects the objects in the group, and you can directly enter the mask of the smallest group to select them. The group icon displayed in red indicates that the current canvas is under the group mask.



- Under a group mask, new primitive objects drawn will be automatically added to the group.



- After entering the group mask, all objects in the group will be temporarily elevated and displayed above all non-group objects, while the hierarchical order of objects in the group will be maintained. Exiting the mask will restore the original hierarchical priority display order.



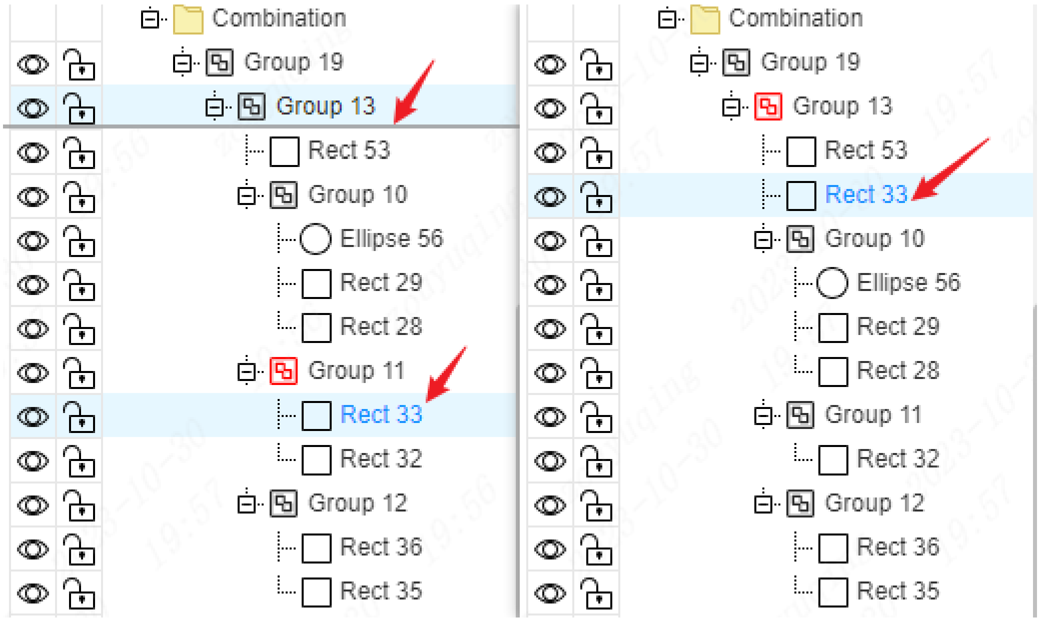
Left group object tree

Anchor positioning

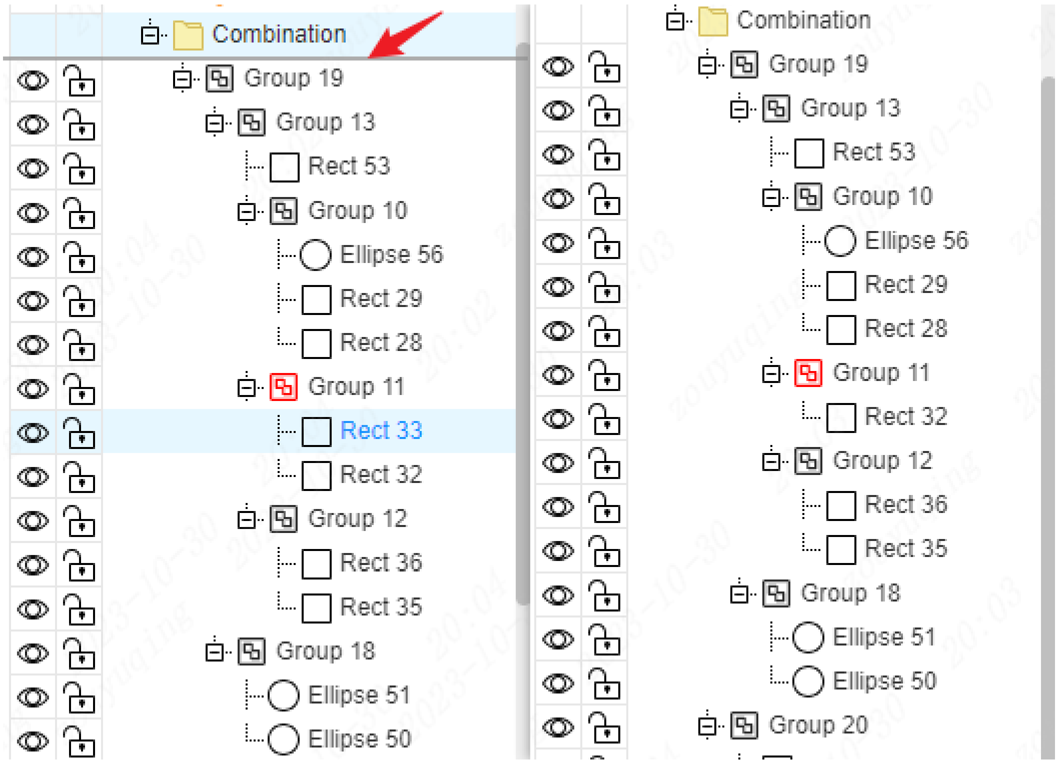
- Anchor positioning settings for layers and group have been added to the top of the object tree on the left. The layer is selected by default, that is, the object on the canvas will be displayed first in the layer object tree.
- Switch the selected group to quickly locate the group object tree section.

Drag adjustment

- Support directly dragging the object adjustment group relationship of the group object tree part. Sub-group can also be dragged and adjusted as a whole.

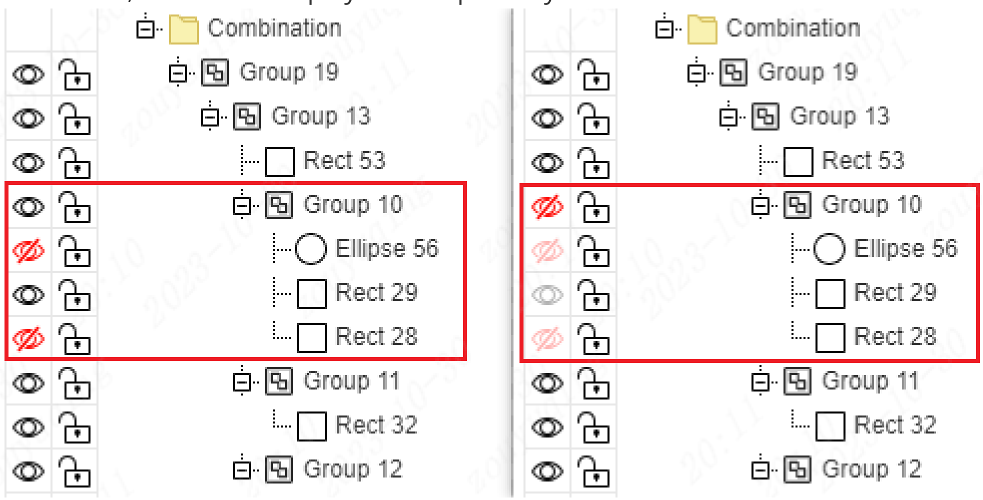


- Dragging to the top of the composite object tree will remove the composite object tree.

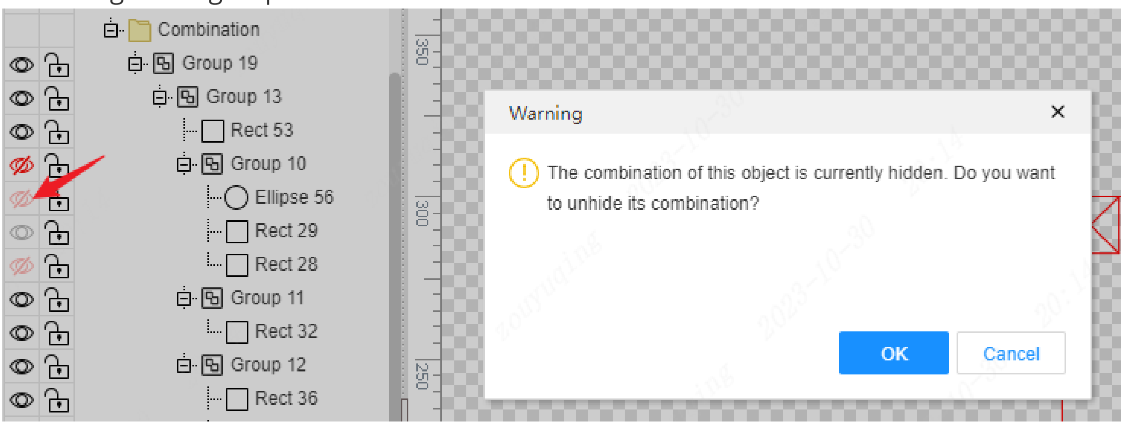


Hidden group

- The operation group is hidden, and the objects inside the group are not visible on the canvas. The icon style in front of the objects inside the group in the object tree on the left will remain, but will be displayed transparently.



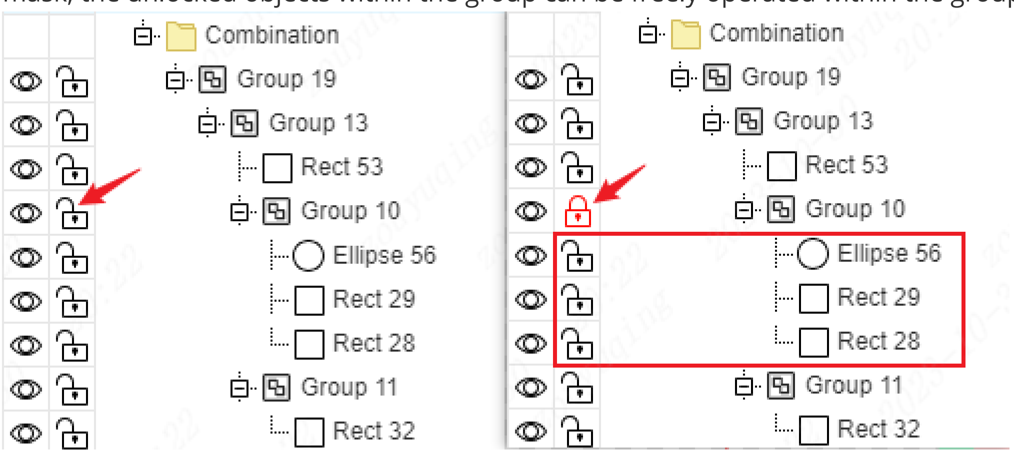
- Click the Hide Show icon in transparent display, and a pop-up window will prompt the current hidden status of the group where the object is located, and ask if you want to cancel the hiding of the group it is in.



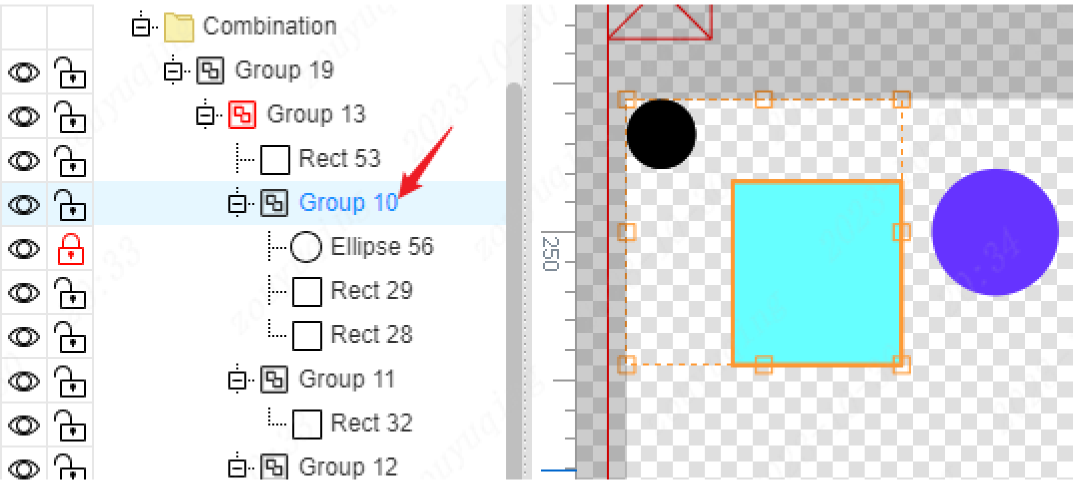
Note: When hiding in a group, if you want to switch the hidden display state of the objects in the group, you must first cancel the hiding of the objects in the group.

Locking group

- Operation group locking only locks the group itself, and does not batch lock the objects within the group. That is, the group as a whole is locked on the canvas, but under the group mask, the unlocked objects within the group can be freely operated within the group.



- If the object in the operation group is locked, it can only be locked within the minimum group where it is located. When its minimum group is not locked, the entire group can be moved, and it cannot operate the locked group object under the group mask.



Simply put, the locking of objects within a group is only locked within a certain level of group.

Other group rules

- The name of the group is assigned by default when it is created, allowing customization, allowing nullness, and allowing duplication.
- Combining in the panel library and placing it on the panel canvas also maintains the group property.
- It does not support multiple selection of objects within and outside the group at the same time across group
- Allow the existence of empty group, double-clicking on the empty group will also enter the mask mode, and the drawing element will be automatically added to the group.

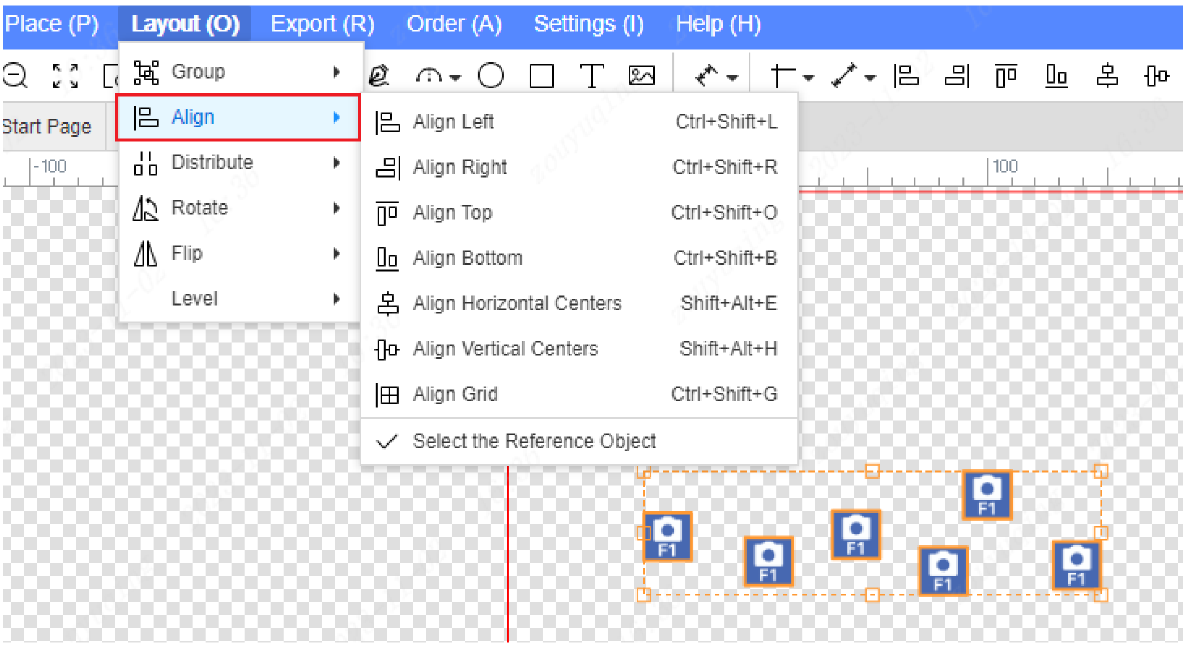
Align

Align

The alignment function of the panel is consistent with the schematic diagram, the operation mode to reuse tiles, symbols, packaging, and PCB.

Method of alignment operation:

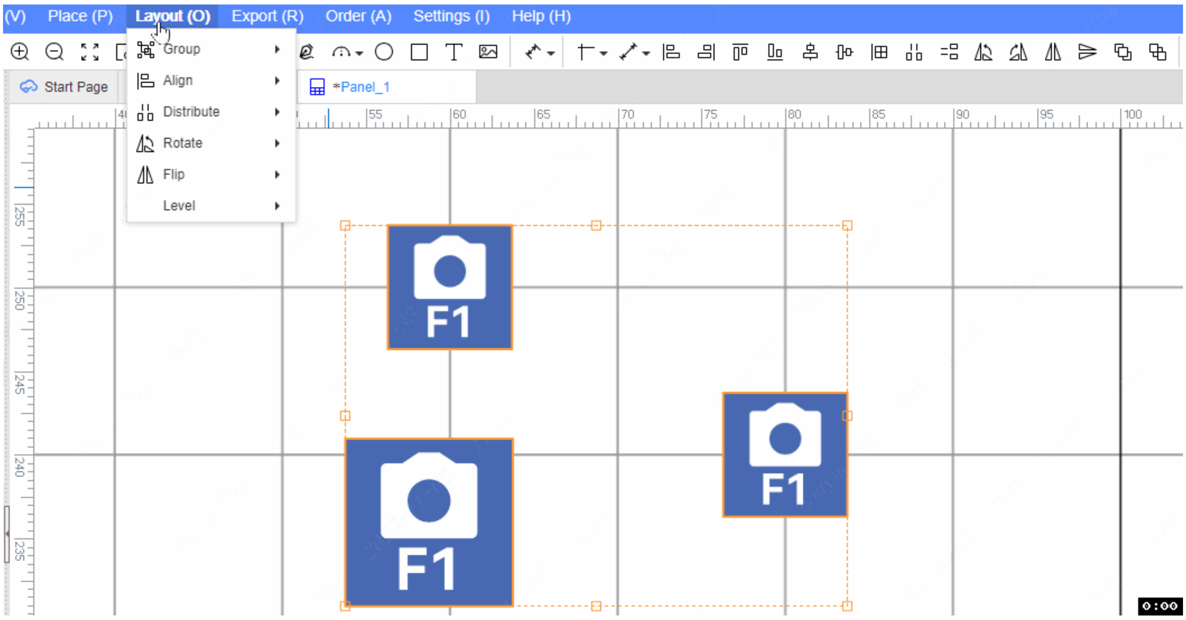
- Select the primitive that needs to be aligned - top menu - layout - alignment

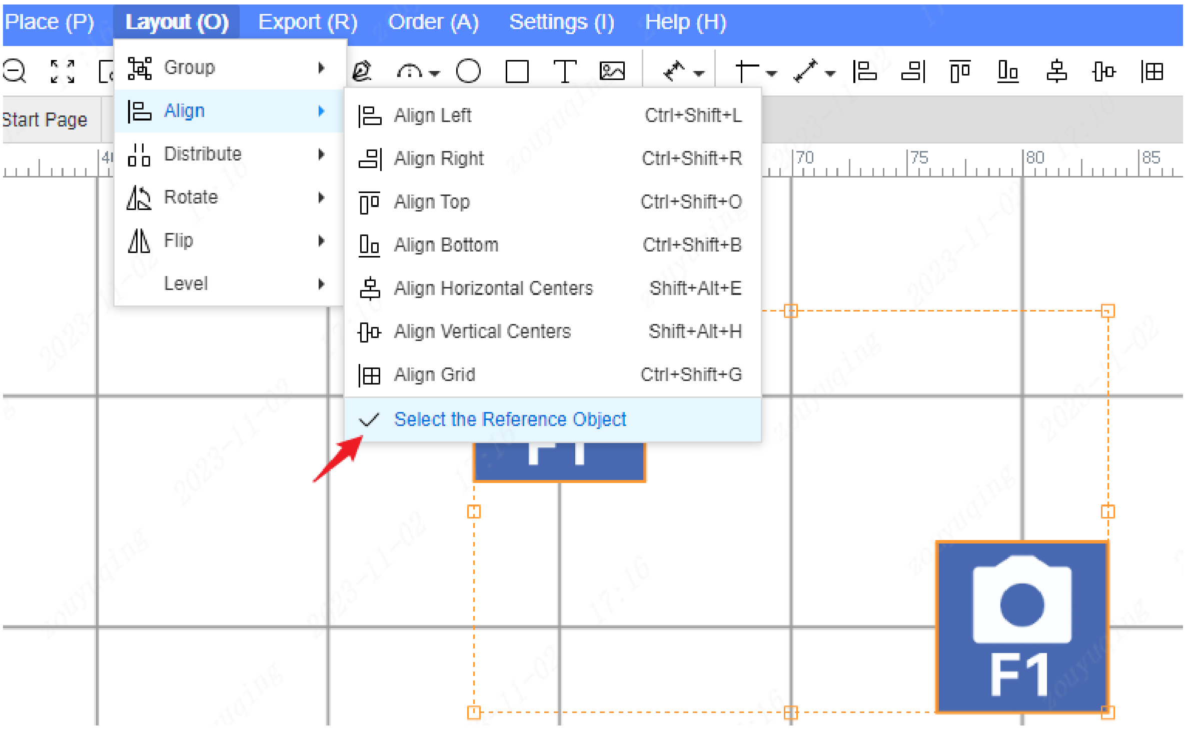


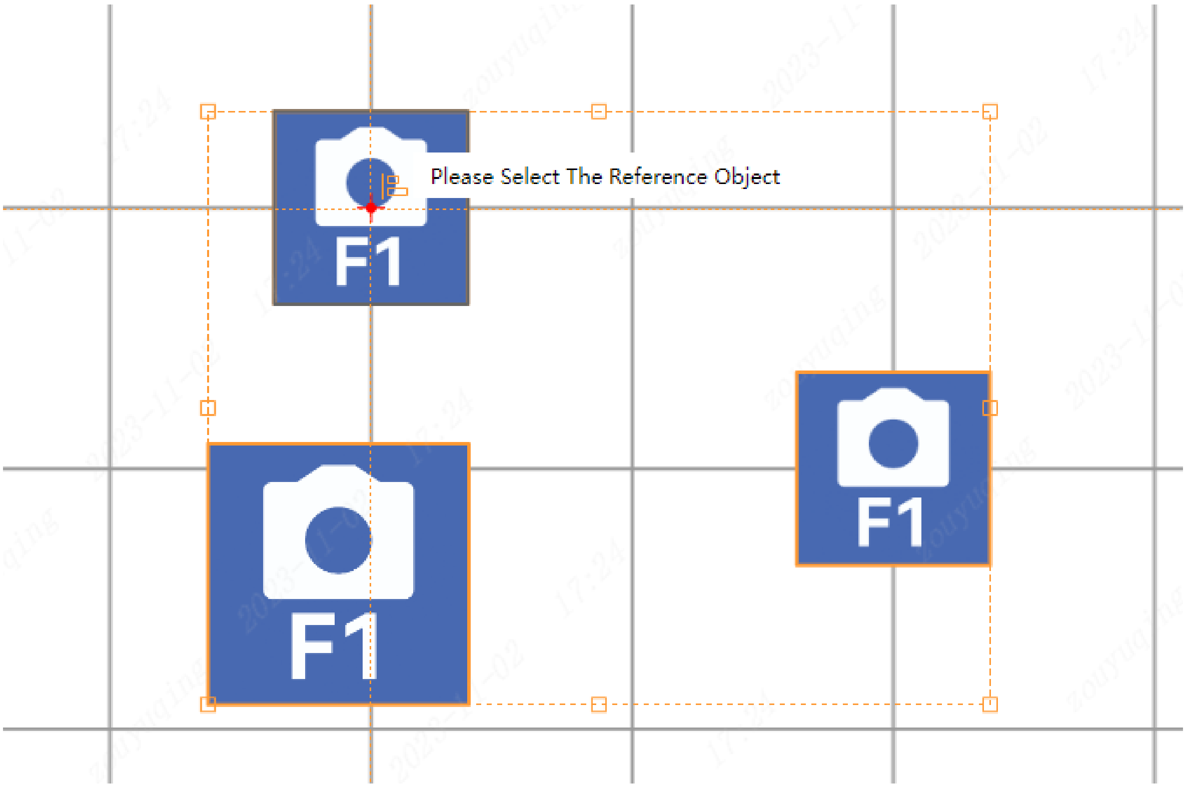
Select the Reference Object

Support custom alignment function to select reference objects.

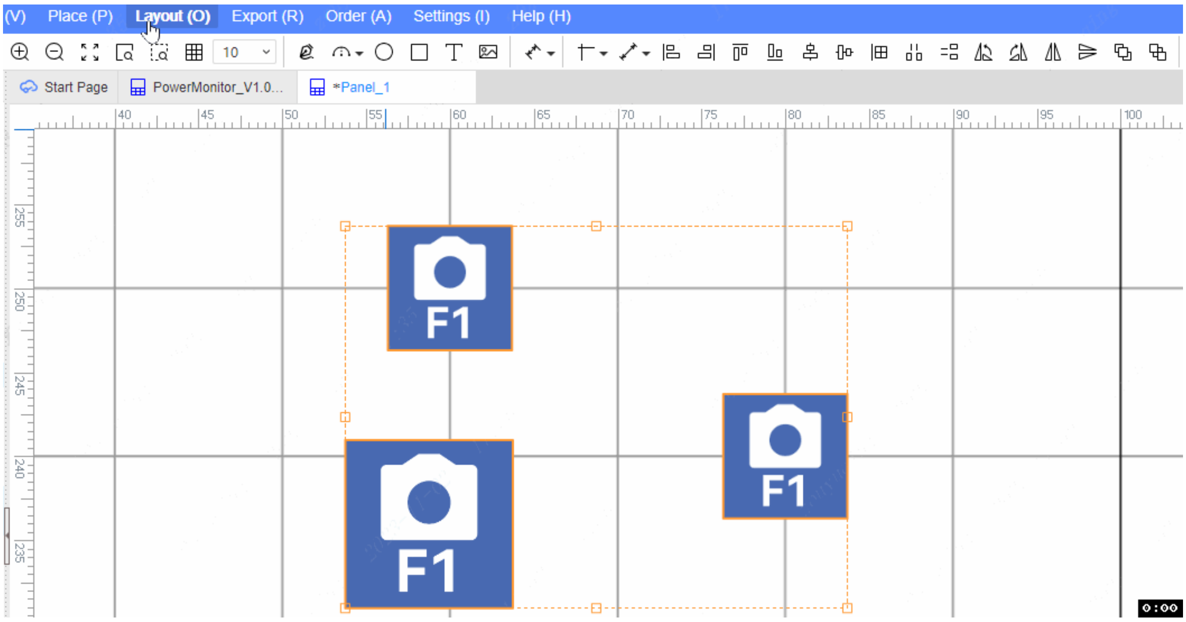
When checking the "Select Reference Object" menu, the operation alignment needs to select the reference object by yourself.

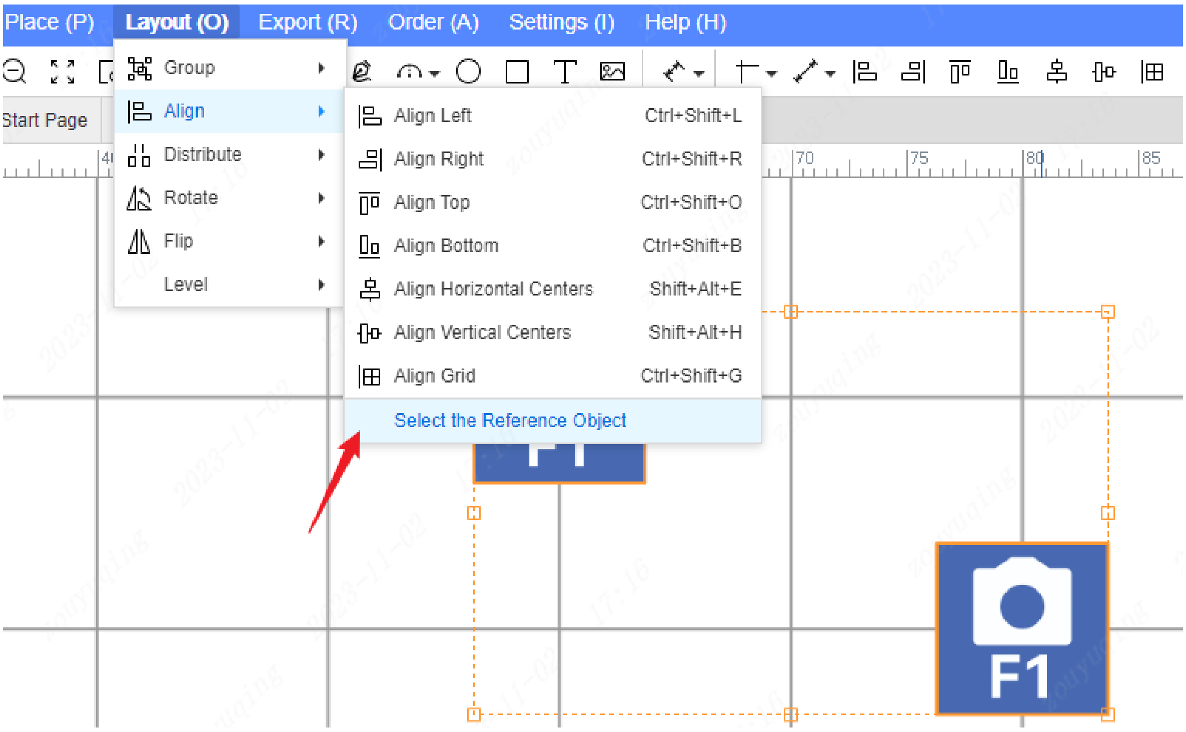






When the "Select Reference Object" menu is not checked, the operation alignment is performed by default.

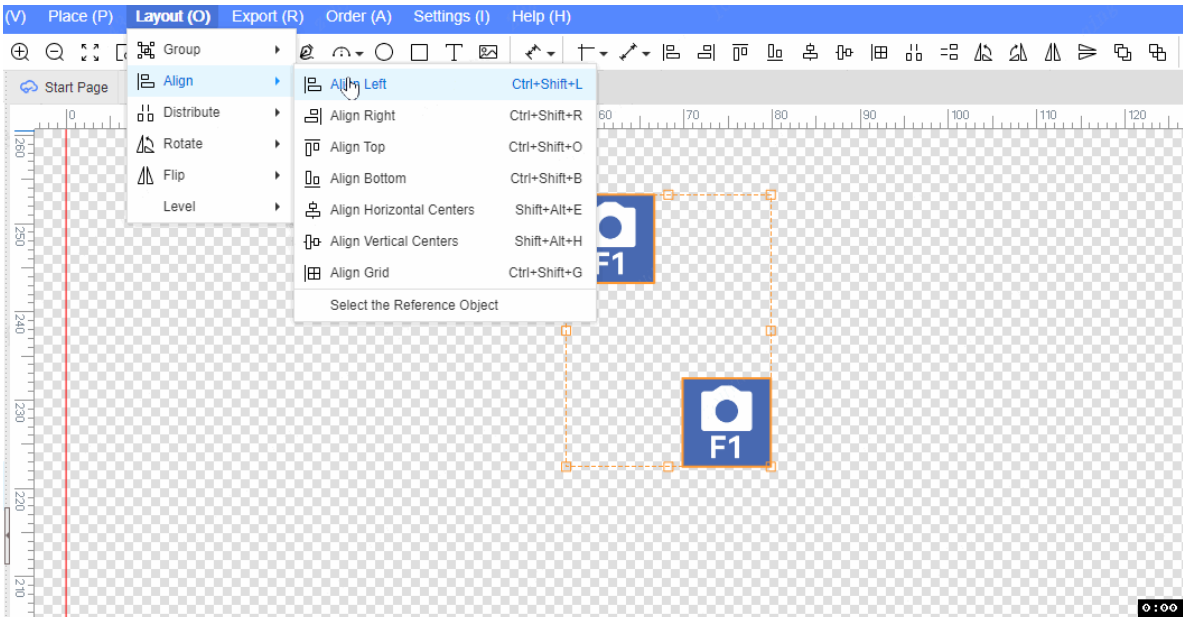


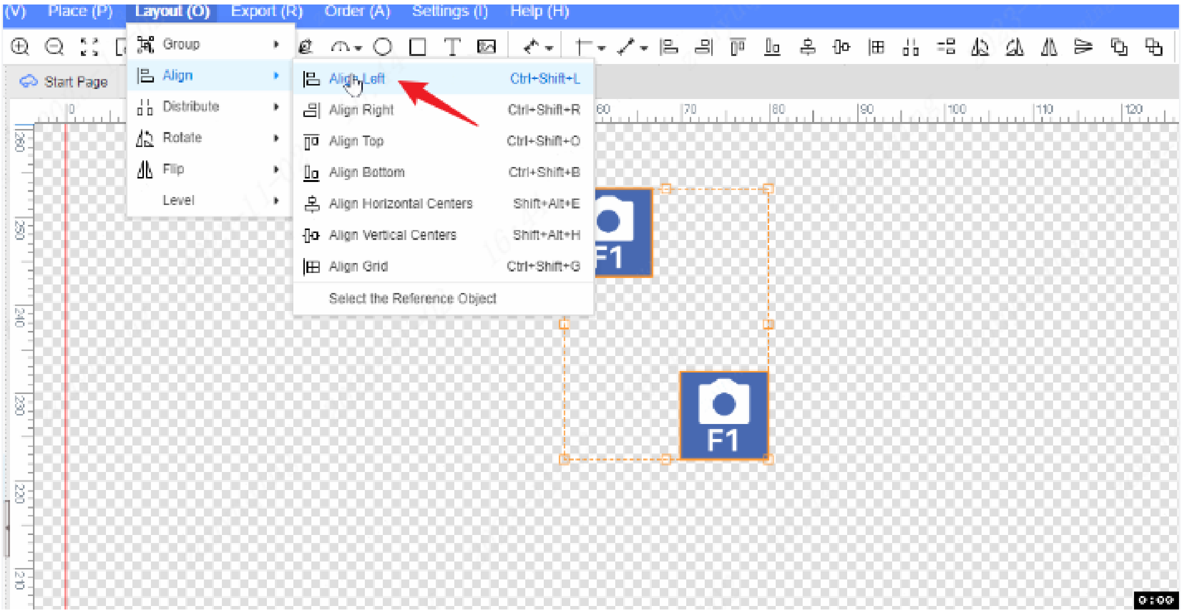


Align Left

Select the elements that need to be aligned. You need to select two or more elements. If each element has no reference point, it cannot be aligned.

- Top Menu - Layout - Align - Align Left
- Shortcut: Ctrl + Shift + L

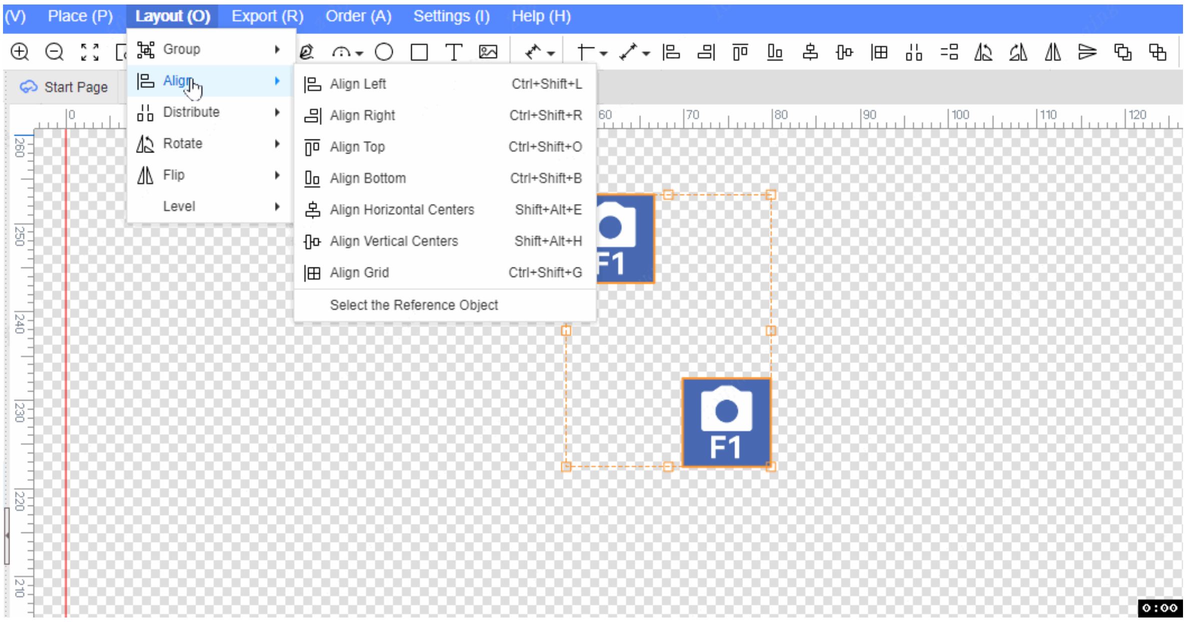


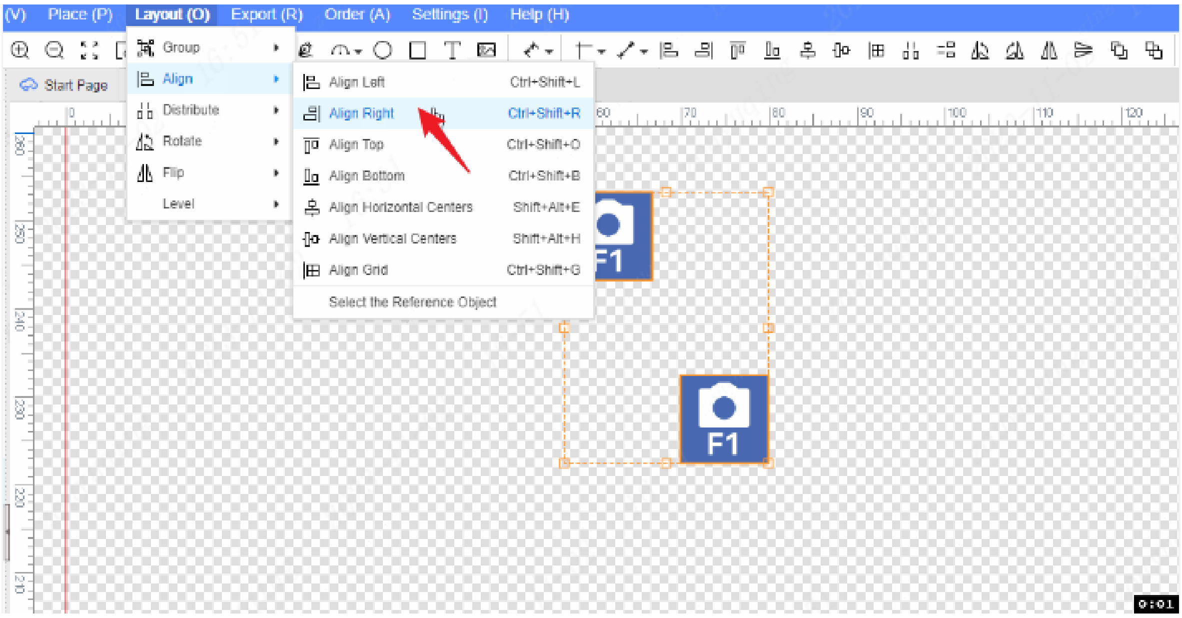


Align Right

Select the elements that need to be aligned.

- Top Menu - Layout - Align - Align Right
- Shortcut: ctrl + shift + R

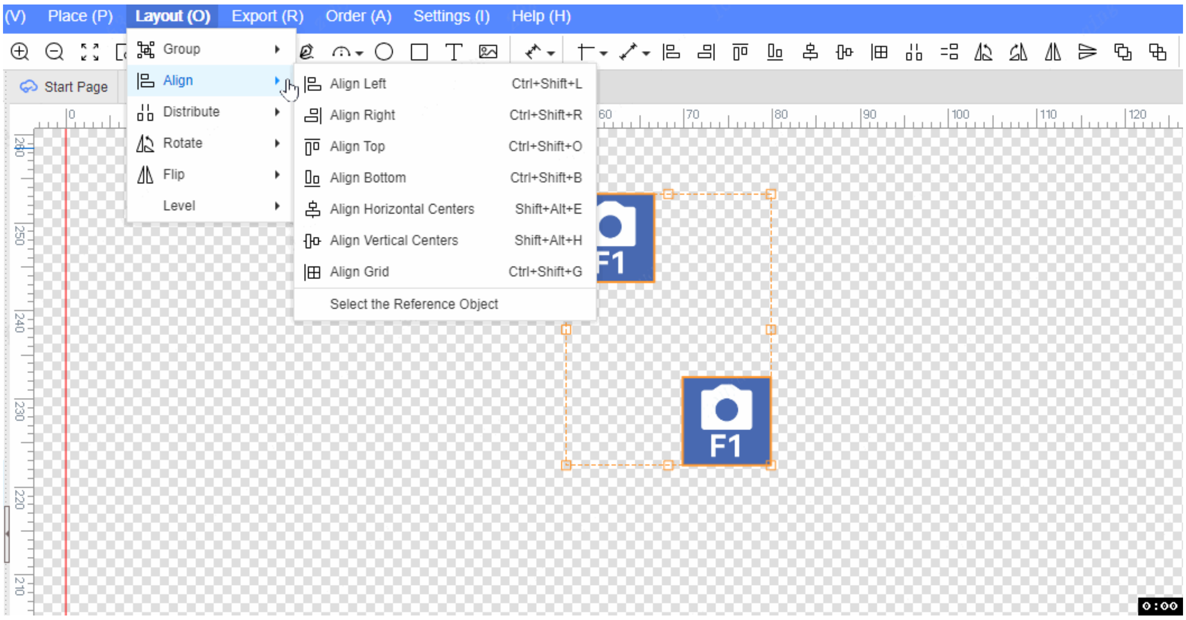


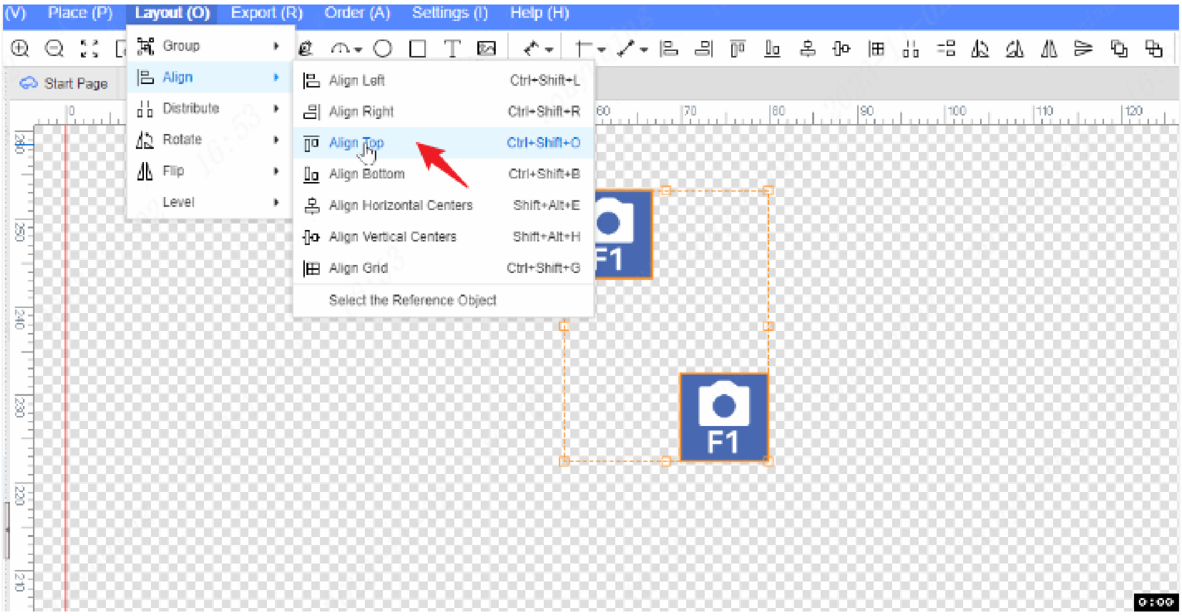


Align Top

Select the elements that need to be aligned.

- Top Menu - Layout - Align - Align Top
- Shortcut: `Ctrl`+`Shift`+`O`

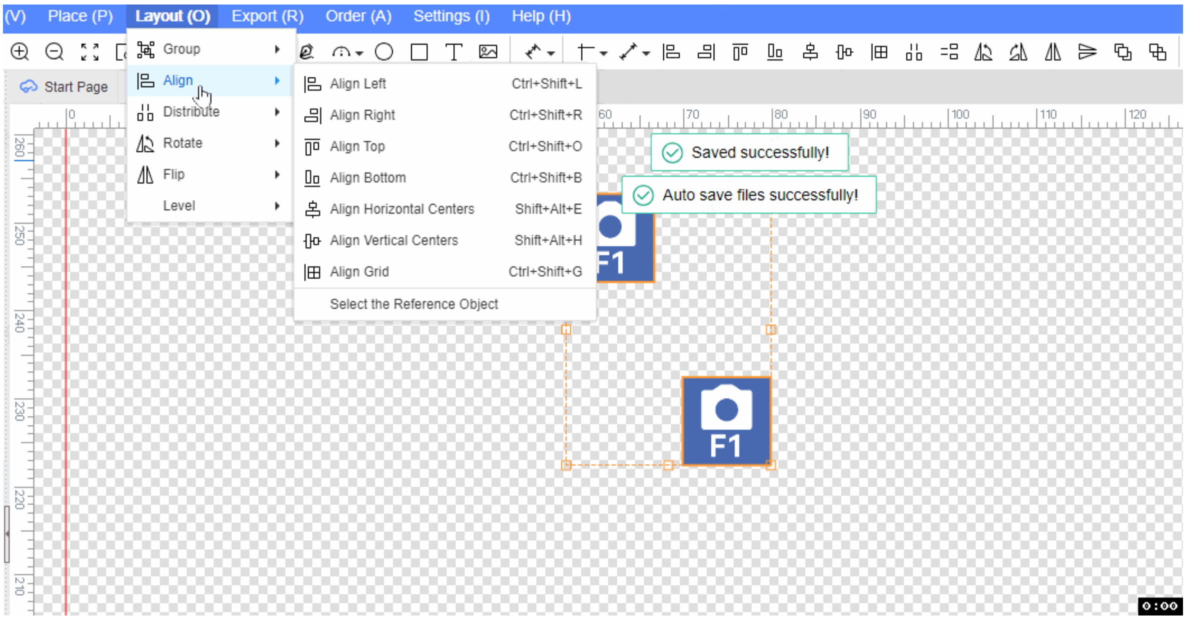


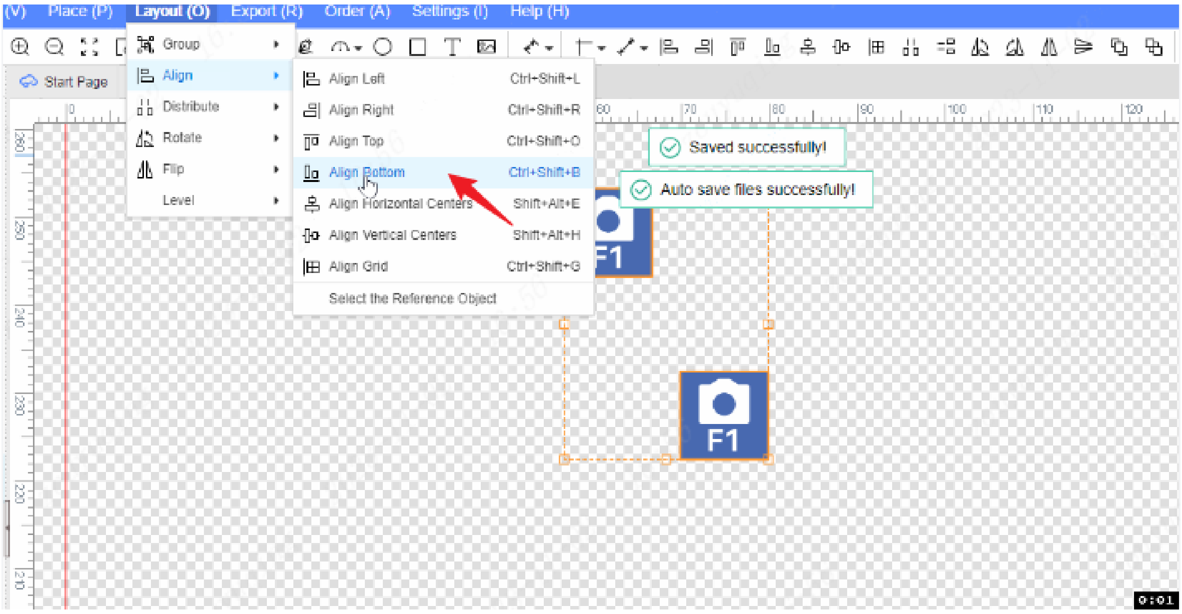


Align Bottom

Select the elements that need to be aligned.

- Top Menu - Layout - Align - Align Bottom
- Shortcut： Ctrl + Shift + B

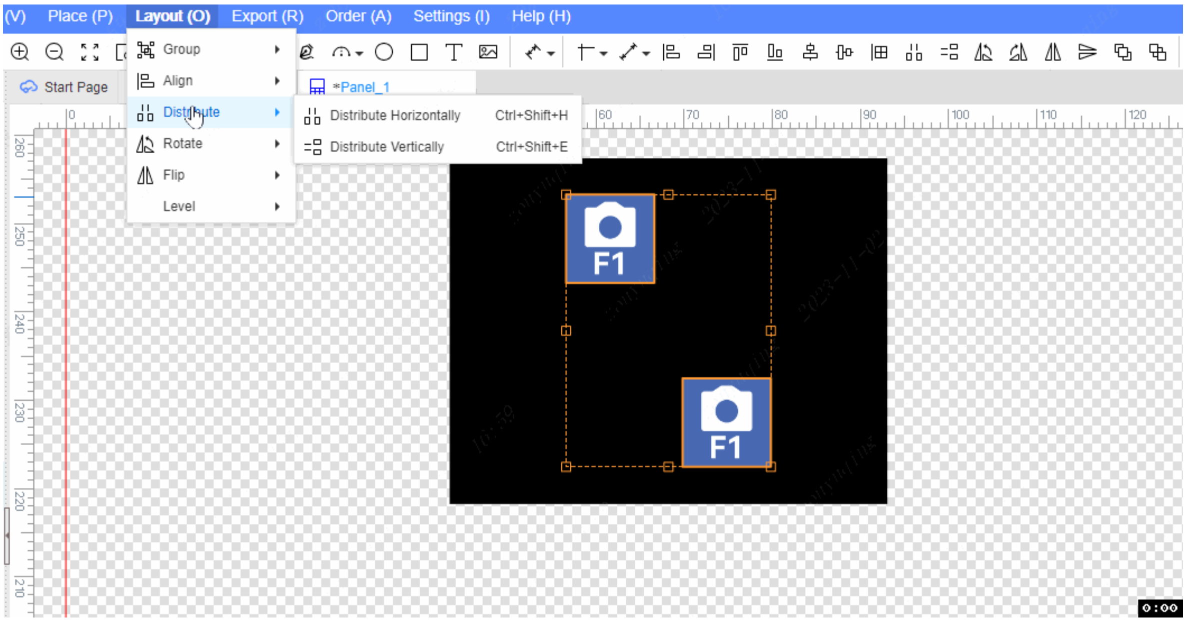


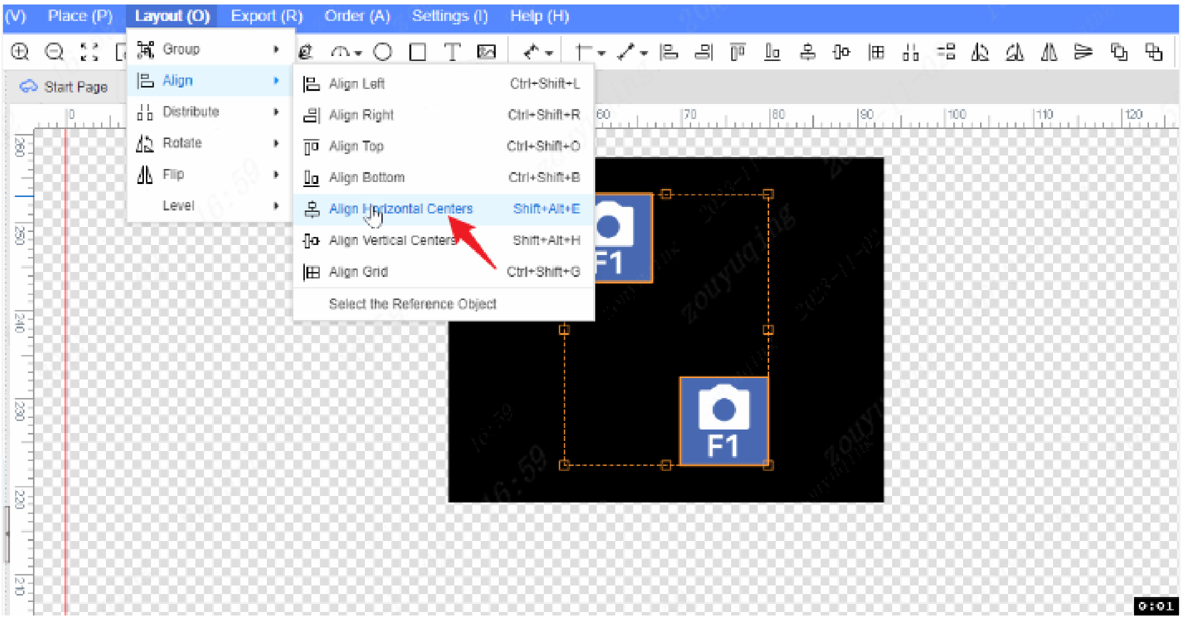


Align Horizontal Centers

Select the elements that need to be aligned.

- Top Menu - Layout - Align - Align Horizontal Centers
- Shortcut: Ctrl + Shift + E

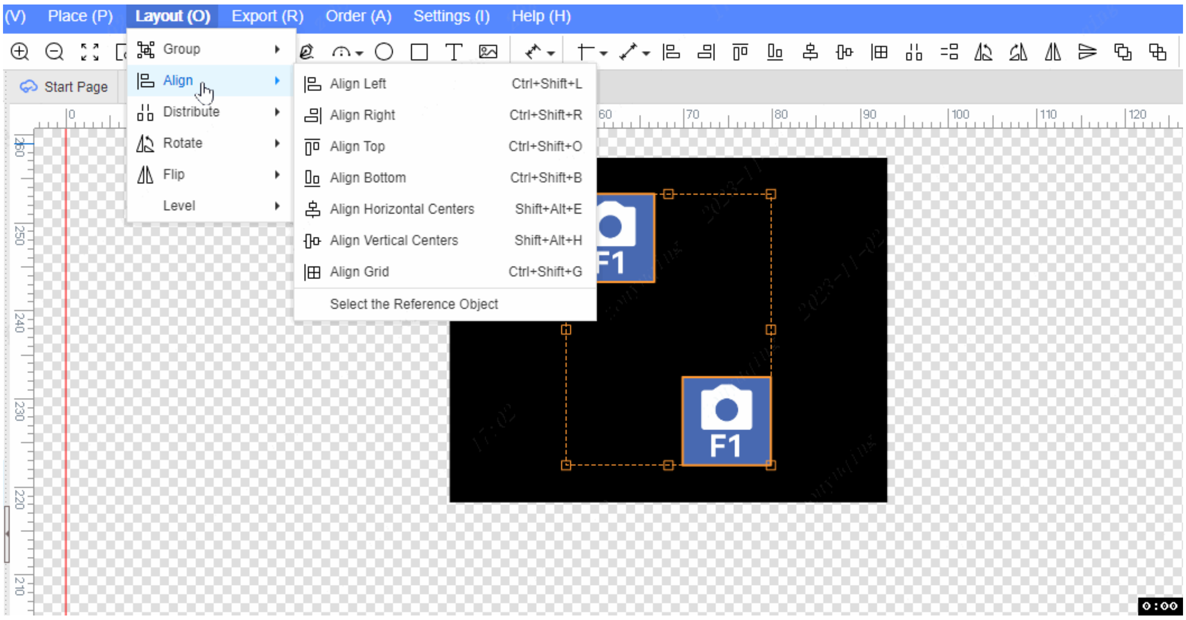


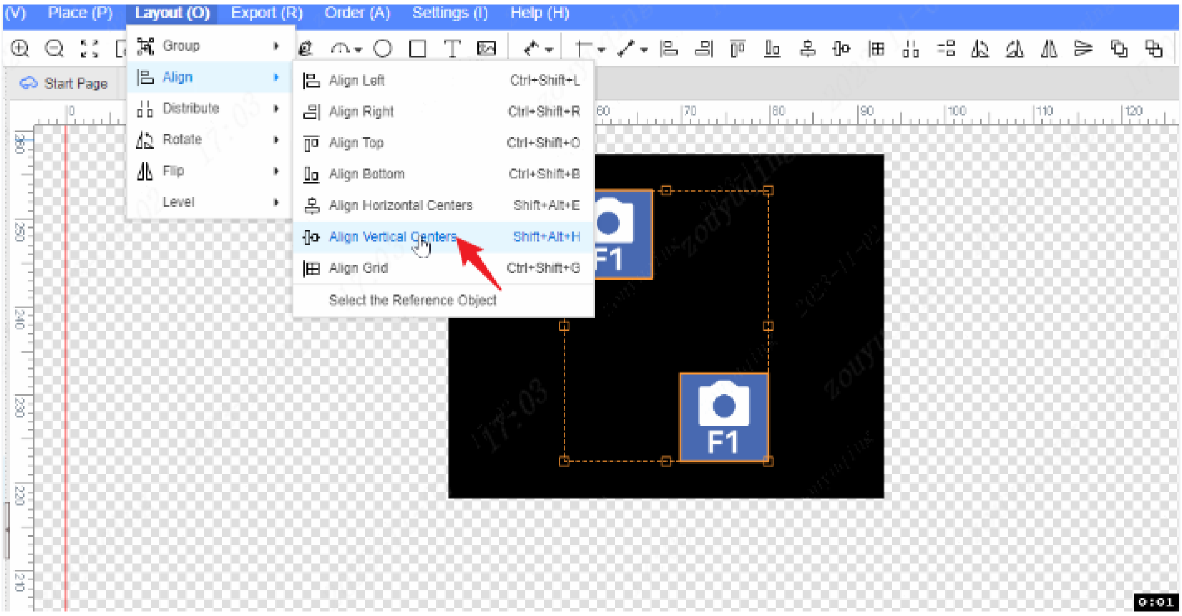


Align Vertical Centers

Select the elements that need to be aligned.

- Top Menu - Layout - Align - Align Vertical Centers
- Shortcut： `Ctrl` + `Shift` + `H`

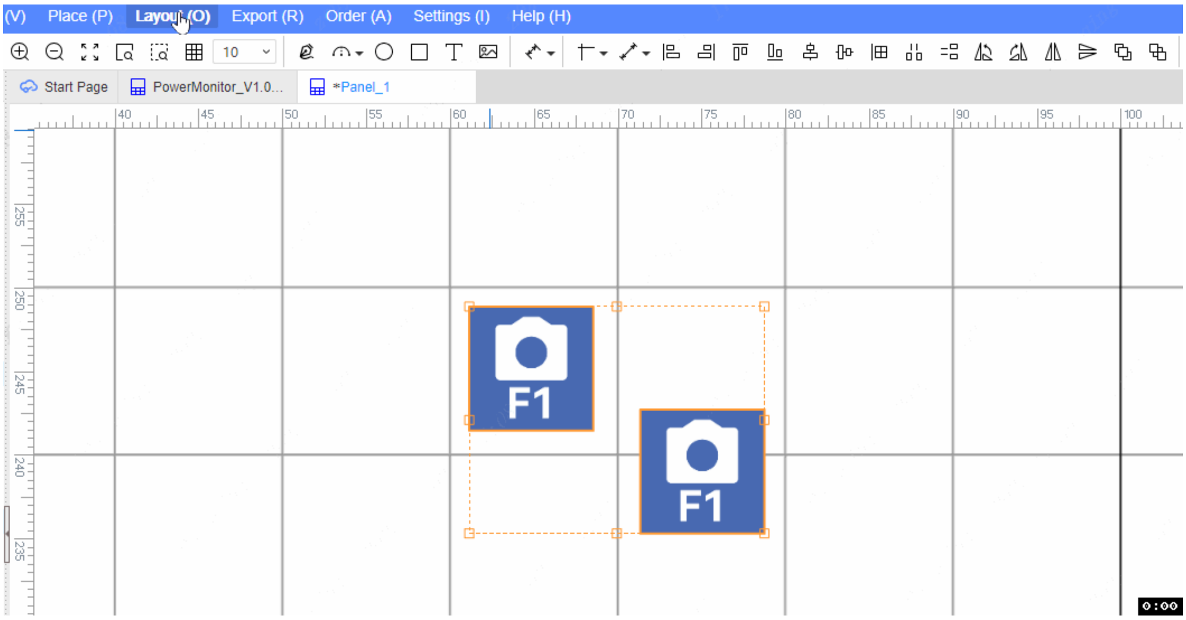


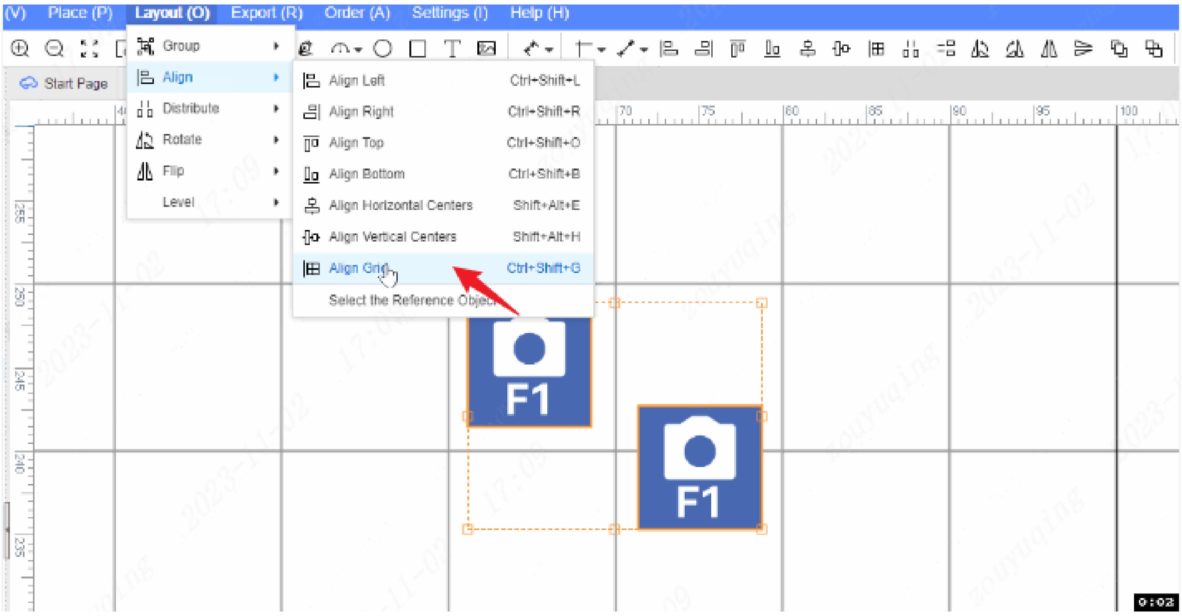


Align Grid

Select the elements that need to be aligned and align the selected elements with the grid of the panel editor.

- Top Menu - Layout - Align - Align Grid
- Shortcut: `Ctrl` + `Shift` + `G`



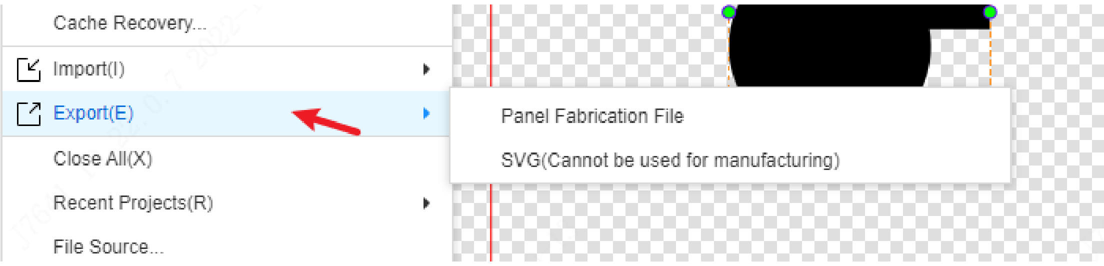


Export Manufacturing File

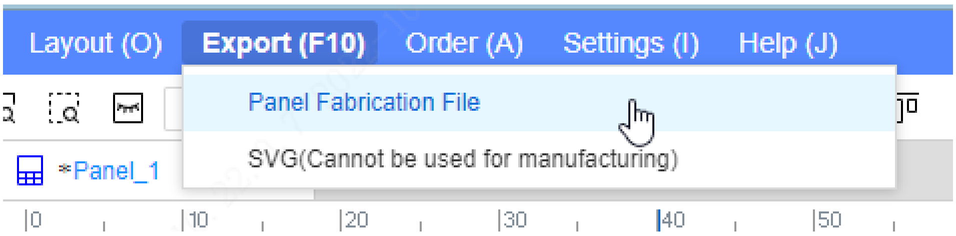
EasyEDA Pro panel editor supports one-click export of .epanm panel manufacturing files, which is more time-saving than SVG.

Export steps:

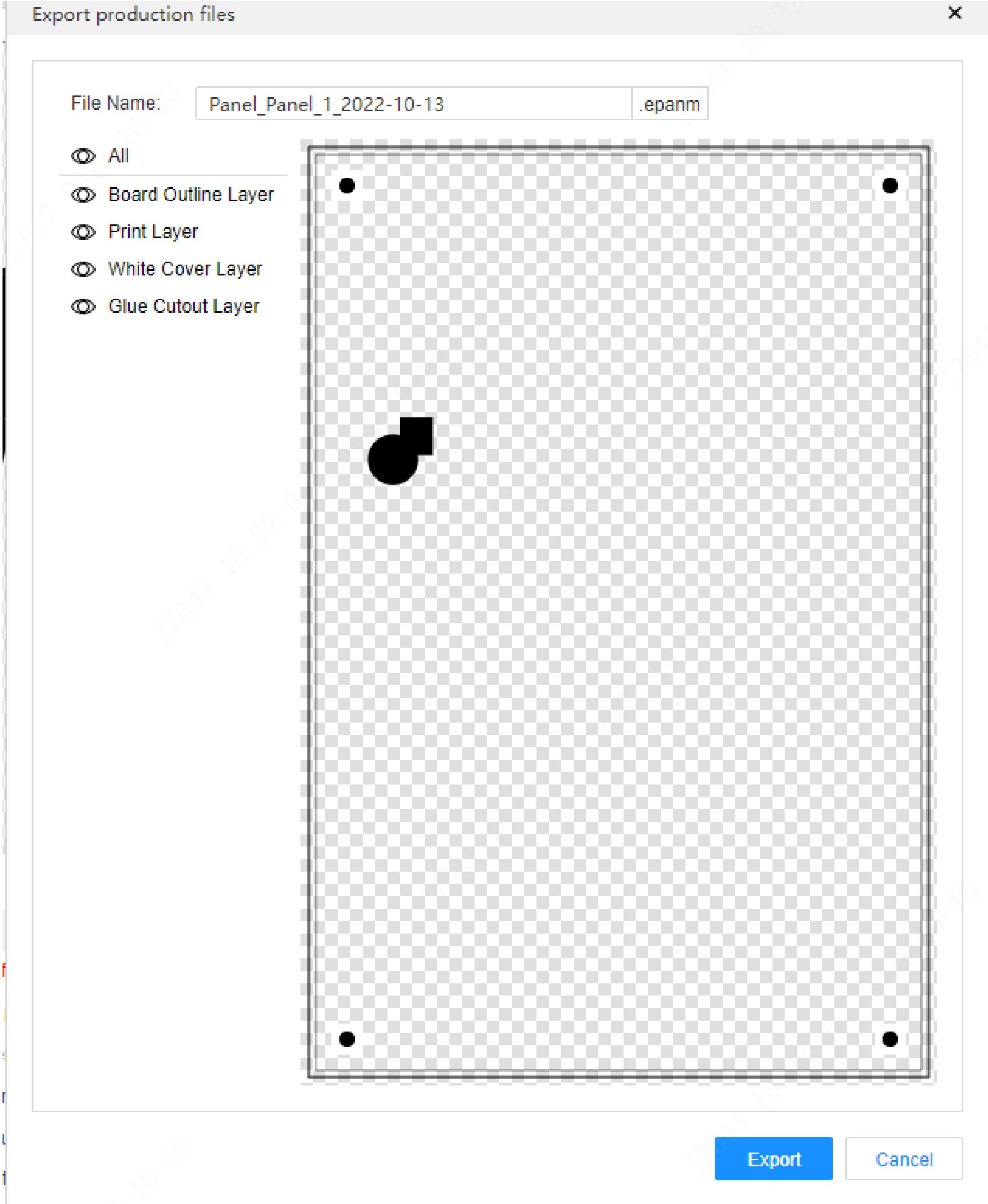
- Top Menu - File - Export - Panel Fabrication File



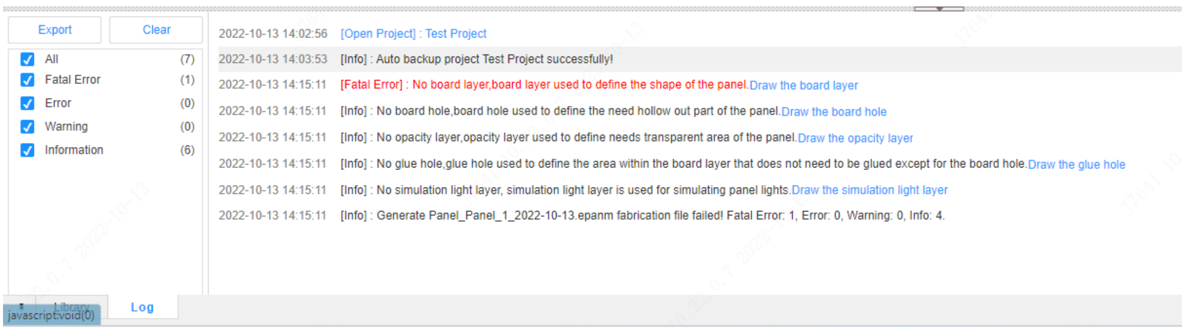
or at Top Menu - Export



you can preview the design before export



If your panel file miss some layer object, when exporting the log will list the error

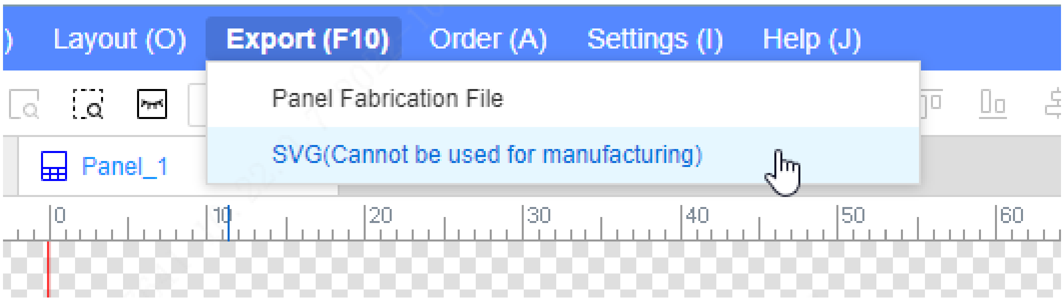


Note: Currently, it is not supported to place orders directly from .epanm files. You need to compress the exported .epanm files into RAR format before uploading them to LCSC Mall for ordering. One-click ordering will be supported in the future.

LCSC doesn't support to accept the panel order yet, please wait a while.

Export SVG

EasyEDA Pro supports exporting SVG files for ordering, but it should be noted that SVG ordering needs to note the file description in the design drawing. It is recommended to export the panel file to place the order.



Column as:

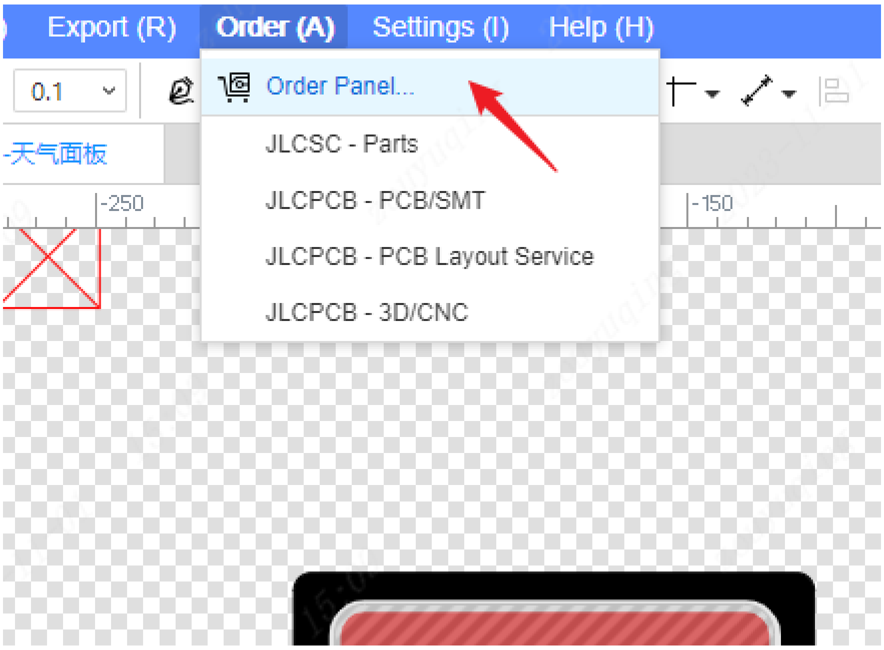
Go to LCSC Mall to place an order: [Panel Printing]

Order Panel

EasyEDA Pro panel editor supports one-click single panel fabrication files.

Export steps:

- Top Menu - File - Export - Order Panel



After clicking, the order file will be generated according to the current panel file. After the generation, click the OK button in the pop-up window to go to the LCSC Mall to continue ordering and payment.

LCSC doesn't support to accept the panel order yet, please wait a while.

Import/Export

Import EasyEDA Standard Edition

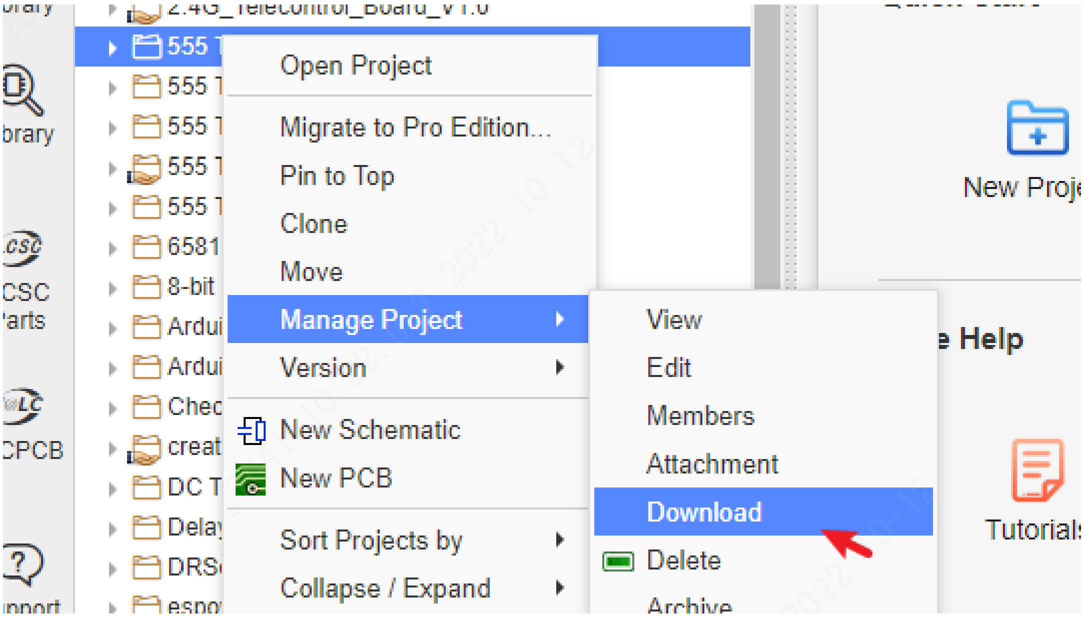
At present, EasyEDA only supports importing and importing from the standard version of the project, and does not support the professional version to export the standard version format.

Notice:

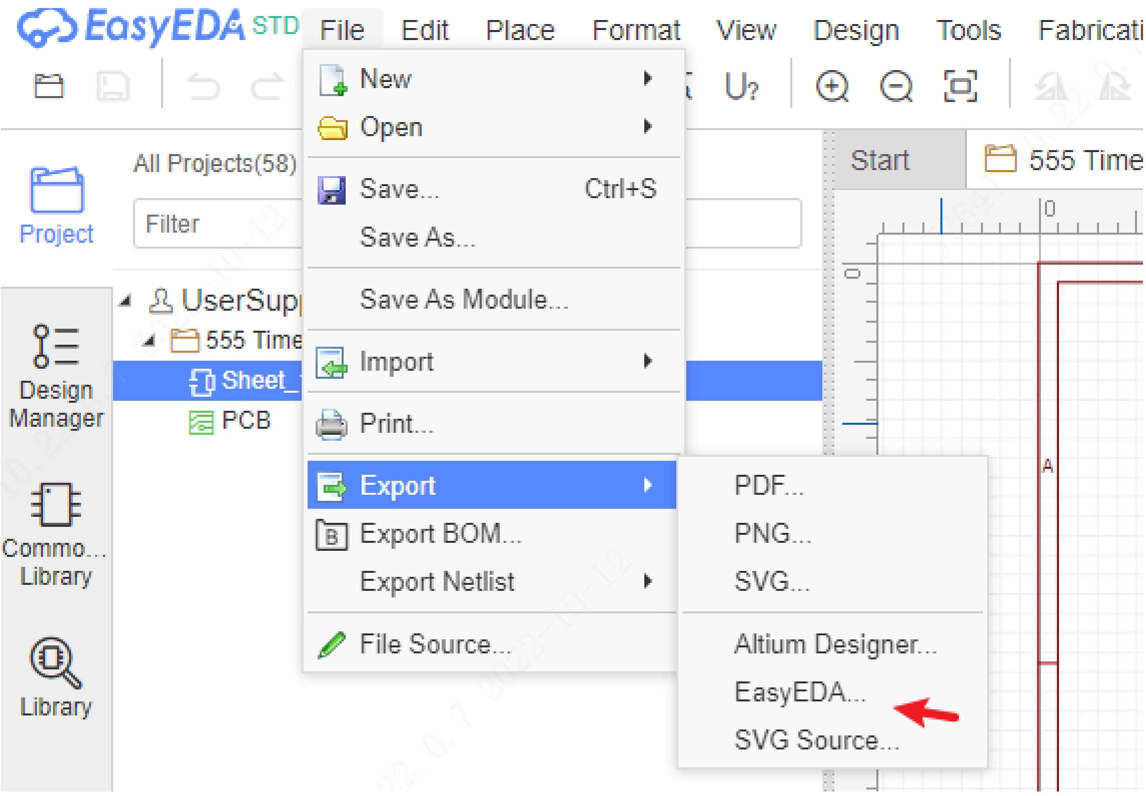
- **Due to the inconsistency of the format and graphic element design, there may be some differences after the graphic element is imported, please check carefully. For specific differences, please refer to the help documentation.**
- **EasyEDA is not responsible for any losses caused by format conversion differences, if you do not agree, please do not import.**

Download file to import

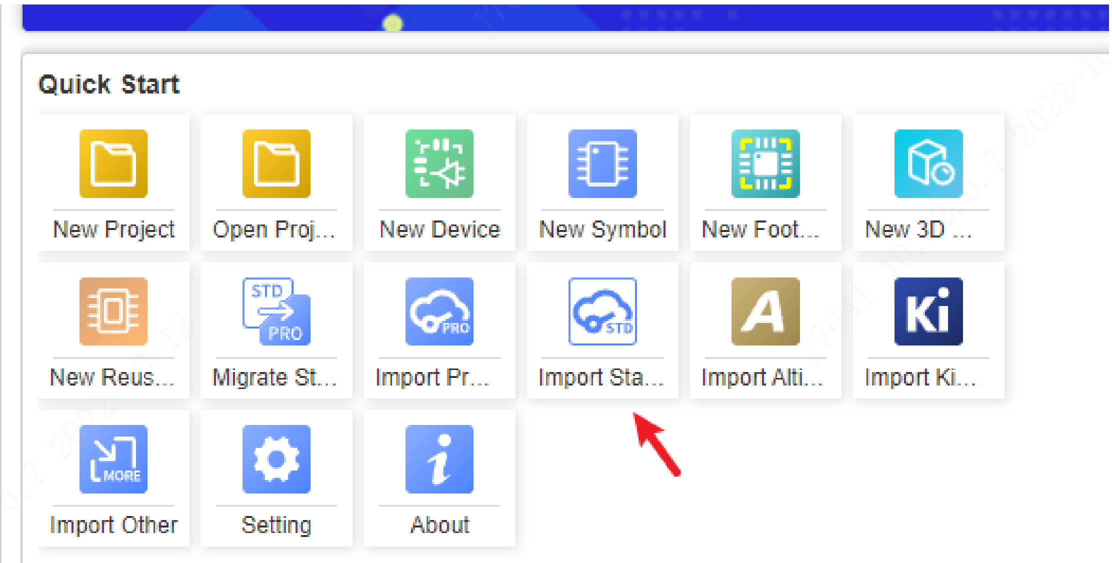
1. Download the standard version of the project
   You can right-click in the project list of the standard version editor, and download the project. The workspace page will be opened, and a compressed footprint will be automatically downloaded.



or after opened the document, export the EasyEDA file format, export schematic and PCB, and then compress as a zip file.



2. If you download the project, you need to unzip the zip file to get the inner project zip file, and import it.



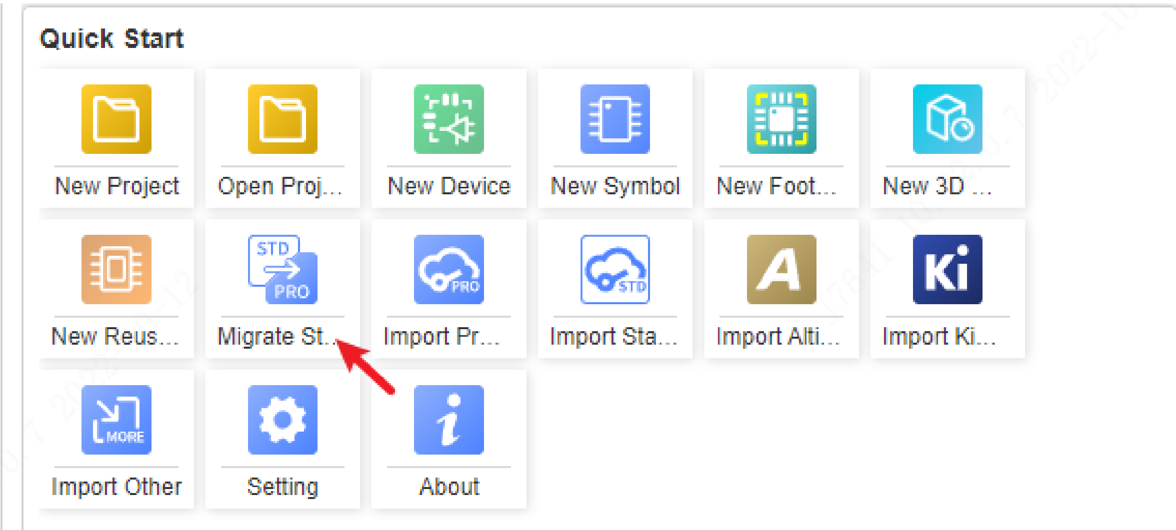
3. You can also import the EasyEDA format json file not the zip file

Note: If there is a PCB, please be sure to compress the schematic and PCB together and import. Otherwise, importing the schematic separately requires all components to be re-bound and footprintd.

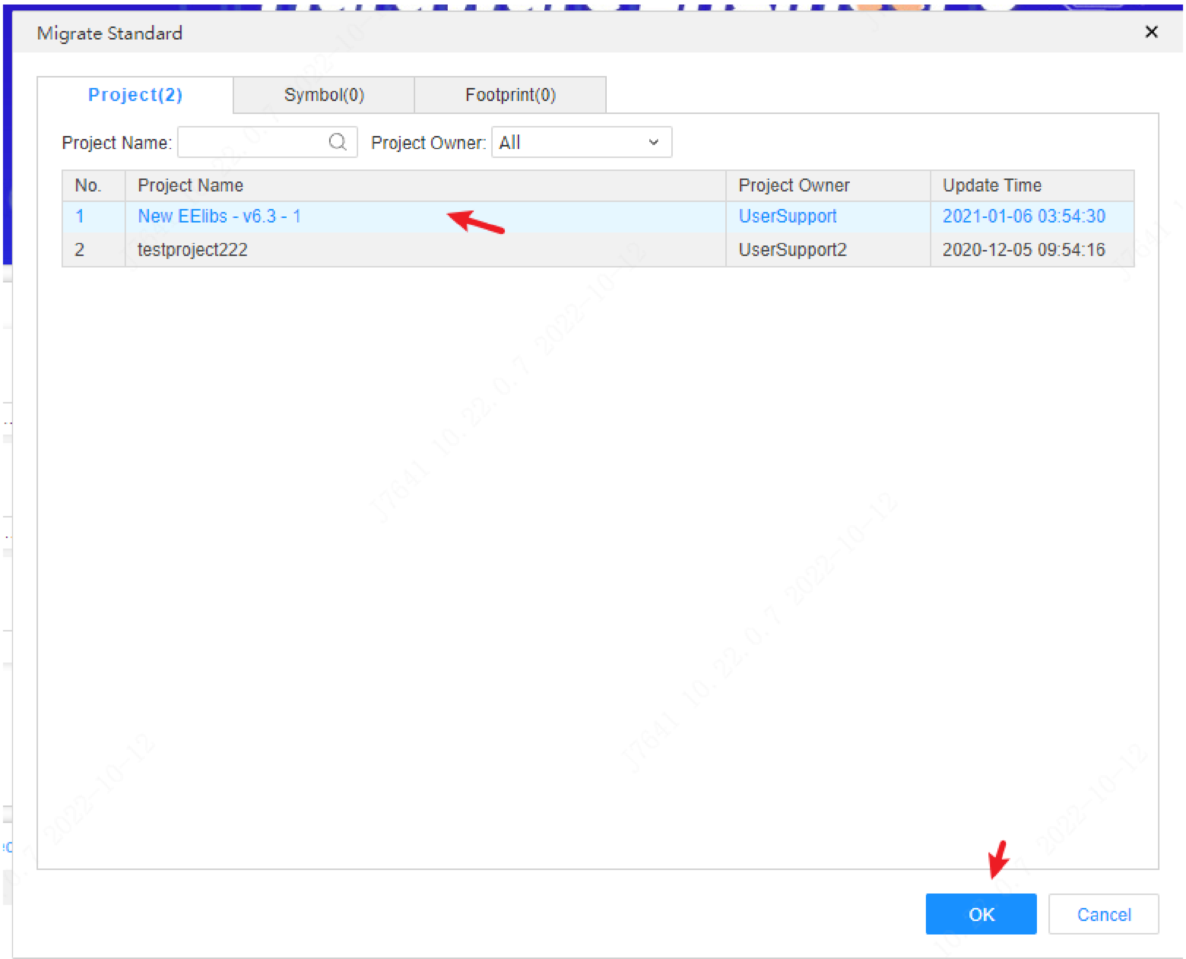
Migration to import

Directly on the Professional Edition start page, use the migration function to import.

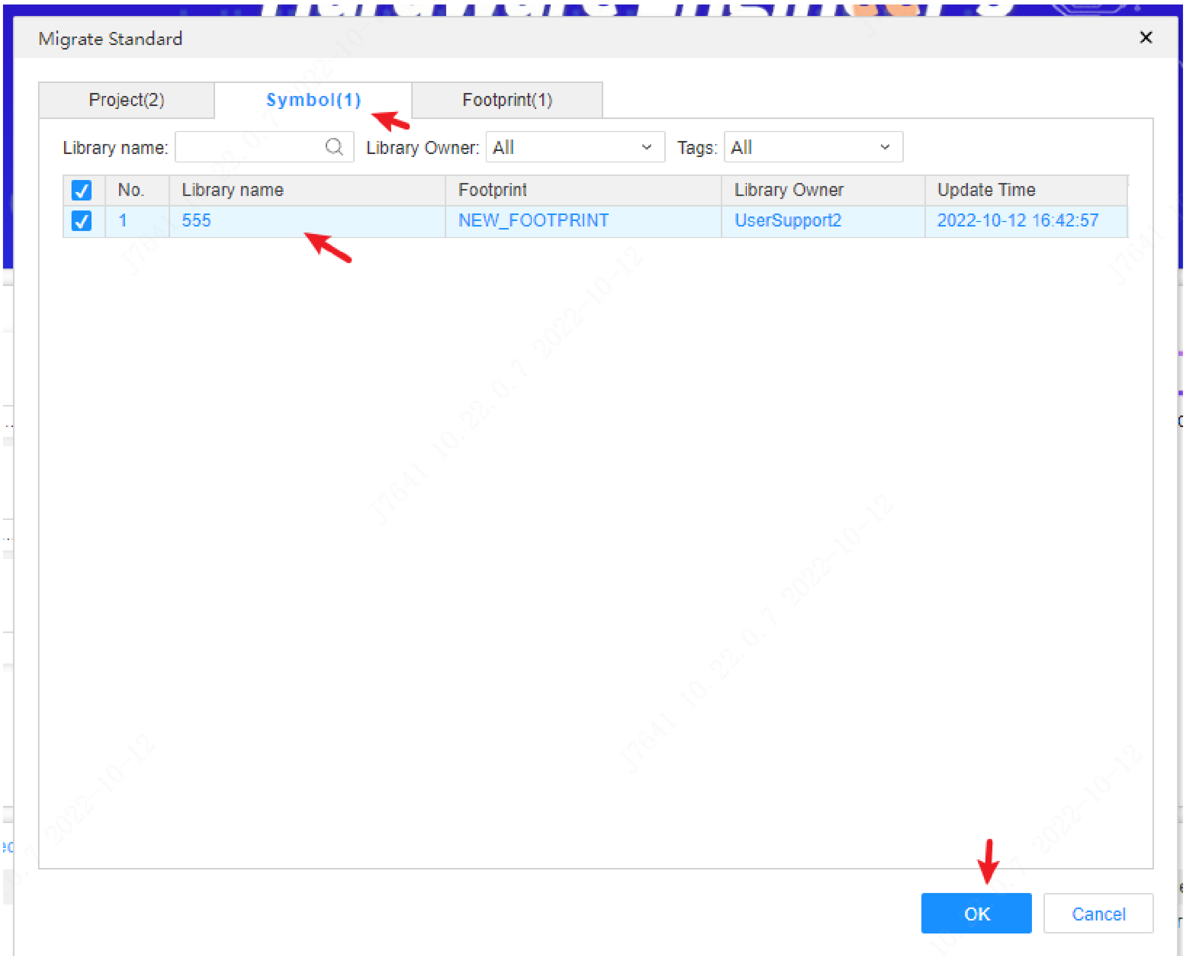
1. Open the Standard Edition editor in the browser and log in. If you do not log in, you will not be able to get the standard version of the data.
2. Open the professional version editor in the browser, on the start page, click "Migrate Standard Edition"

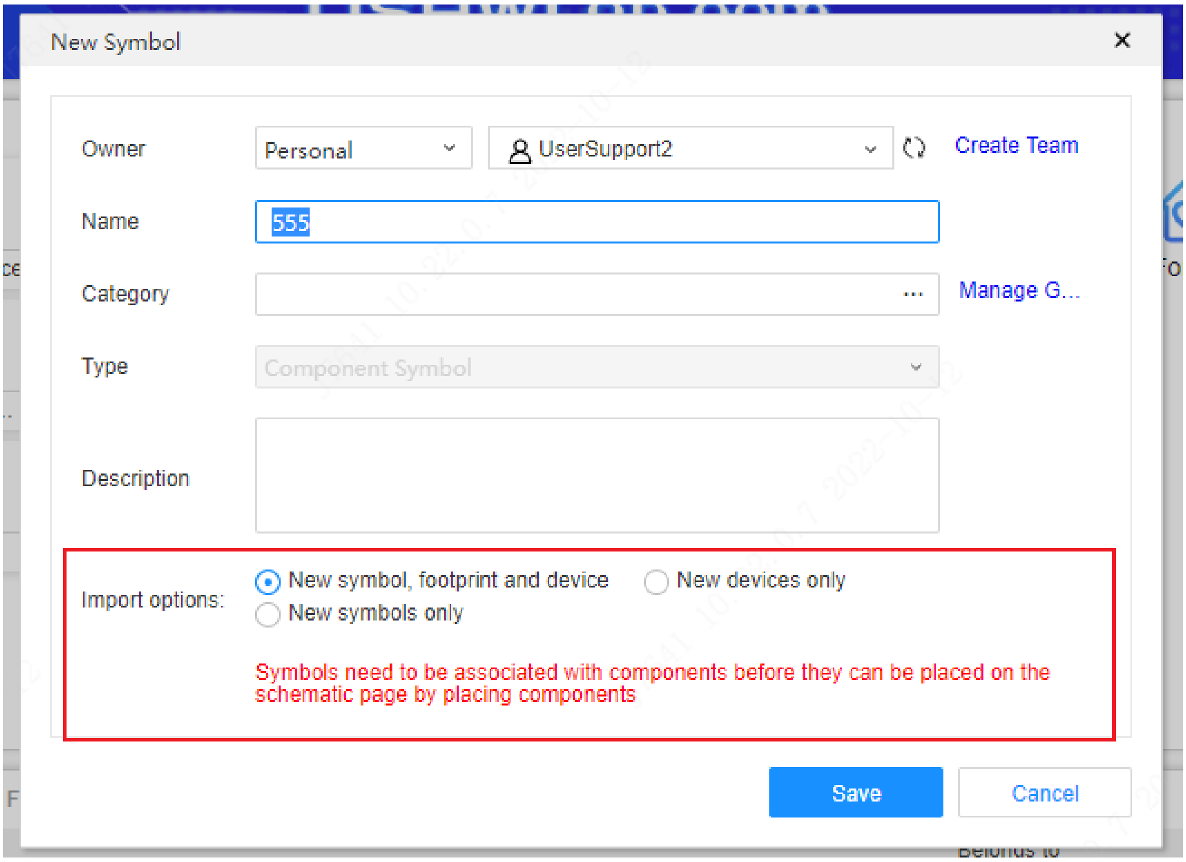


3. Select the project or library to be migrated and import it.



If it is necessary to import symbols and device libraries at the same time, when using migration, only select symbols for migration, and the corresponding device library, symbol library, and import symbol binding footprint will be automatically created according to the symbol library, and automatically associated with the device.





If only the footprint library is imported, the device library will not be automatically created, only the footprint library will be imported separately.

common problem

common problem

Please check carefully for other possible differences. If you find a large difference, please contact us to see if it can be optimized or repaired.

The difference before and after format conversion is as follows:

Schematic / component library:

Elements/Layers	After Import	Notes
Bus/bus branch	Import is not supported	The design of the bus and bus branch of the professional version is different from that of the standard version, and cannot be directly imported and used

Elements/Layers	After Import	Notes
Text	Import as normal text	The professional version does not support comments at the moment. If the corresponding font is not installed in the operating system, the default font will be used after the text is imported, because the font of the text may have a slight offset in different positions; save the AD file as In ASCII, Chinese characters may be garbled, and the garbled characters will be automatically converted to underscores after import
Voltage Probe	Import as Network Logo	Professional version does not support voltage probe primitives
Non-connected logos	Free non-connected logos are not imported	Professional version non-connected logos can only be placed on pins, so free non-connected logos are not imported
Pins	The free pins of the schematic diagram are not imported	The professional version does not support placing pins separately in the schematic diagram
Bezier curve/ellipse/free drawing	Import as polyline	Professional version does not support Bezier curve, Bezier curve, ellipse arc, ellipse
Canvas origin	The default is the lower right corner of the drawing	If there is no drawing, it will be imported according to the original canvas origin
Arrows	Import is not supported	Arrows are not supported in Pro version
Grid size	Import in proportion	The grid of the standard version is pixels, and when the grid is 5, the professional version corresponds to 0.05inch
Pin type	Unsupported types are converted to input types	The professional version only supports three pin electrical types
Hidden pins	Show after import	Professional version does not support hidden pins

Elements/Layers	After Import	Notes
Component footprint	After importing, the "original package" attribute will be generated	The professional version does not associate the package with the package name, but with the uuid of the package, so when importing, the original associated package name will be added as a common attribute. If the imported schematic and PCB When compressing and importing together, the package will be automatically associated
Component Mirroring	Will be imported as different component templates	The standard version of the component has not been recorded in the file whether it has been mirrored (flipped), so when importing the professional version, no matter whether the mirrored or not mirrored component will be imported as a different template library (in The project library at the bottom can see two devices with the same name)
Theme style	The default theme of the professional version is used in the import dialog box	The default theme is that the rounded rectangle is converted to a right angle, the fill color is not preserved, and the color of the primitive is not preserved, so as to switch the schematic theme, and the theme of the original file follows the original file Style, color does not change when switching themes

PCB/Package Library:

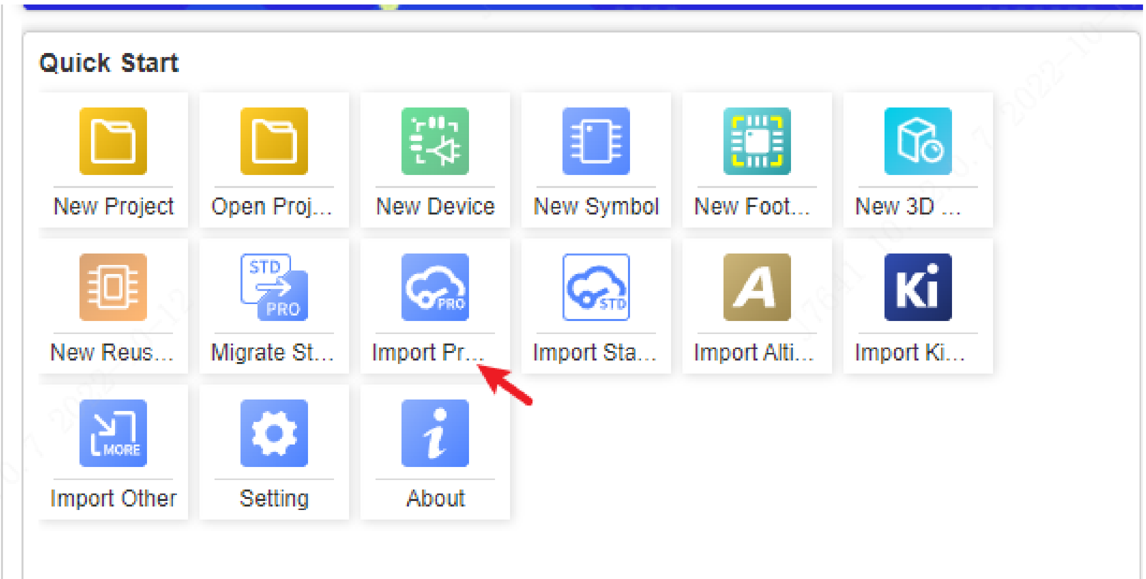
Elements/Layers	After Import	Notes
copper area	copper area will be re-laid after importing	Because the copper area logic is different, the copper area filling of the PCB will also be slightly different. For example, the heat welding generation method, the heat welding width, whether the heat welding is generated (it will give priority to avoid generating heat welding for the graphics elements with wrong DRC spacing), and flying lines may be generated
Text	Font changes and positions will be slightly shifted	If the corresponding font is not installed in the operating system and the corresponding font is not added in the font setting of the professional version, the default font (Arial and Arial) will be used after the text is imported, because the fonts of the text are in different positions There may be a slight offset; if it is the same font, because the font display logic is different, it cannot be completely consistent with the original file, and there will be position offset and size difference; strokes and barcodes will be converted to normal text

Elements/Layers	After Import	Notes
Internal electrical layer	The network of the block may change after importing	The internal electrical layer of the standard version and the professional version are implemented differently. When there are multiple internal electrical blocks, the network of the blocks may not be completely consistent. Check carefully; after importing, the internal electrical layer will rebuild the block, and the internal electrical layer block division may also be different
Solid Region	Go to different layers or types depending on type	Cutout type to Slot Region; Solid to Fill Region; No Solid to Prohibite Region
Ratlines	Flying lines appear after importing	It may be that the reconstruction of copper area after importing leads to disconnection in some places, such as no network primitives, heat welding cannot be generated due to DRC spacing, etc.
3D model library	does not support import	The 3D binding design of the professional version is different from that of the standard version. The 3D library import of PCB and package binding will not be imported together in the professional version, and the 3D model library needs to be re-bound

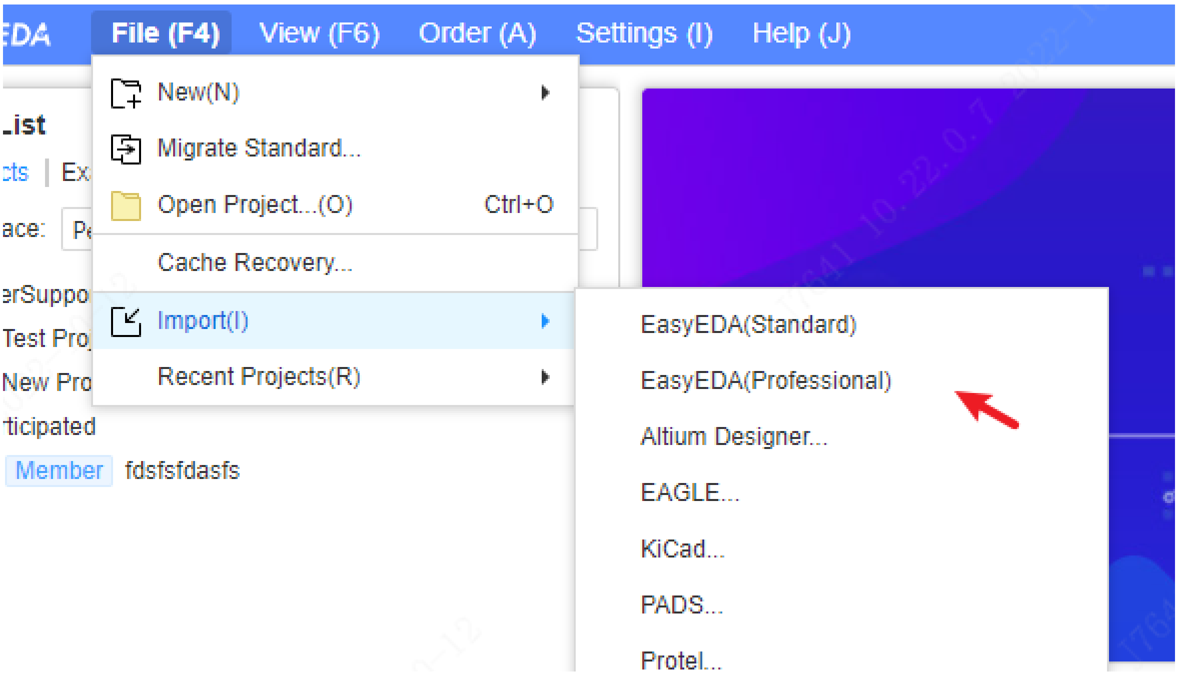
Import EasyEDA Pro

Import EasyEDA Pro

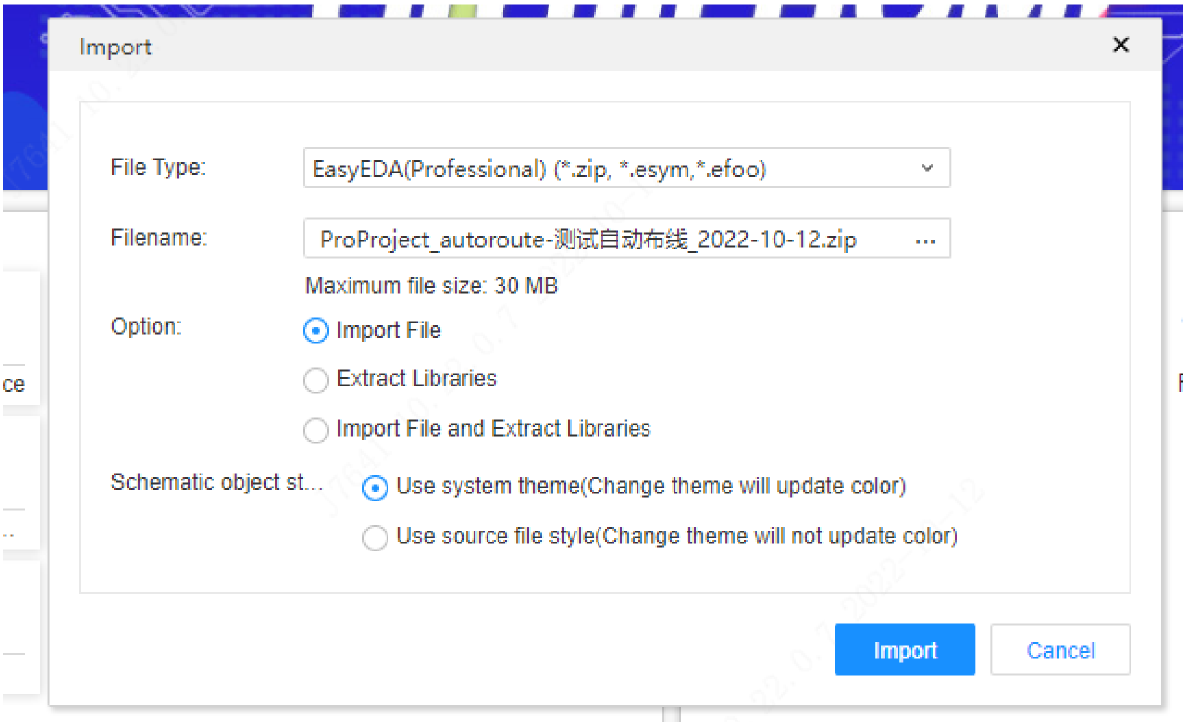
1) When the project is not opened in the professional version, on the EasyEDA Pro start page - import EasyEDA Pro (standard version), select the ZIP archive to import.



2), or at Top Menu - File - Import - EasyEDA Professional



Select the operation type of the file and click Import to complete the import of the standard version



Import Altium Designer

EasyEDA Pro supports importing into Altium Designer.

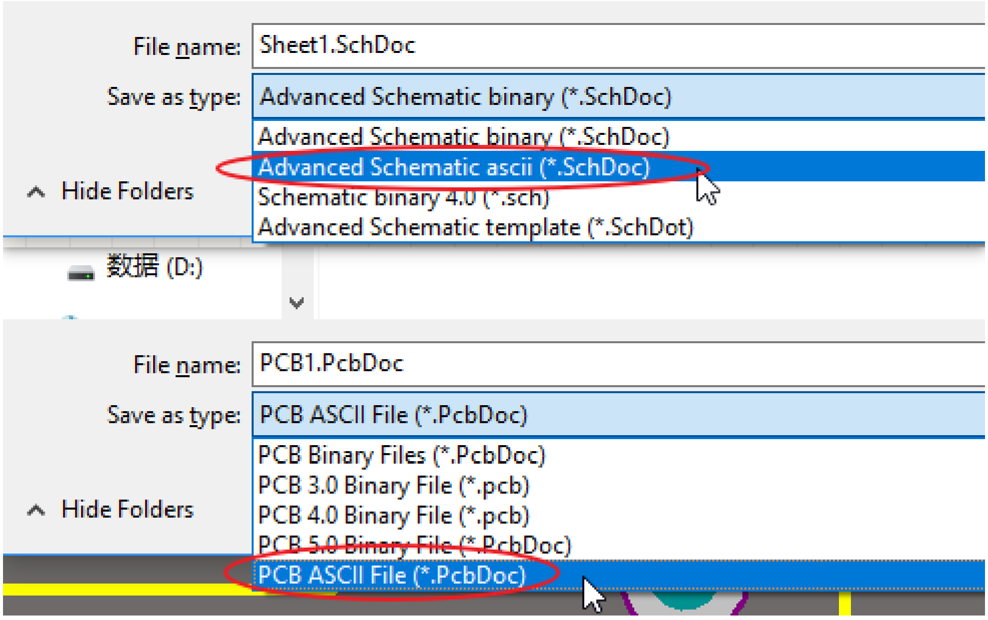
Several versions of Altium Designer have been supported, supporting plaintext ASCII format. At present, there is a problem with importing binary format, which is not supported for the time being.

Notice:

- **Due to the inconsistency of the format and graphic element design, there may be some differences after the graphic element is imported, please check carefully. For specific differences, please refer to the help documentation.**
- **Jia Lichuang EDA is not responsible for any losses caused by format conversion differences, if you do not agree, please do not import.**

Import Altium project file

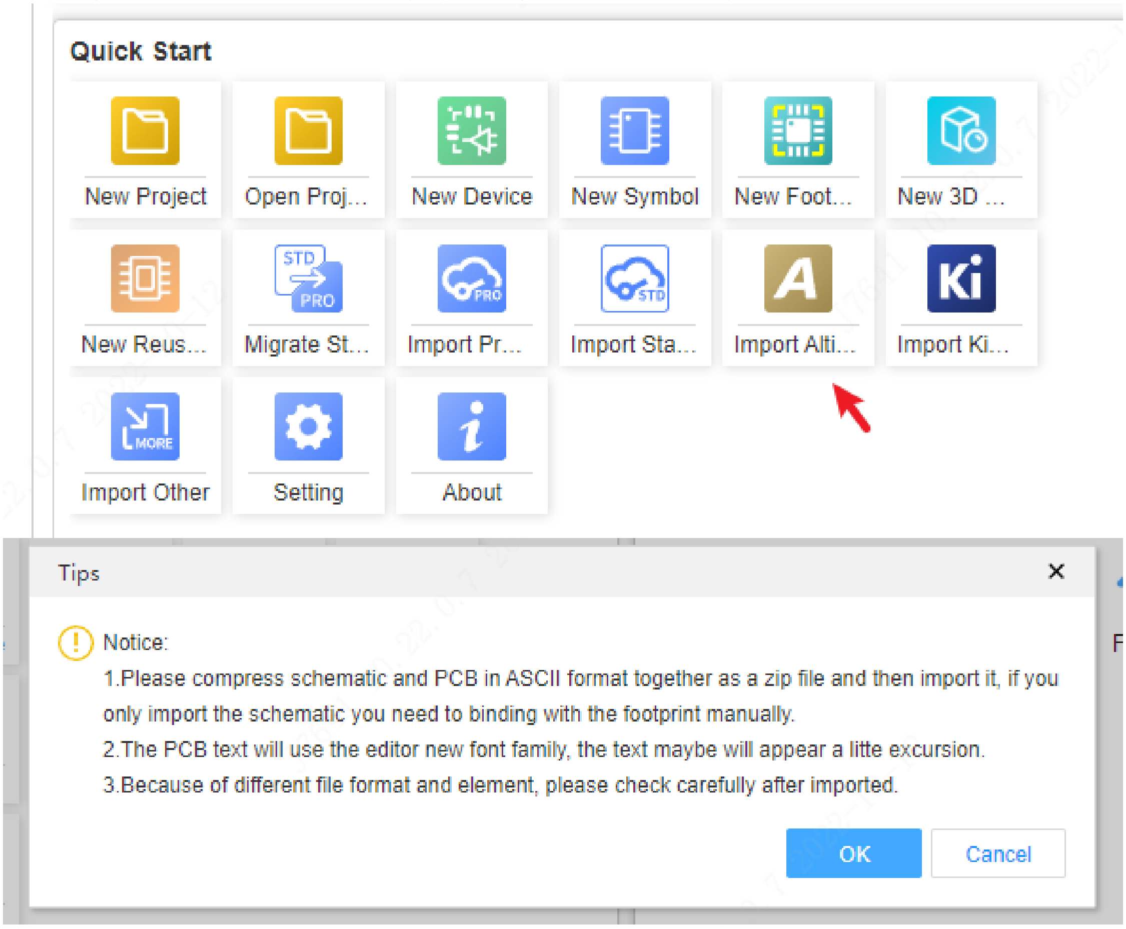
1. Open the schematic diagram and PCB in Altium Designer, in "File - Save As", select "altiumvanced Schematic ascii(*.SchDoc)" or "PCB ASCII File(*.PcbDoc)"



2. footprint the exported schematic diagram and PCB files into a compressed footprint ZIP format. Compression format only supports zip.

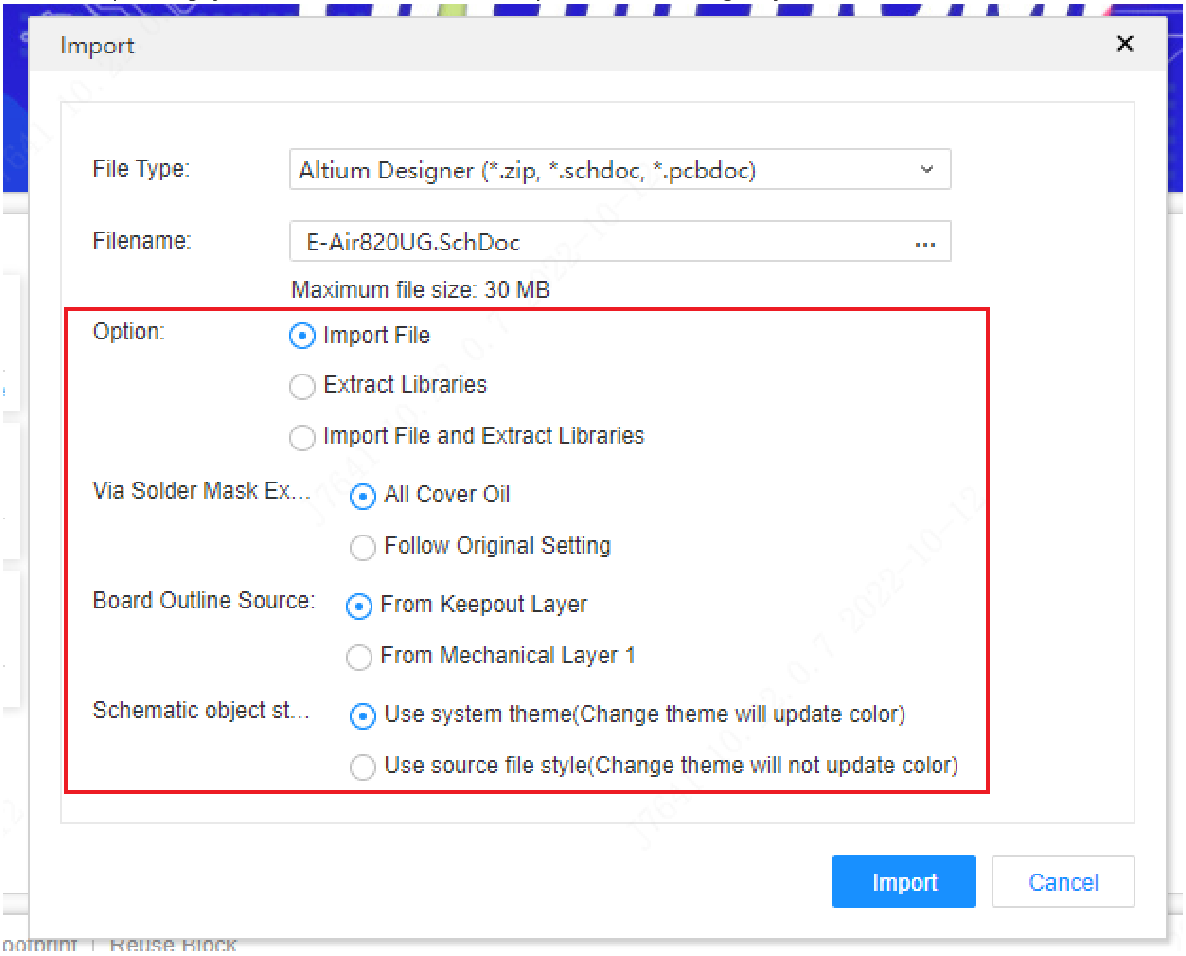
Note: It also supports the import of a single schematic or PCB, but the import of a single schematic cannot automatically bind the footprint, and it needs to be manually bound after import.

3. On the professional version start page - import Altium to import.



Click OK and select the imported file.

When importing, you can choose different options according to your needs.



options:

- Importing files
- Extract library files
- Import files and extract libraries

Via solder mask extension:

- All are covered with oil by default. It will force all vias to be covered with oil (solder mask extension is set to -1000)
- follow the original setting. It will be set according to the solder mask parameters of the vias in the original altium file.

Border source:

- From the Keepout layer. Many users use the keepout layer to draw borders, so this layer is used as the border by default.
- From mechanical level 1. When mechanical layer 1 is selected, the closed keepout layer will be turned into a forbidden area, and the unclosed keepout layer will be transferred to the mechanical layer.

Precautions

The difference before and after format conversion is as follows:

Schematic/Component library:

Elements/Layers	After Import	Notes
Bus/bus entry	Import is not supported	The bus and bus branch of the professional version are designed differently from altium and cannot be directly imported and used
Pictures	Import is not supported	altium pictures are stored in local file paths, and ASCII files do not contain pictures
Off Sheet Connector	Converted to Circular Network Identifiers	Offmap Connectors are not supported in Pro Edition
Sheet Symbol	Imported as hierarchical icons	Pin styles for page breaks are not supported
Sheet Entry	Does not support import	Professional version does not support drawing entry
Device Sheet symbol	Does not support import	Professional version does not support device page symbol
Harness	Import is not supported	Harness entities are not supported in Professional Edition
Directives	Some does not support import	The professional version supports No ERC labels, and does not support other indication entities (differential pairs, parameter settings, coverage areas, compilation shields)
Text/Text Frame/Comment	Import as normal text	The professional version does not support comments at the moment. If the corresponding font is not installed in the operating system, the default font will be used after the text is imported, and the position of the text may be slightly shifted due to different fonts; When the altium file is saved as ASCII, the Chinese may be garbled, and the garbled characters will be automatically converted to underscores after import
Bezier curve/ellipse arc/ellipse	Import as multi-segment polyline	Professional version does not support Bezier curve, Bezier curve, ellipse arc, ellipse

Elements/Layers	After Import	Notes
Component Designator	Multi-part designator as U1A, U1B after import is U1.1, U1.2	The professional version does not support the multi-part designator method like altium
Pin type	Unsupported types are converted to input types	The professional version only supports three pin electrical types
Hidden pins	Show after import	Professional version does not support hidden pins
Component footprint	After importing, "original footprint 1, original footprint 2, etc." will be generated	The professional version is not associated with the footprint name, but with the uuid of the footprint, so when importing, multiple previously associated footprint names will be altiumded as common attributes
Theme style	The default theme of the professional version is used in the import dialog box	The default theme is that the rounded rectangle is converted to a right angle, the fill color is not preserved, and the color of the primitive is not preserved, so as to switch the schematic theme, and the theme of the original file follows the original file Style, color does not change when switching themes
IEEE symbols	Some IEEE symbols are not yet supported for import	The new version of altium has new IEEE symbols, some of which are not supported for import

PCB/Footprint Library:

Elements/Layers	After Import	Notes
Polygon Pour	polygon pour will be re-laid after importing	Because the polygon pour logic is different, the polygon pour filling of the PCB will also be slightly different. For example, heat welding generation method, heat welding width, whether heat welding is generated (priority will be avoided to avoid heat welding for graphics elements with wrong DRC spacing), and flying lines may be generated; horizontal and vertical polygon pour will be converted to grid polygon pour; altium polygon pour on the grid will heat and weld the vias separately, which is not supported by the professional version, and some vias may not be connected to the polygon pour after rebuilding the polygon pour (when the grid is relatively large); the professional version does not support non-signal layers polygon pour, so the polygon pour on other layers of altium will be transferred to the top layer polygon pour
Design Rules	Some design rules do not support import	For example, custom altiumvanced design rules, design rules not supported by the professional version, common safety spacing, etc., the parameters of the imported PCB file rules need to be altiumjusted by yourself

Elements/Layers	After Import	Notes
Text/Text Box	Font changes and positions will be slightly offset	If the corresponding font is not installed in the operating system and the professional version of the font setting does not altiumd the corresponding font, the text will use the default font (Arial and Times New Roman) after importing, due to the text There may be a slight offset in different positions of the font; if it is the same font, because the font display logic is different, it cannot be completely consistent with the original file, and there will be position offset and size difference; strokes and barcodes will be converted to normal text
Internal Electrical Layer	The network of the block may change after importing	The implementation of the internal electrical layer of altium and the professional version is different. When there are multiple internal electrical blocks, the network of the blocks may not be completely consistent, so you need to be careful Check; the internal electrical layer will rebuild the block after importing, and the internal electrical layer block division may also be different
keepout layer	is converted to frame layer by default	In the import pop-up window, it is supported to set whether the source of the frame is the keepout layer or the mechanical layer 1; the keepout closed lines inside some boards will automatically be converted to the corresponding forbidden area according to the target; Closed keepout primitives will be imported to the mechanical layer. The professional version does not support independent lines as forbidden areas
Board Shape	Go to the document layer by default	The professional version does not have a board shape but has a border layer. Since most people use keepout and mechanical layer 1 as the board frame, the board shape goes to the mechanical layer
Define Board Cutout	Go to the frame layer or convert to the cutout area	This function corresponds to the cutout area of the professional version
Polygon Pour Cutout	Convert to prohibited area	Convert to multi-layer prohibited area, the prohibited content is polygon pour

Elements/Layers	After Import	Notes
Mechanical Layer	Mechanical layer 1 is transferred to the mechanical layer by default, and other mechanical layers are transferred to the user-defined layer	The professional version uses the user-defined layer to realize the functions of other mechanical layers of altium
Ratlines	Flying lines appear after importing	It may be that the reconstruction of polygon pour after importing lealtiums to disconnection in some places, such as no network primitives, heat welding cannot be generated due to DRC spacing, etc.
Room	Does not support import	Professional version does not support Room primitives
Layer Stack Settings	Import is not currently supported	To be supported later
3D Body/3D Model Library	Does not support import	The 3D binding design of the professional version is different from altium
Drill table	Import to Drill layer	Professional version has separate Drill layer
Solder Mask and Paste Mask Extension	Only pad/vias support import	Solder mask and Paste Mask extensions of other wires, arcs and other primitives do not support import. If they are independent primitives, they will be in the solder mask or solder flux layer (solder paste layer) generate an expanded primitive
Pad	Import to the bottom layer or top layer or multi-layer	Professional version of the pad does not support setting in any layer, the pad of the unsupported layer will be converted to the filling area of the corresponding layer; the rectangular drilling is not supported, it will be converted to Groove Drilling

Elements/Layers	After Import	Notes
Component/Footprint	Import to the top layer or bottom layer	If the footprint attribute drawn in altium is on the top layer, but the whole is on the bottom layer (belonging to wrong data), it will automatically correct the layer attribute to be on the bottom layer after importing; if there are multiple files with the same name but different sizes footprint, the import will take one of them as a template by default to associate with other components
Coordinates	Import as Text and Lines	Coordinate primitives are not supported in the Professional Edition
Dimensions	Partial support for import	Support for importing linear dimensions, angles, and dimensions, but not for partial import as text and lines
Object from file	Import not supported	This calls an object outside the file, not contained in the file, and does not support import
Work Guides	Import is not supported	This element is not supported in Professional Edition

Other considerations:

- Please do not repeatedly export your schematic or PCB to altium format and import them again, this operation may cause loss of details! ! !
- When importing Altium files, if there are unsupported characters (such as garbled characters), they will be automatically converted to underscores, so after importing, you may find that there is an extra underscore in network labels, device names, footprint names, attributes, etc. Please modify it manually. When the current version of altium is saved as an ASCII file, the Chinese will become garbled, so it is also garbled when imported. You need to open the ASCII file with a text editor such as Notepad and then import it after correction.
- The imported altium file supports a maximum size of 100MB. Larger files will lealtium to longer import time or import failure. It is recommended to manually reduce the file size. For example, delete the copper filling first (set the copper filling to no filling type) and then import.

Common Problem

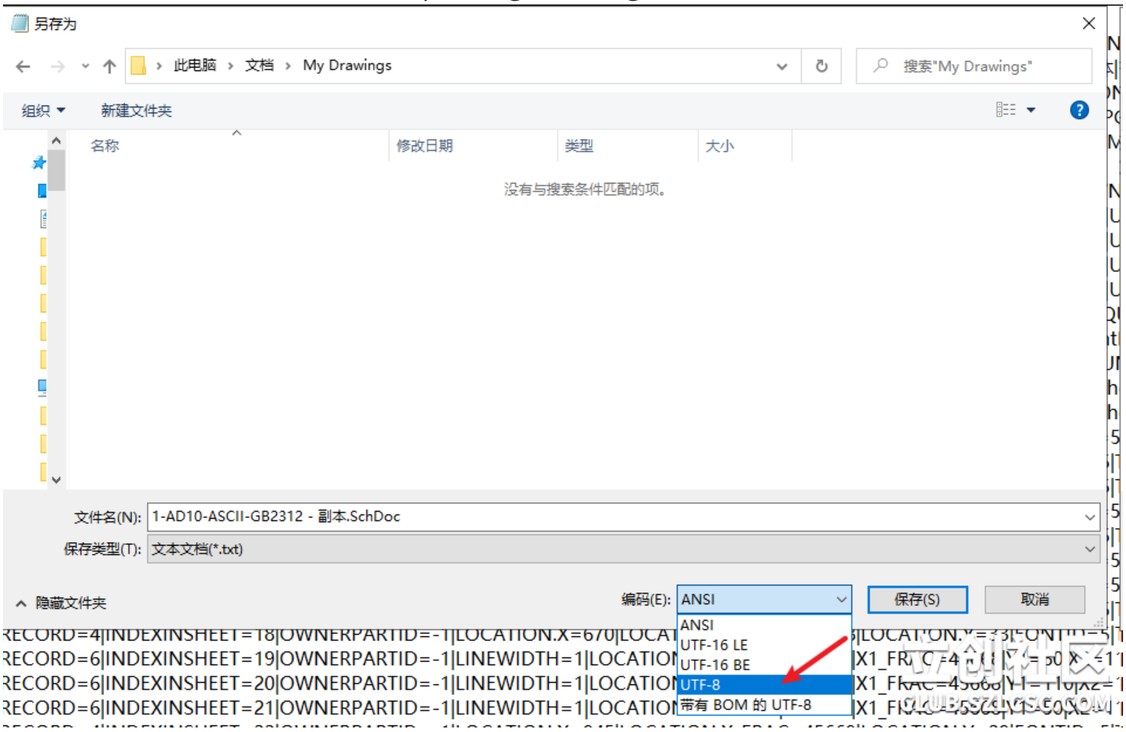
1.Chinese garbled characters appear when importing altium schematic

In versions below altium17, the encoding of the ASCII file saved as may be GBK2312, and the encoding of the ASCII file needs to be converted from GBK2312 to UTF-8. You can save as a UTF-8 encoded file using a text editor.

method:

- Save as ASCII file with altium17, which defaults to UTF-8 encoding

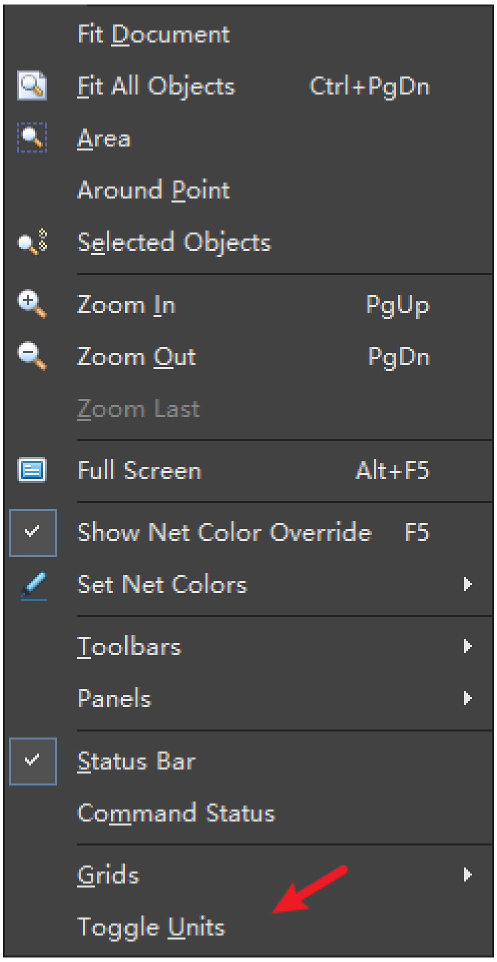
- Or open the ASCII file with the system's Notepad, and select UTF-8 encoding when saving as. Other text editors also have corresponding encoding conversion functions.



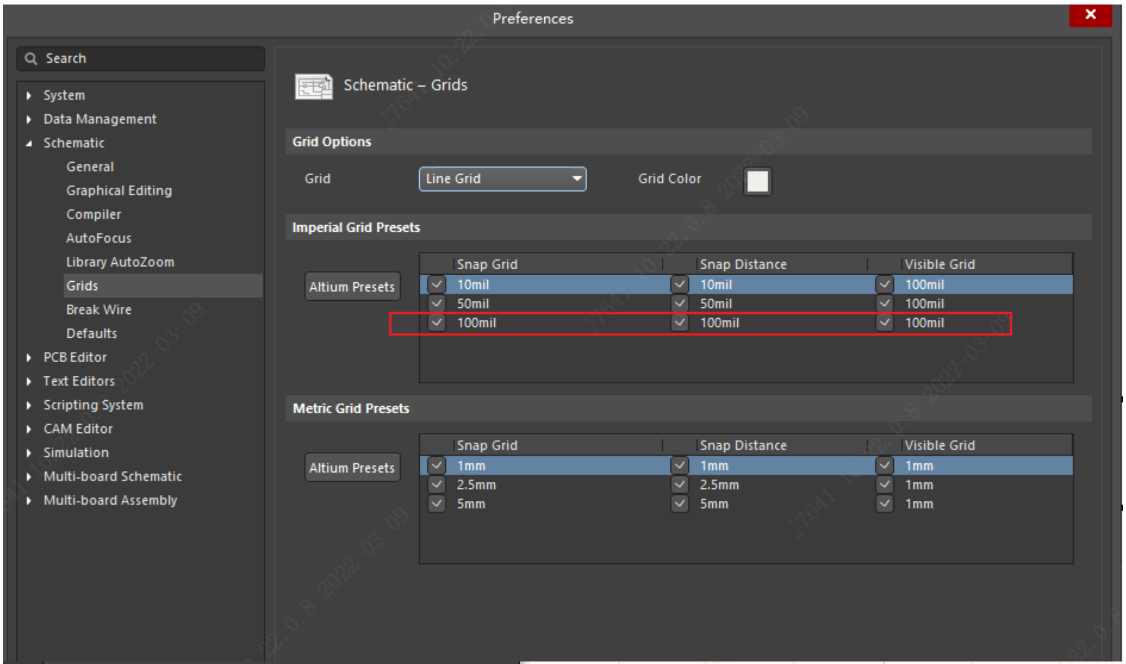
If after the text encoding is converted to UTF-8, there are still garbled characters when viewing the content with a text editor, then the import will also be garbled characters, please correct it in the text editor before importing.

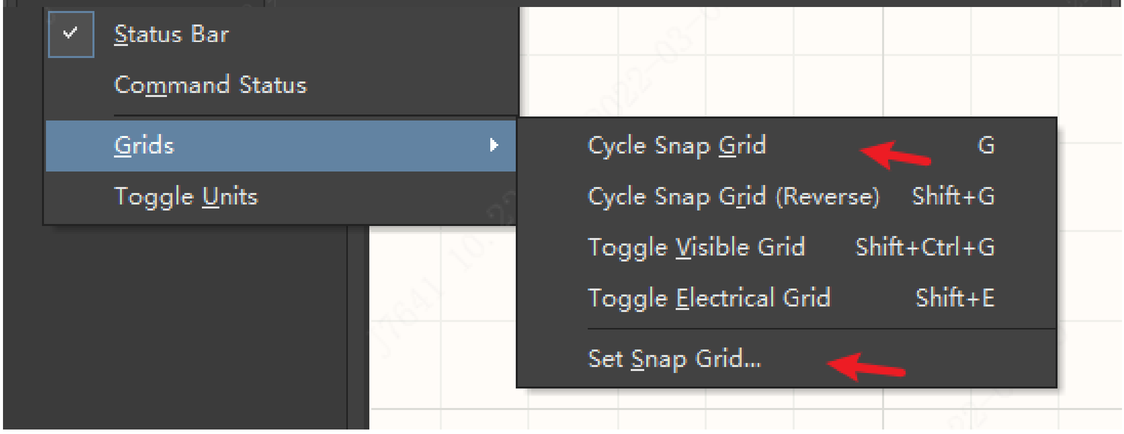
2.After importing the altium file, there are wires and device pins that are not aligned with the grid, or the pins are biased

- First set the unit of the schematic diagram in Altium Designer to imperial mil, in: view menu - switch unit

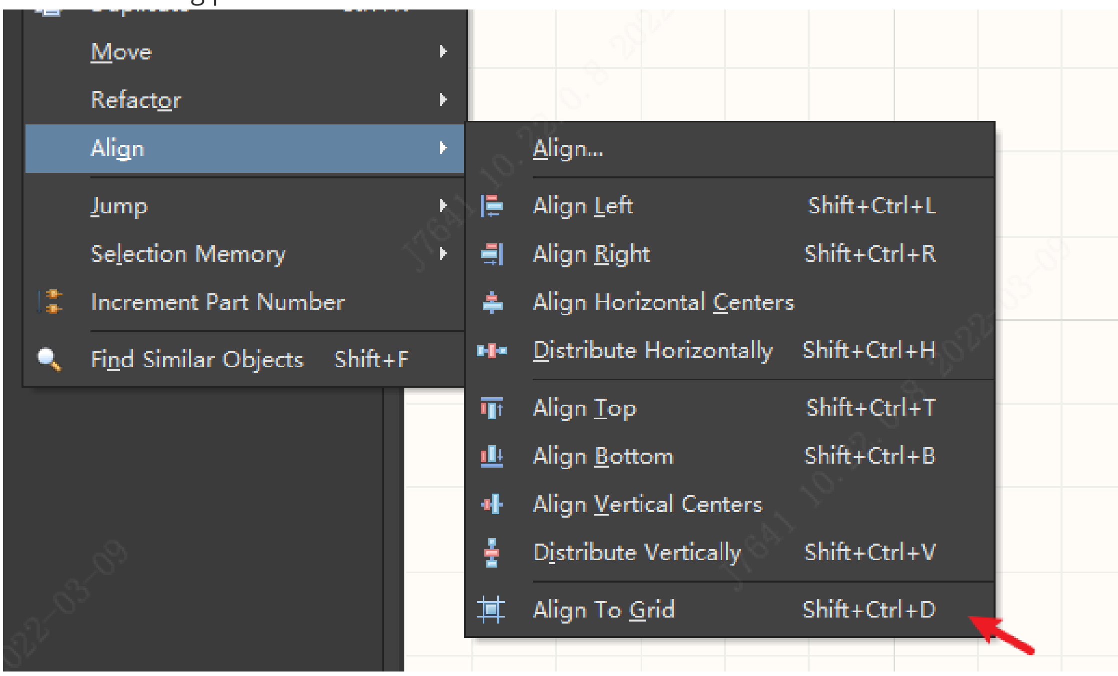


- Then the canvas right-click menu or system settings: Options - Grid - set the display grid, electrical grid, altiumsorption grid to 100mil, switch grid to 100mil





- Select all schematics CTRL + A , use Edit menu: Edit - Align - Snap to Grid function. Check for broken or wrong places to correct.



- Save as ASCII and import

3.The footprint size will change after importing.

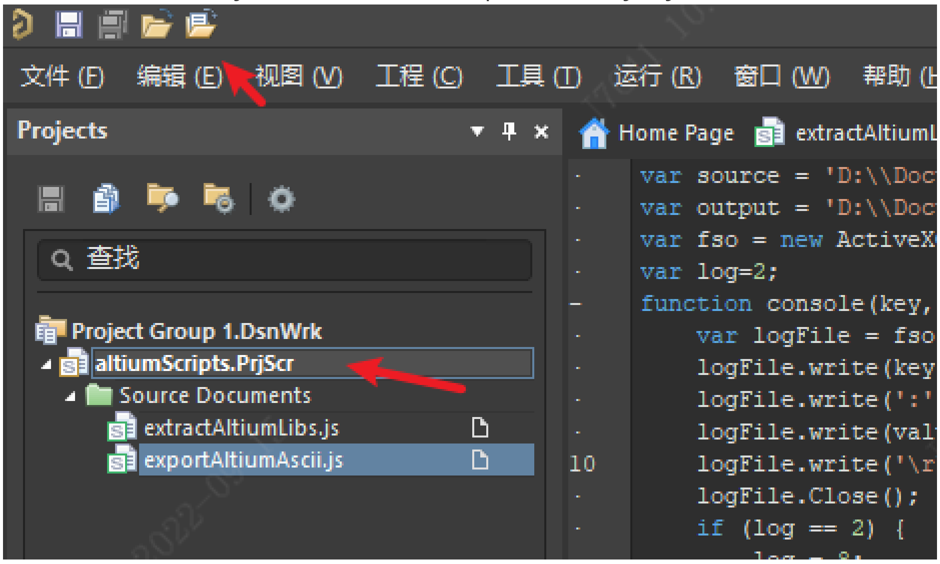
This is because the imported PCB has a footprint with the same name, but the footprint size is different. When importing, the bottom log will prompt that the footprint has the same name, and only one of them will be imported as a template.

Solution: It is recommended to manually modify the footprint names with the same name and different sizes in altium to ensure that the footprint names are not repeated.

Batch import Altum files

Please refer to the method below.

1. Download the AltiumScript script. [AltiumScripts.zip]
2. After decompression, open the alitumScripts.PrjScr project in Altium Design, if the version of AD is too low, it may not be able to run, please verify it yourself.

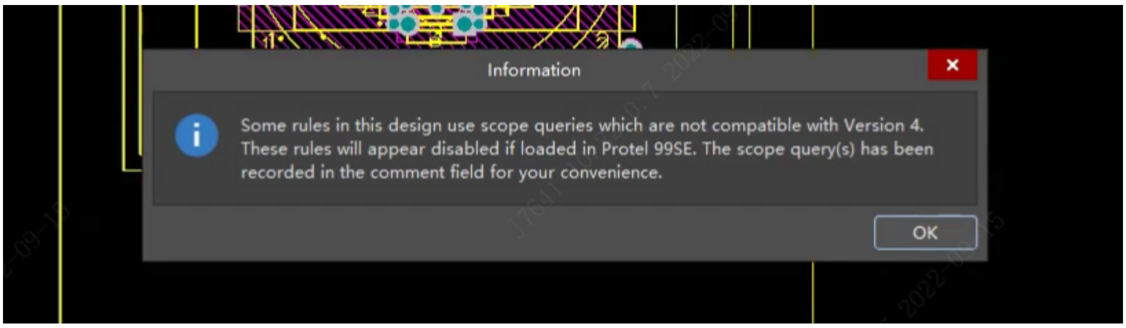


3. On the top menu - File or DXP - Run Script (File/DXP - Run Script) to open the script dialog box
4. Select the name of the script to be run, and click Run.

convertDesignToAscii.js: Convert the selected schematic and PCB files to ASCII format

convertLibToAscii.js: Convert the selected schematic library and PCB library files to ASCII format

5. After running the script, the script will automatically create a schematic or PCB file, and put the libraries in the currently installed library file into the document one by one. During the period, there may be multiple pop-up windows, and you need to manually click Confirm to continue.



6. After completion, you will find the automatically generated ASCII file in the custom output directory.
7. Compress the generated schematic diagram and PCB file together into a zip package and import it into EasyEDA, and choose to extract the library or import the file.

Import Allegro/OrCad

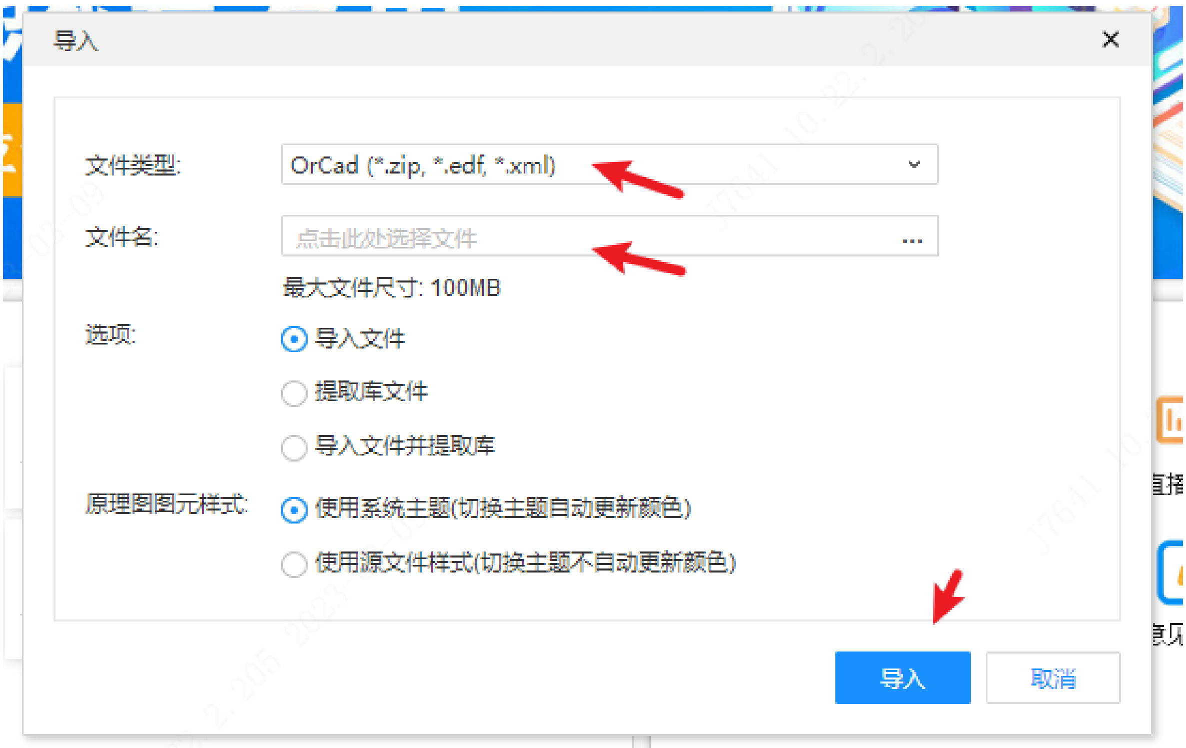
EasyEDA Professional Edition supports importing OrCad schematic files and Orcad schematic library files. You need to export the schematic diagram to EDIF format in Orcad first, and export the schematic library file to XML format before you can import it in Jiali Chuang EDA Professional Edition.
Starting from v2.1, it will support importing Allegro files. It needs to use script conversion to convert Allegro to ASCII format before importing.

Steps

Import OrCad files

1. Open the OrCad software and open the schematic DSN file
2. In the top menu - File - Export - EDIF or Library XML, get the schematic EDF suffix file or the schematic library file XML. If you are exporting XML, you need to manually select the OLB file and output directory to be exported.
3. On the EasyEDA Pro start page, choose to import other

Select Allegro/Orcad from the drop-down list of the file type, select the file and import it. You can choose to import the file or extract the library according to your needs.

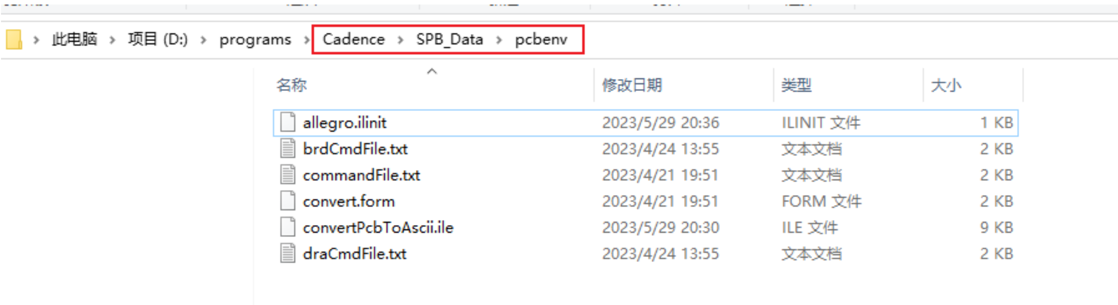


If the OrCad schematic diagram and Allegro's PCB are the same project, you can convert the Allegro files and compress them together in a zip archive and then import them into Jialichuang EDA. When importing, the package will be automatically associated with the device.

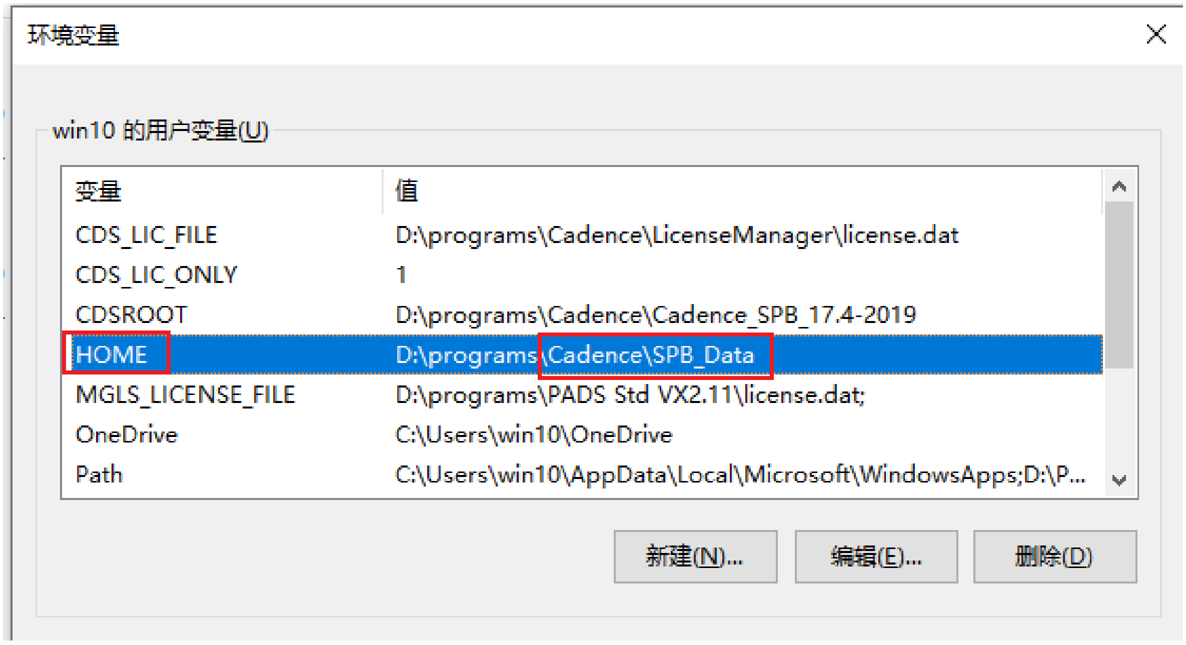
Import Allegro file

Since v2.1, EasyEDA Pro support to import Allegro. Because Allegro's file is an encrypted database file, you need to use one of its extraction tools to convert the database into an ASCII file. Download the conversion plug-in below, and you can export ebrd and edra files after installation.

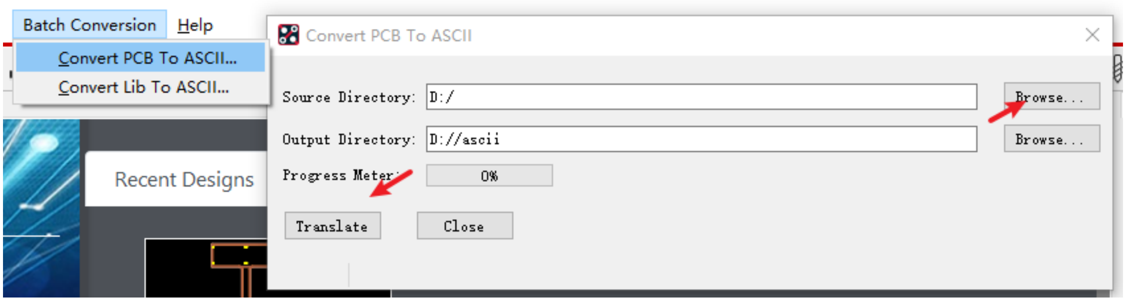
1. Download the Allegro to ASCII plug-in: [AllegroScripts.zip]
2. Install the plug-in according to the tutorial in the compressed package. Note that plugins may not be compatible with lower versions of Allegro, please verify by yourself.
   1) Find the Allegro installation directory, find the directory `xxx\Cadence\SPB_Data\pcbenv`, and copy all the files in the compressed package to this directory. If the `allegro.ilinit` file already exists under `pcbenv`, add a line of code `loadi("convertPcbToAscii.ile" "ascii")` in `allegro.ilinit`.



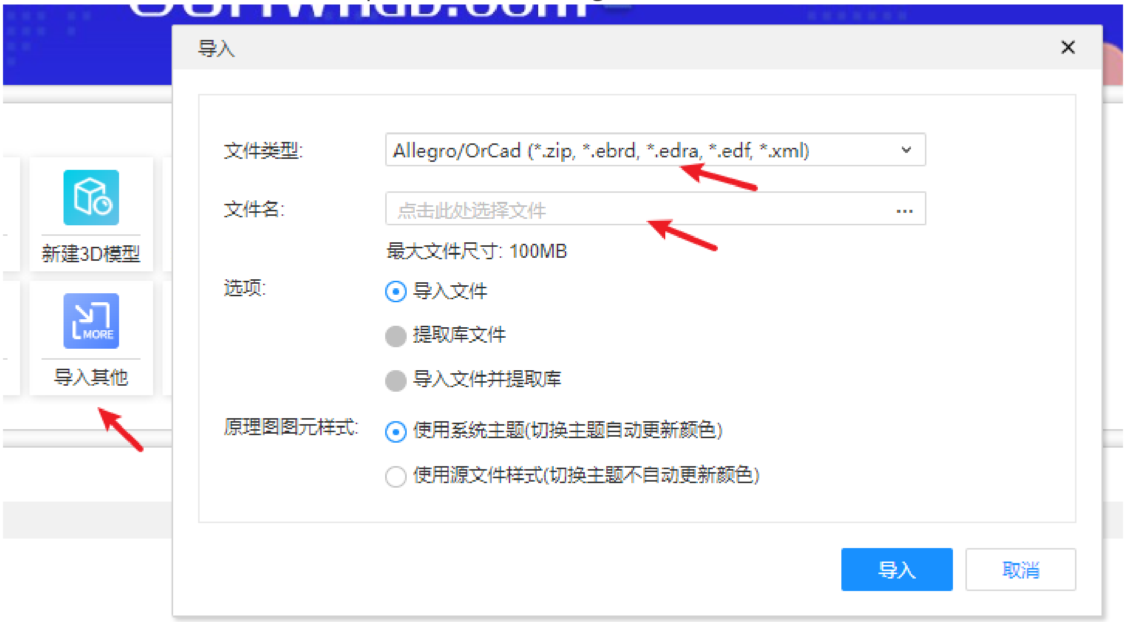
2) In the environment variable setting of the operating system, add the directory `HOME=xxx\Cadence\SPB_Data`.



3. Open the Allegro software, and open the Convert PCB to ASCII or Convert Lib to ASCII pop-up window in the Batch Conversion menu on the top menu.



4. After opening the pop-up window, select the directory where the brd or dra file to be converted is located, and set the directory to be output, and click the Translate button to convert.
5. Find the PCB file with the suffix of ebrd in the output directory, or the package library file with the suffix of edra, and import it in Jialichuang EDA



Batch Conversion

Batch convert OrCad

If you have a large number of Orcad schematics or schematic libraries, you can use the following plug-in for batch conversion, which will scan and convert according to the folder and subfolder you choose.

1. Download the conversion script: OrcadScripts.zip
2. Install the script after decompression. Perform the installation script according to the Readme.txt inside.
3. Perform batch conversion of schematic DSN files or schematic library OLB files according to the tutorial.
4. After the conversion is completed, find the file in the output directory, and then import it in Jiali Chuang EDA. If the OrCad schematic diagram and Allegro's PCB are the same project, you can convert the Allegro files and compress them together in a zip archive and then import them into Jialichuang EDA. When importing, the package will be automatically associated with the device.

Batch convert Allegro

If you need to convert Allegro files to ebrd or edra format in large batches, you can use the conversion plug-in provided above to perform batch conversion after selecting a folder.

The plug-in will automatically traverse the subfolders and all brd files or dra files under the folder. It will take a long time for batch conversion, please wait patiently.

Precautions

1. Because the Orcad schematic diagram does not include the PCB, so far after importing the Orcad schematic diagram to Jialichuang EDA, the package attribute of the component will be empty, and the package needs to be re-bound. This is because the package binding design is different from that of Orcad. Jiali Chuang EDA needs to be associated and bound, and cannot be associated by entering the package name.
2. The graphics element design of Allegro is different from that of Jialichuang EDA, and there may be differences in details, such as copper laying, thermal welding of pads, square holes, porous pads, etc., will be imported according to the graphics of Jialichuang EDA. Meta properties are automatically adjusted.

Difference before and after import

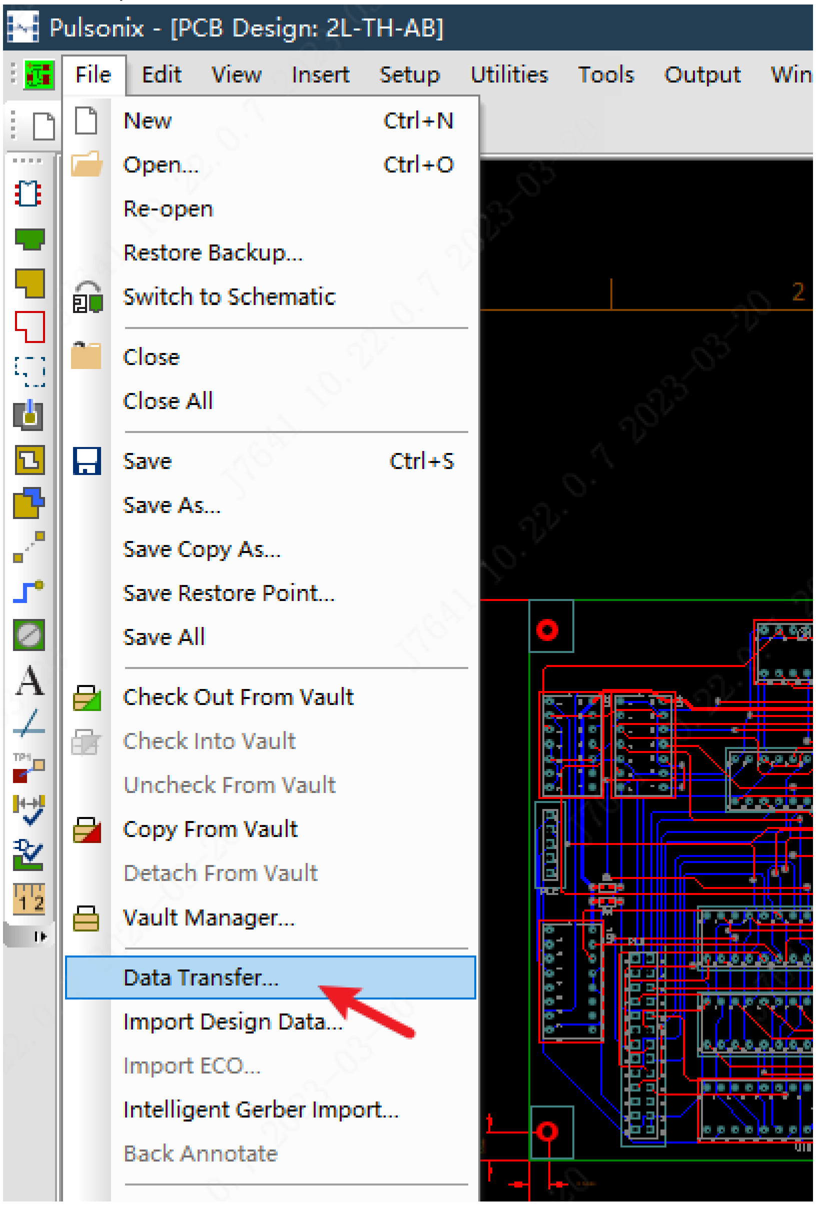
Graph Element/Layer	Allegro	EasyEDA Pro	Supplementary Explanation
Layer	PLACE_BOUND_TOP, PLACE_BOUND_BOTTOM	Component Outline Layer	Import the professional version of the component outline layer.
Layers	-	DOCUMENT	Some layers whose role is unknown are resolved to the document layer
Layer	-	Custom Layer	Layer not corresponding in Board Geometry
Font	Stroke Font	Default Font	Slight Deviation
Pad solder mask, solder flux	can be any graphics	solder mask, solder flux expansion	-
Pad number	Position can be set	Ignore the position information	Display in the center of the pad
Square drill holes	-	Replace with round holes	-
-	Drilling symbol	-	This symbol is not required
Line width	The line width of some primitives that need line width is 0	-	Set the default line width to 1mil
Multiple holes in the pad	-	Only one hole is reserved	Multiple holes in the pad are not supported at the moment
Pad inner layer graphics	Each inner layer graphic element can be defined separately	The inner layer graphics are the same	Take the default internal graphics in Allegro and import
Hot Air Pad	Thermal Pad	It can be replaced by thermal welding rules	Insufficient information in the package file, use the default rules
Isolation Pad	Anti Pad	It can be replaced by the inner layer network spacing rules or forbidden areas	It is not analyzed in the package file because the scene is not applicable
Inner layer graphics in the package	Corresponding to PCB inner layer graphics with the same name	Corresponding to PCB inner layer graphics with the same layer id (physical stacking order)	This is not a problem of importing, but users need to pay attention
Route Keepin	Routing Region	Does not support Keepin Region	-
Prohibited area	ROUTE, PACKAGE, VIA KEEPOUT	-	Judging prohibited items according to CLASS, there may be deviations in specific prohibited items
Copper configuration	-	Keep copper outline; use default copper rules	-
Heat Weld	-	Wire	Wire in ASCII format
Constraint Rules	-	Support for Partial Spacing and Physical Rules	Missing association of rules to networks
Constrained area	-	-	Insufficient information in ASCII format, not supported yet
Canvas configuration	-	-	Not documented in ASCII format, not supported yet
Element Color	-	-	Not documented in ASCII format
Picture activation, display state	-	-	Not recorded in ASCII format

Import CADStar

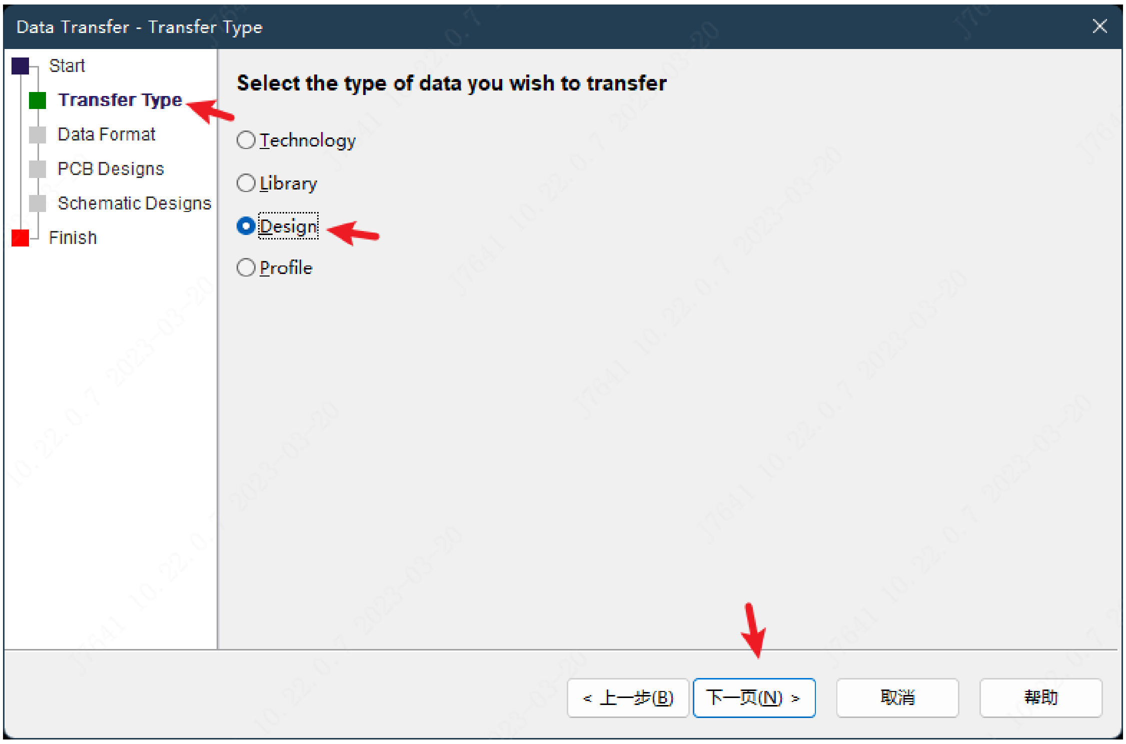
EasyEDA Professional Edition does not support direct import of CADStar format files. But you can use other EDA tools to convert it once and then import it.

method:

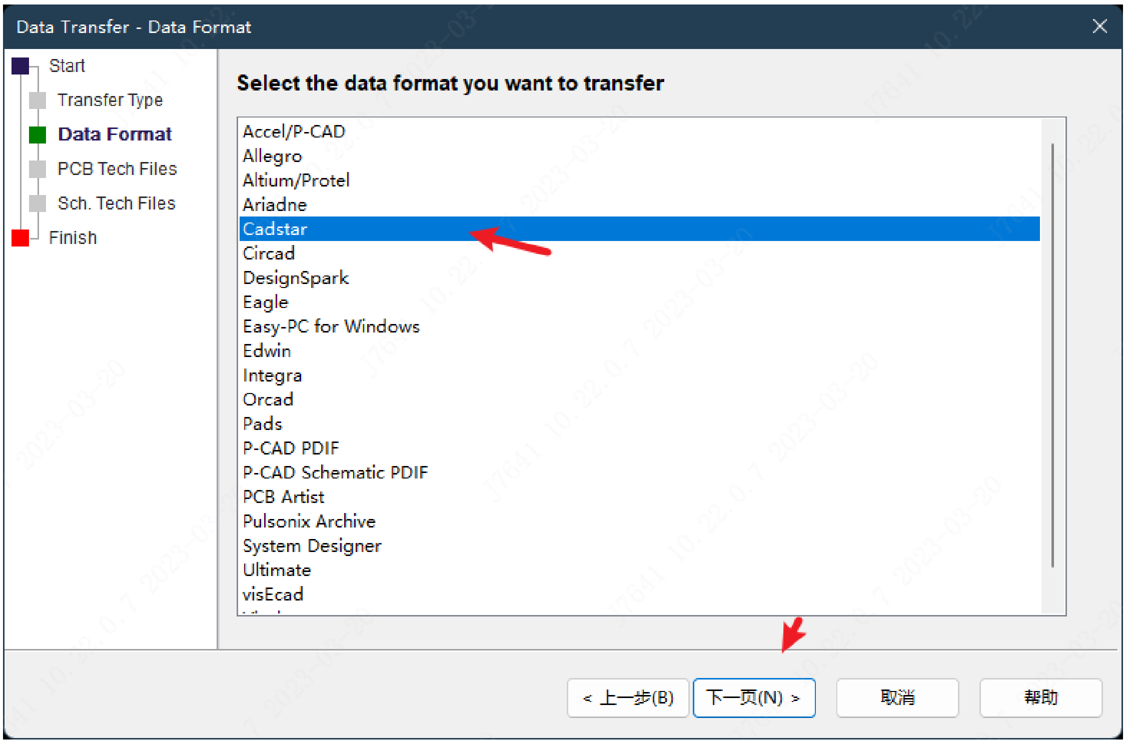
1. Download and install Pulsonix and open it.
2. On the top menu - File - Data Transfer.



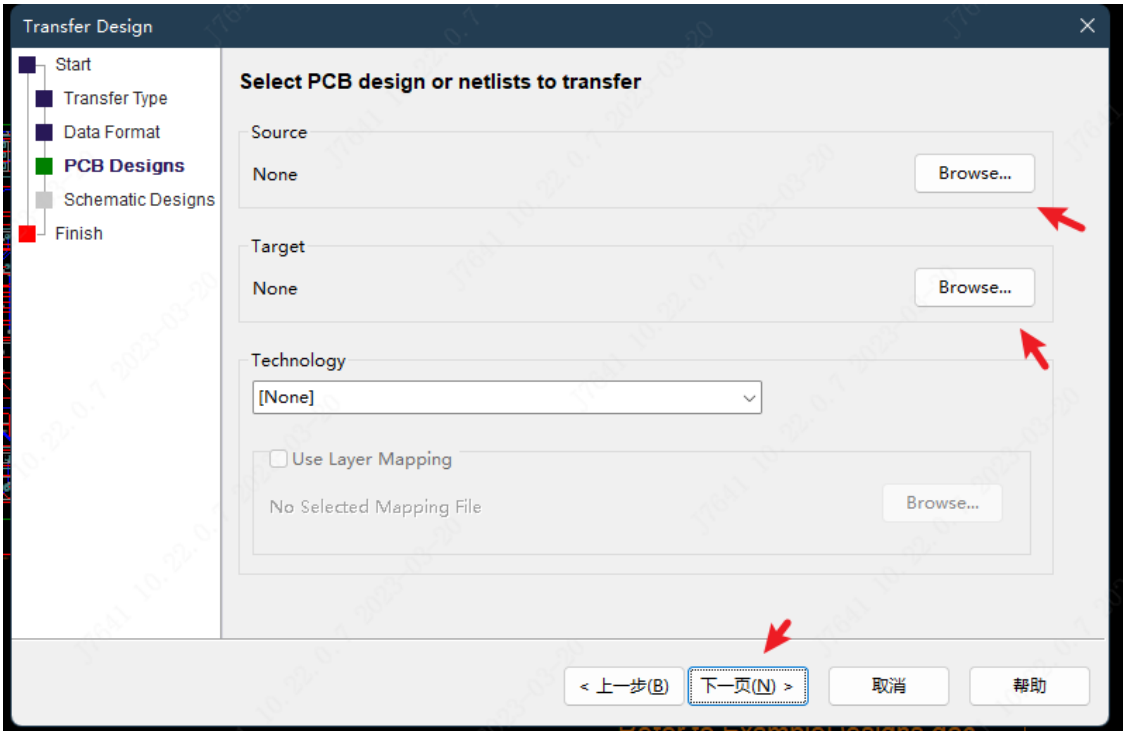
Select the design file to import:



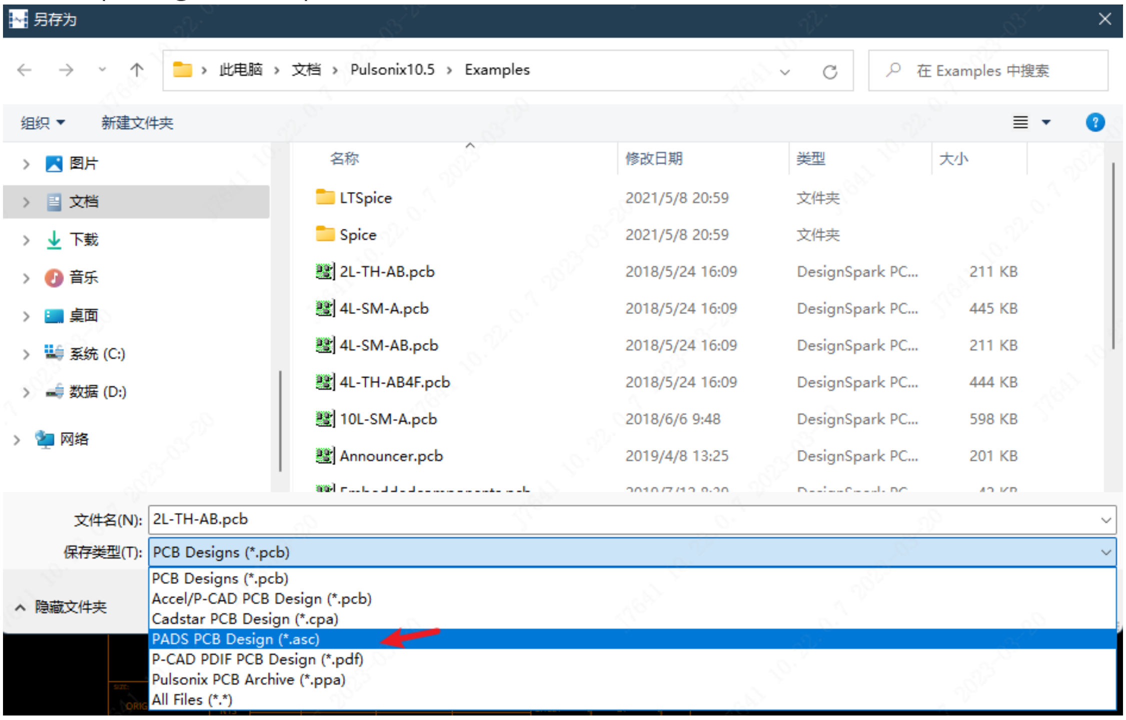
Select the CADStar file:



Set the directory where the file needs to be imported, and the directory where the output file is located, and then convert:



3. After importing, in the top file menu - file - save as, select PADS format to save.



4. Select the PADS file type in Jiali Chuang EDA to import it.

Notice:

1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

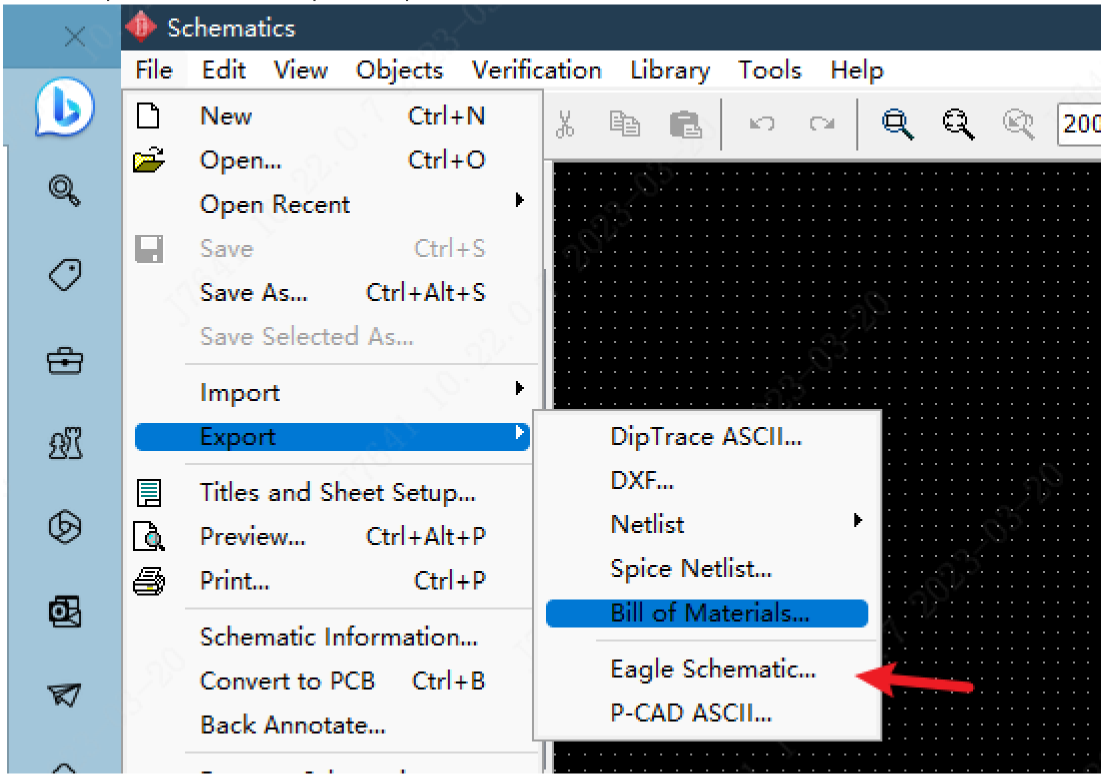
Import Diptrace

EasyEDA Professional Edition does not support direct import of Diptrace format files. However, other EDA formats can be exported through Diptrace and then imported.

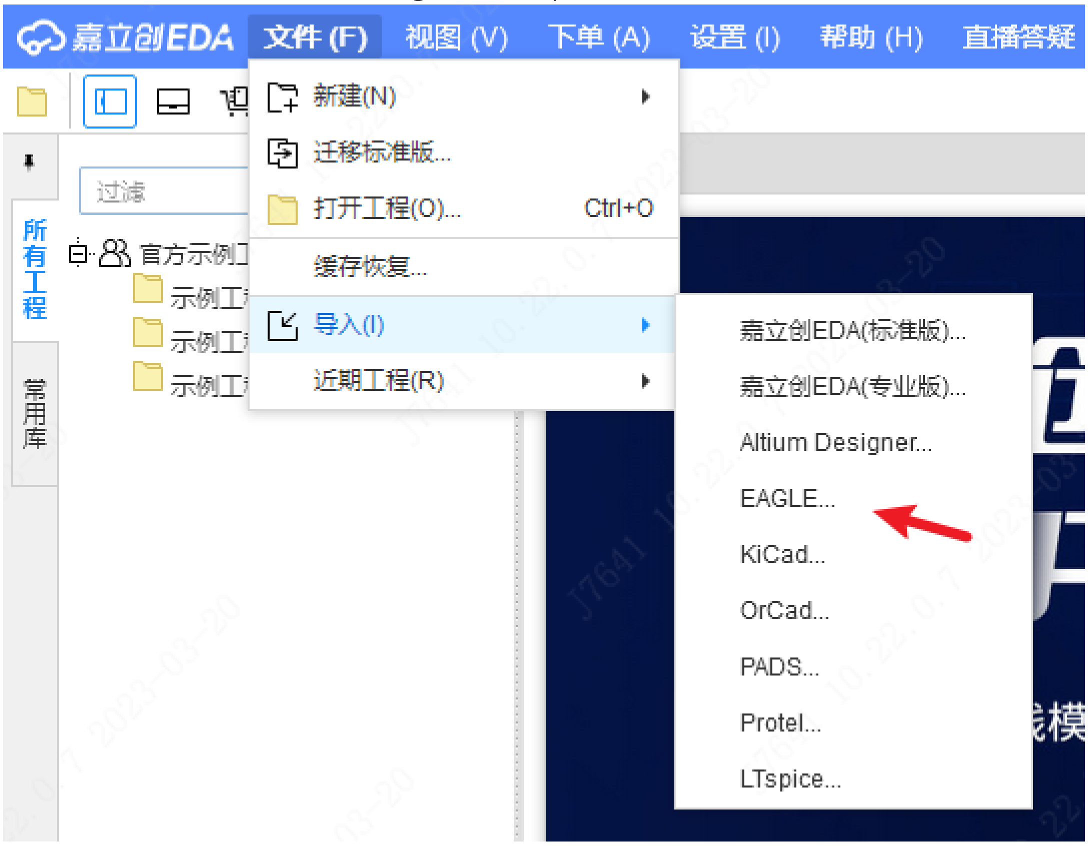
method:

1. Open the schematic or PCB file of Diptrace

2. In the top menu - File - Export, export the EAGLE format file.



3. Select the EAGLE file in Jiali Chuang EDA to import it.



Library files are the same

Notice:

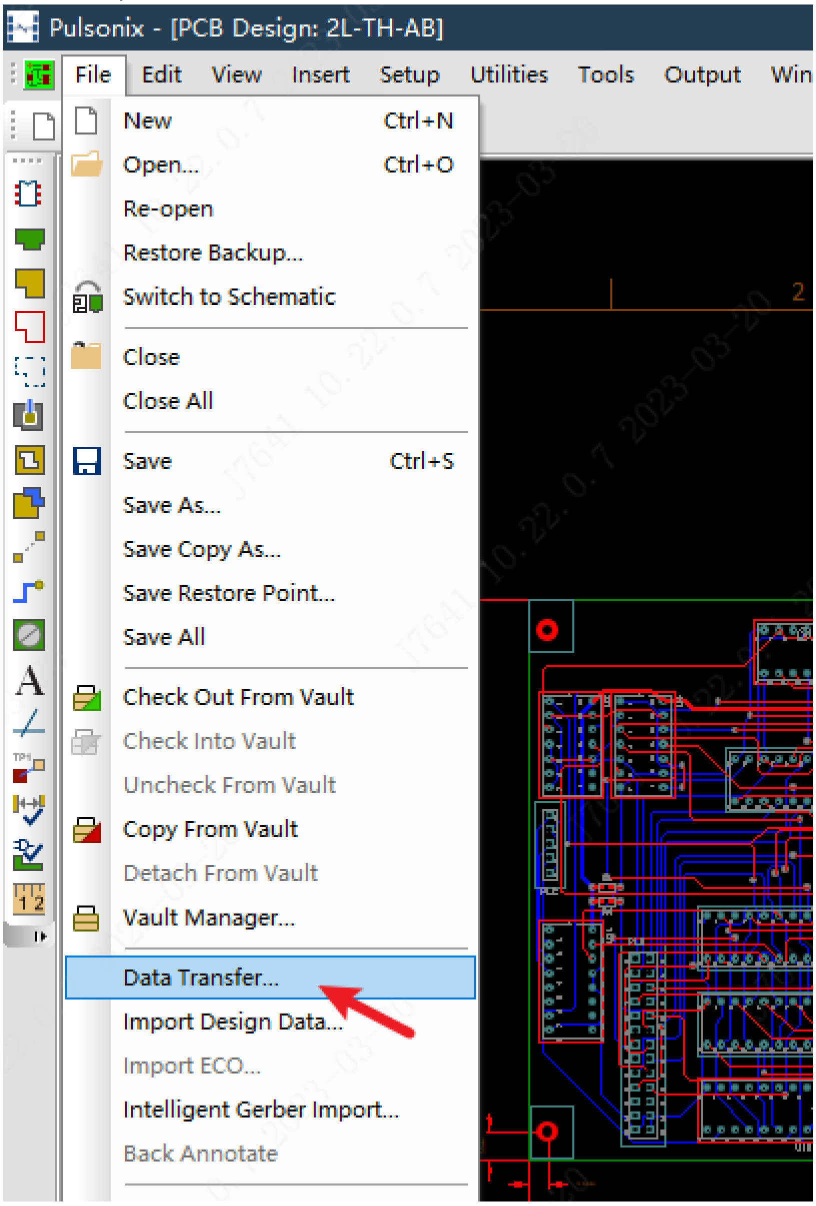
1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

Import DesignSpark

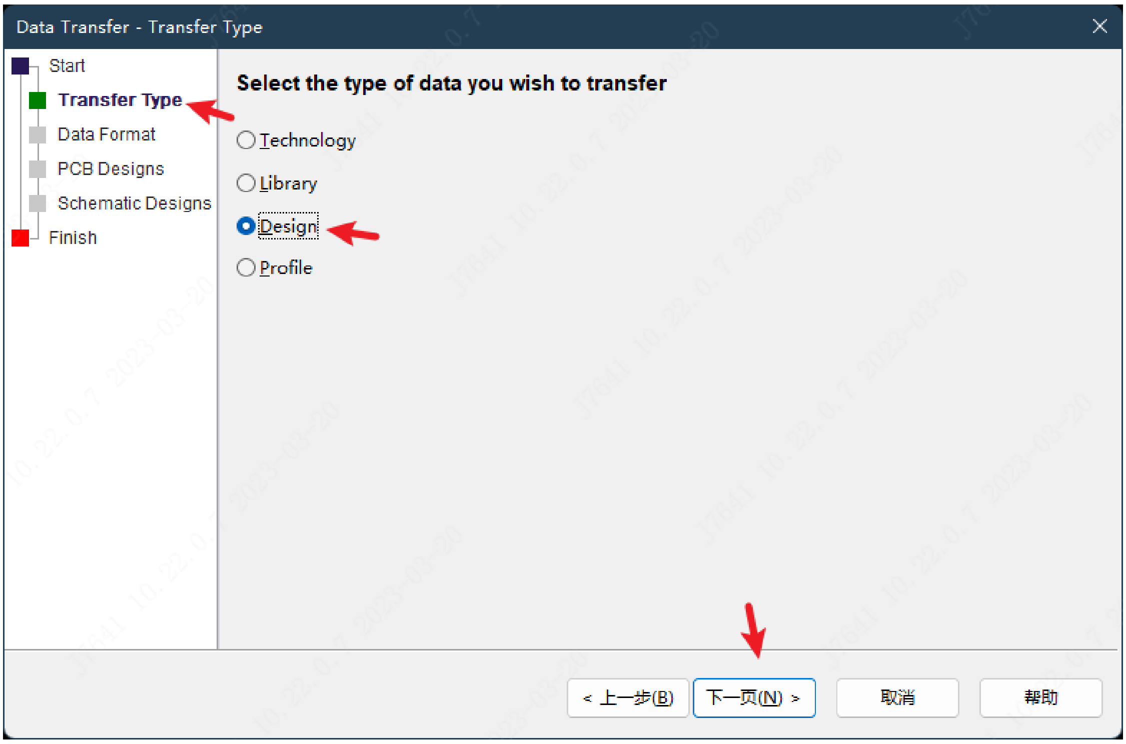
EasyEDA Professional Edition does not support direct import of Pulsonix format files. But you can use other EDA tools to convert it once and then import it.

method:

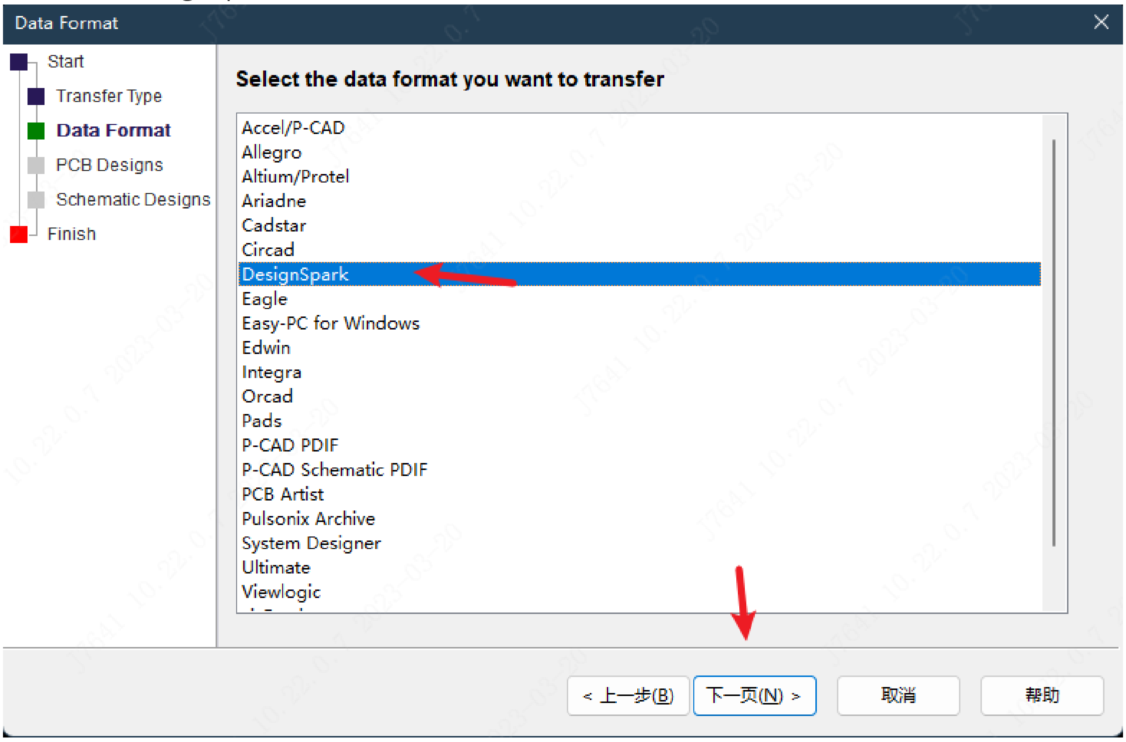
1. Download and install Pulsonix and open it.
2. On the top menu - File - Data Transfer.



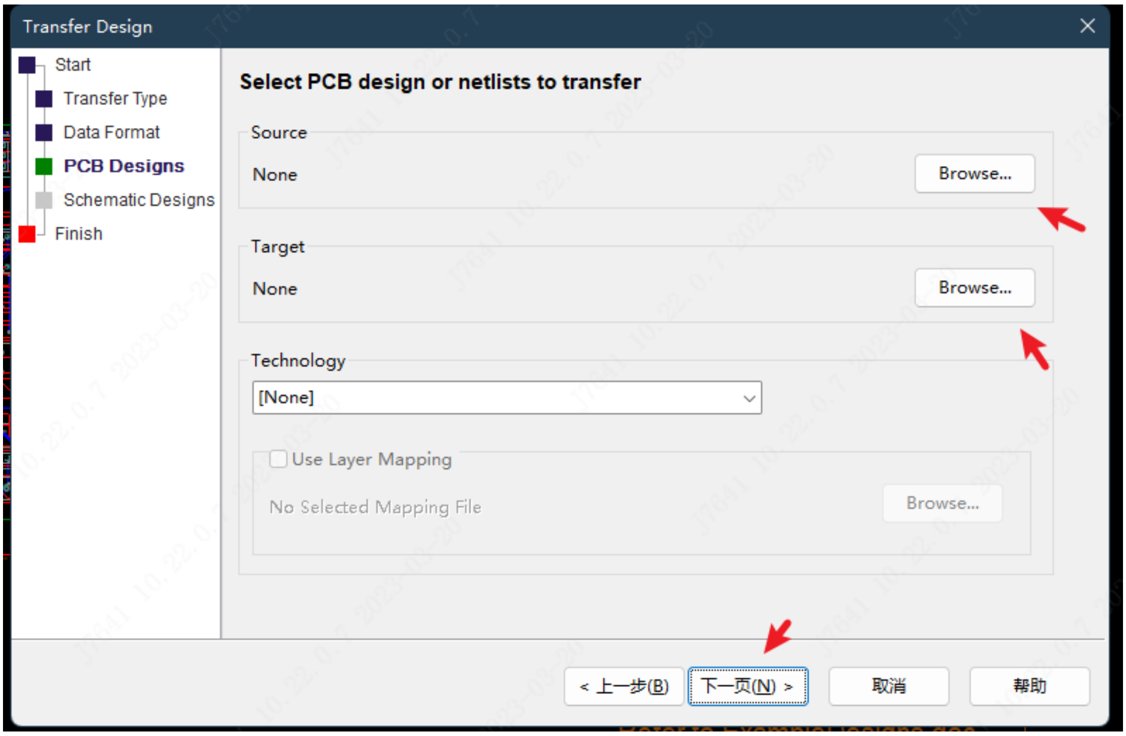
Select the design file to import:



Select the DesignSpark file:



Set the directory where the file needs to be imported, and the directory where the output file is located, and then convert:



3. After importing, in the top file menu - file - save as, select PADS format to save.
4. Select the PADS file type in Jiali Chuang EDA to import it.

Notice:

1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

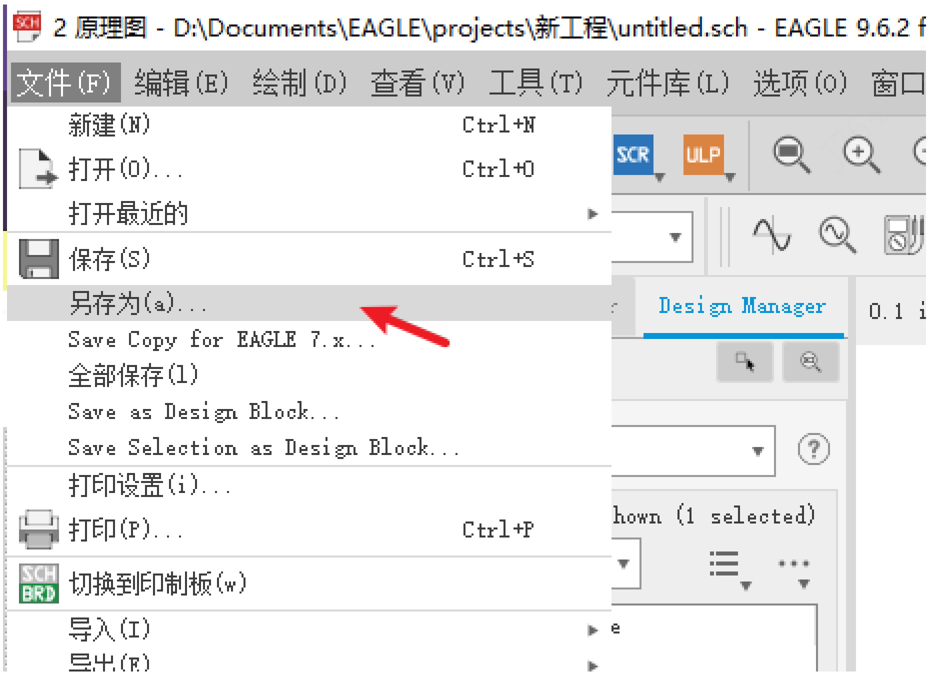
Import EAGLE

EasyEDA Pro supports importing EAGLE 6.0 and above format files. Direct import of EAGLE library files is also supported.

If it is an earlier version of the file, please refootprint and import it after saving the new version of EAGLE. EAGLE files prior to v6.0 are encrypted binaries and do not support direct import.

method:

1. Open the engineering schematic and PCB in EAGLE
2. On the File menu, save a copy as. Select 7.x and above.

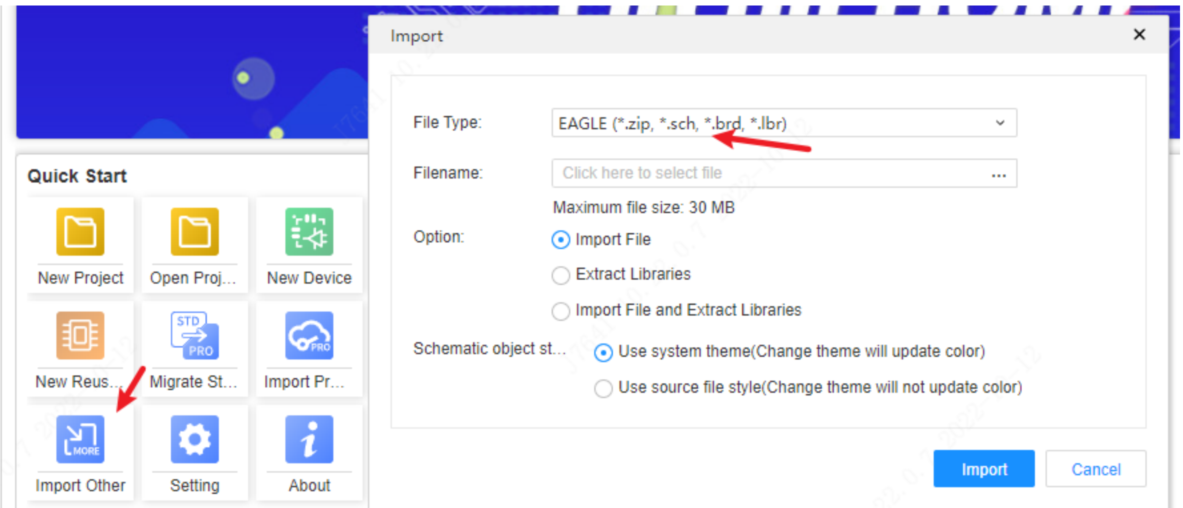


3. Find the schematic and PCB saved as, footprint them as zip and import them in EasyEDA Pro.

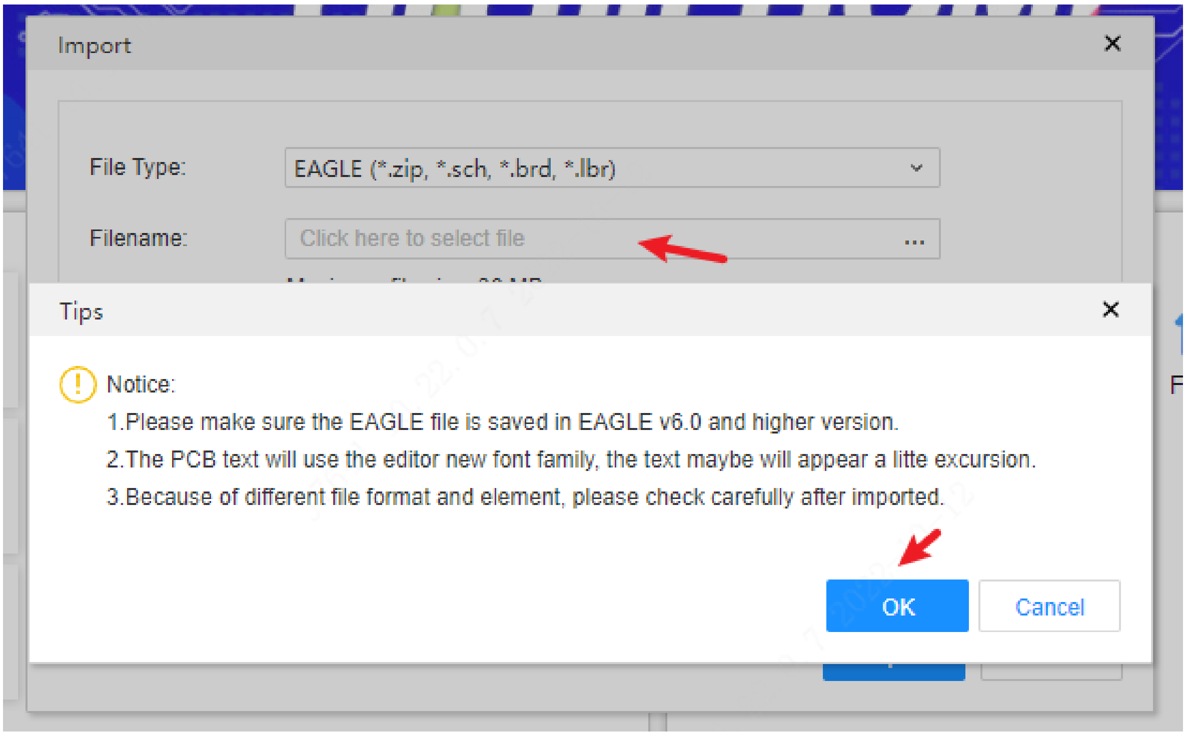
Some EAGLE files have garbled text, which may cause import failure. If you encounter it, please open it in the new version of EAGLE, save another copy, and import. If it still fails, please contact us and we will try to fix it.

Import EasyEDA Pro:

Entry: Start Page - Quick Start - Import Other - Import EAGLE. Or after opening the file: Top Menu - File - Import - EAGLE



Select file to import



Notice:

1. Due to the different functions and designs of the primitives, it is impossible to ensure the complete consistency before and after the import, please check carefully.
2. After PCB is imported, it will automatically rebuild and copper area. The result of copper area will be different, please check carefully.

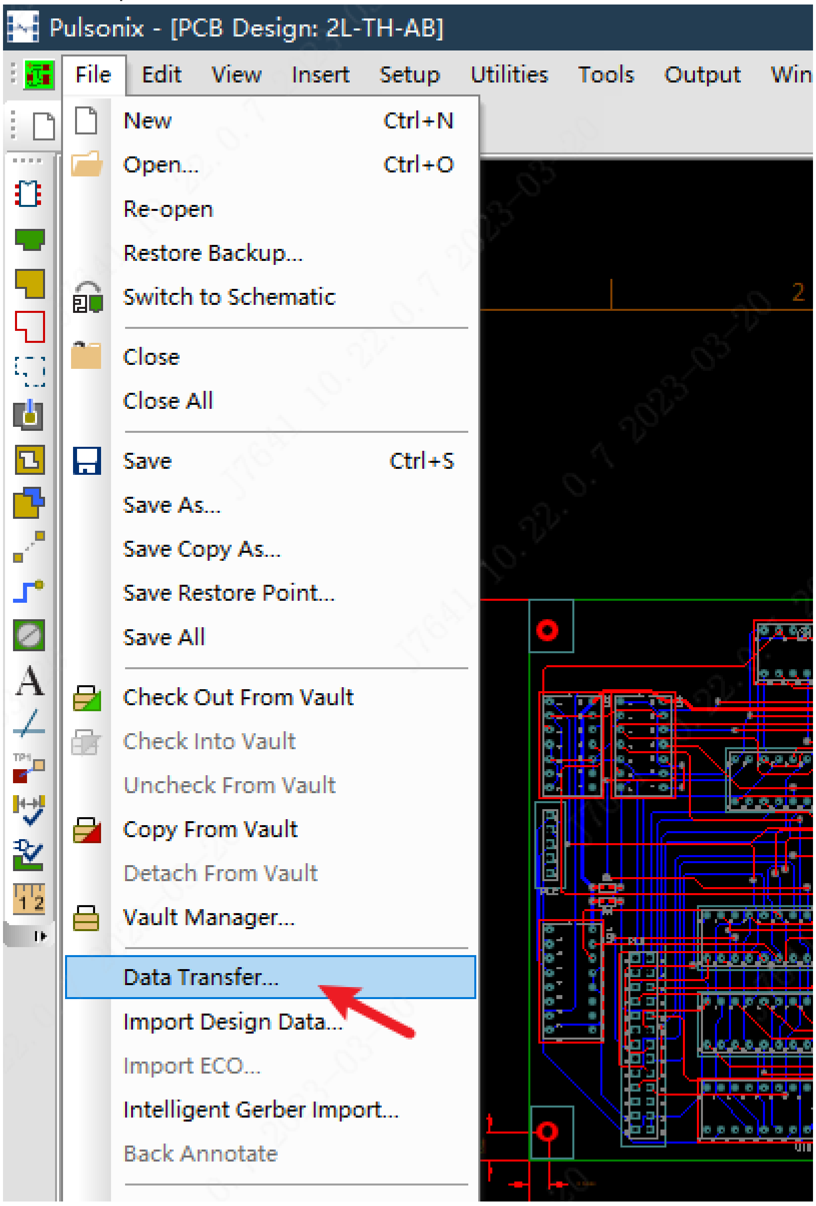
Import Easy-PC

EasyEDA Professional Edition does not support direct import of Easy-PC format files. But you can use other EDA tools to convert it once and then import it.

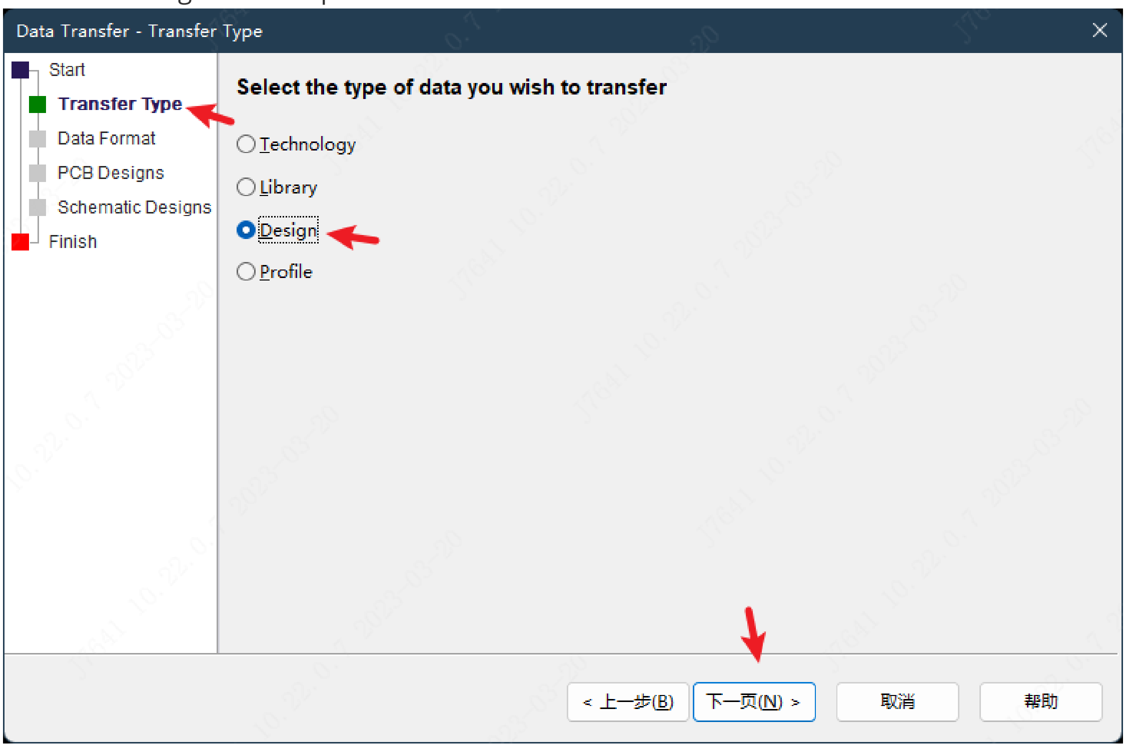
method:

1. Download and install Pulsonix and open it.

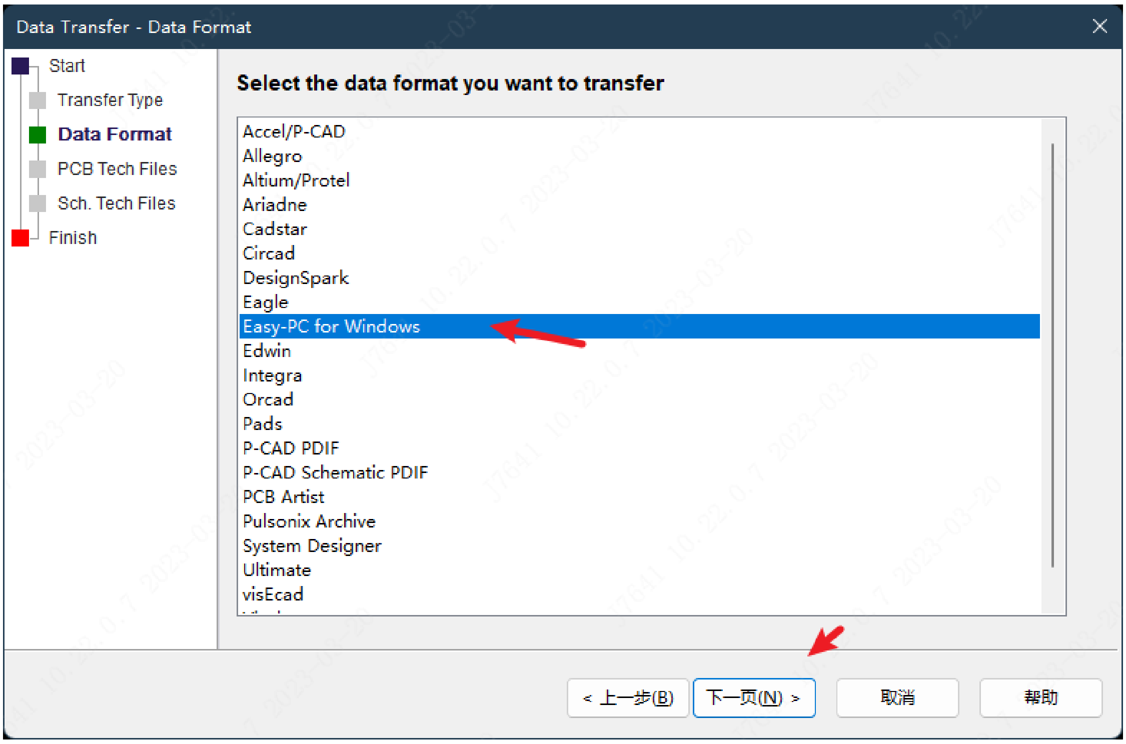
2. On the top menu - File - Data Transfer.



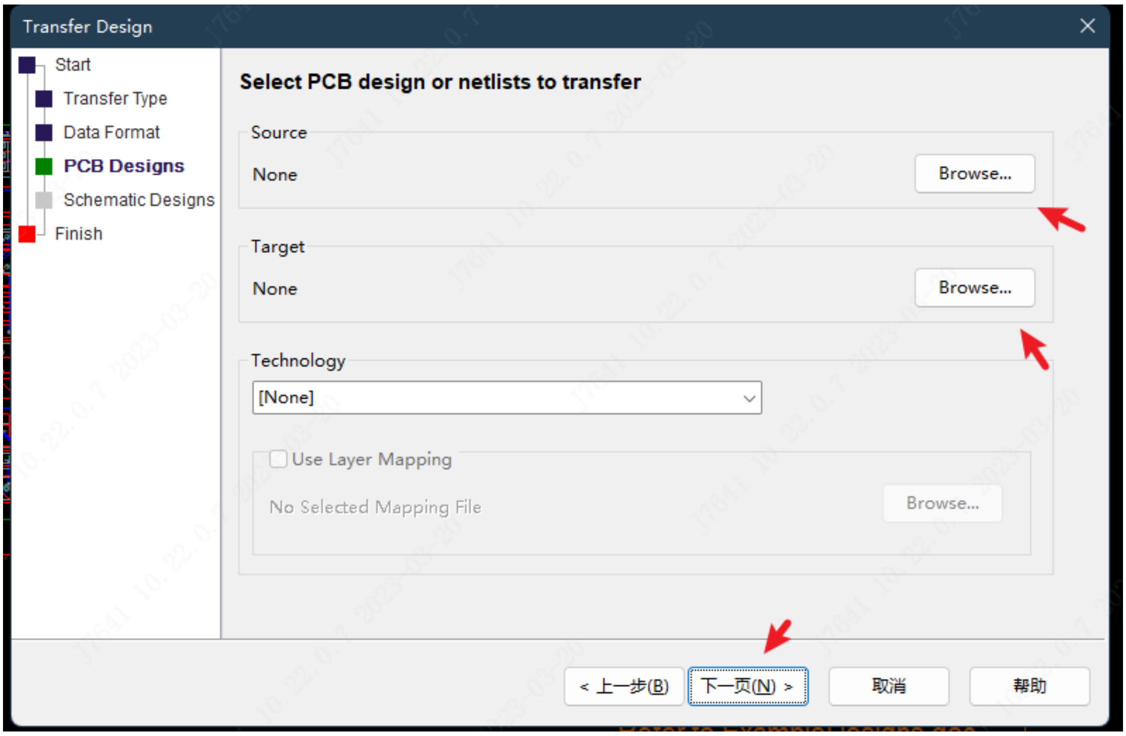
Select the design file to import:



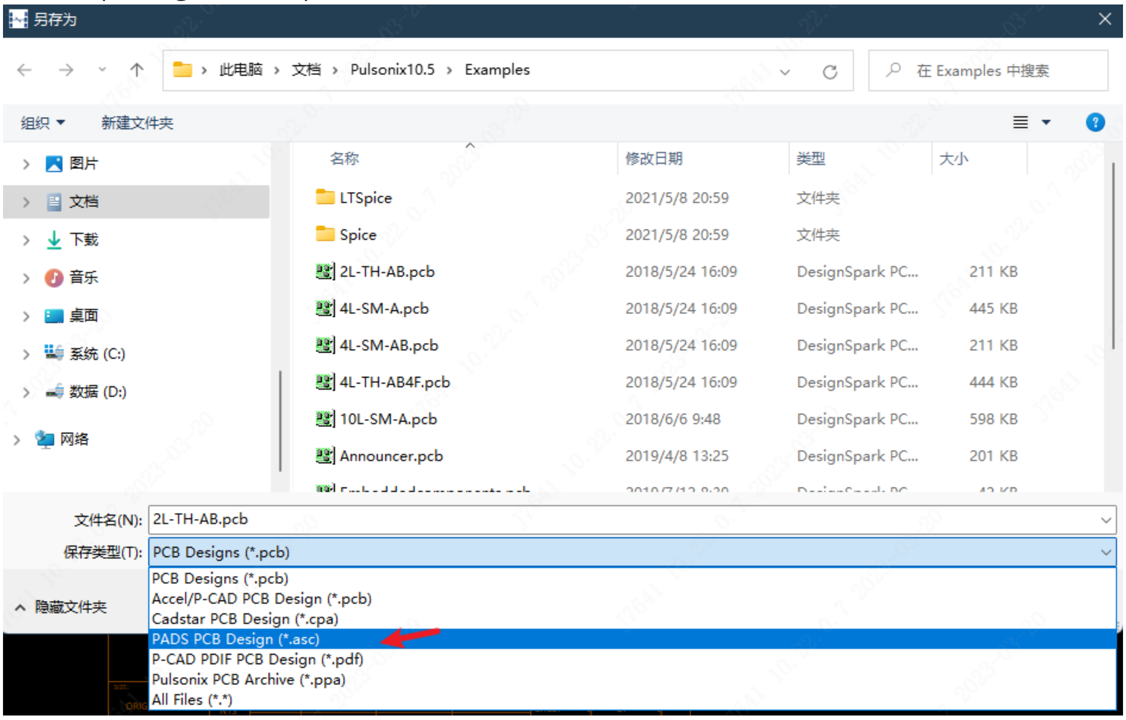
Select the Easy-PC file:



Set the directory where the file needs to be imported, and the directory where the output file is located, and then convert:



3. After importing, in the top file menu - file - save as, select PADS format to save.



4. Select the PADS file type in Jiali Chuang EDA to import it.

Notice:

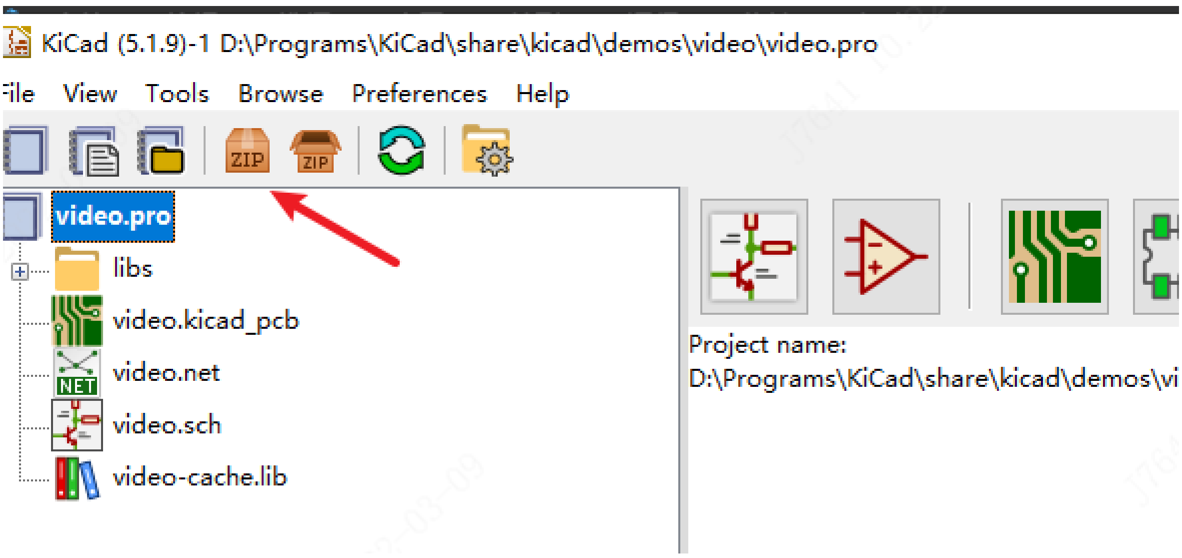
1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

Import KiCad

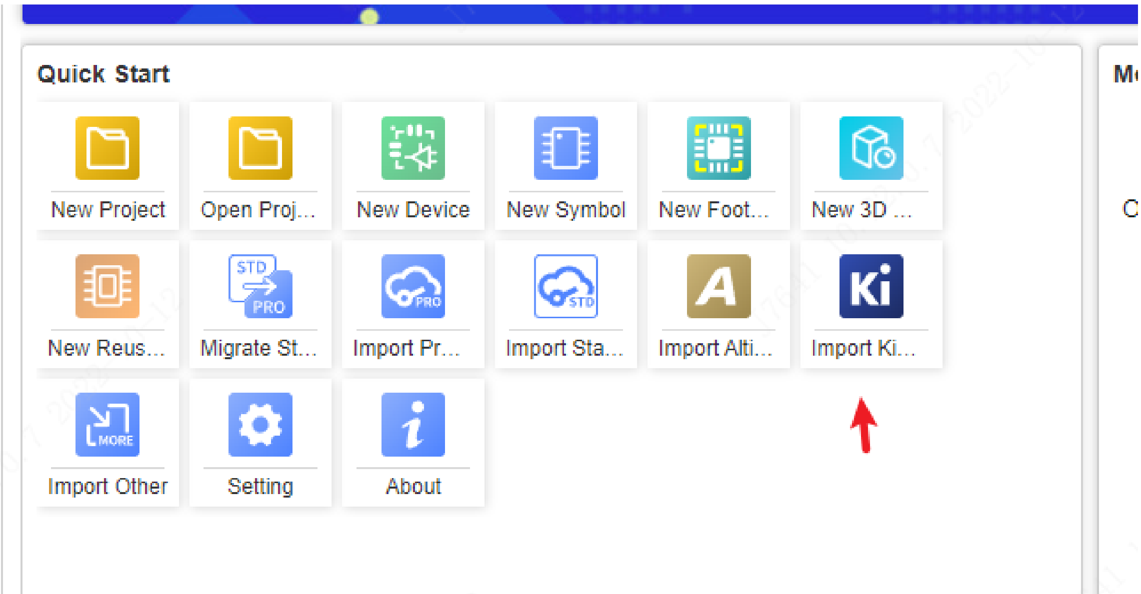
EasyEDA Pro supports importing files in KiCad 5.1 and KiCad 5.9 versions.

If it is an earlier version of KiCad file, please refootprint and import it after saving in 5.1.

1. After opening the project in KiCad, use the compression function to footprint it at the top



2. Find the footprintd zip file and import it on the start page of the professional version.



Direct import of library files is also supported.

Notice:

1. For KiCad project compression, please use the packaging function that comes with KiCad. Do not pack it in the folder yourself, because the built-in packaging function will automatically footprint the library files used in the schematic.
2. The KiCad version that is too low may not support it very well. Please save it again in the version above v5.1, and then footprint it into a zip file for import.
3. After PCB is imported, it will automatically rebuild and copper area. The result of copper area will be different, please check carefully.

Import PADS

EasyEDA Pro supports importing PADS 9.5 format files. You need to save or export an ASCII file using PADS or vx2.4. Select 9.5 as the format. The original file is an encrypted binary file and cannot be imported directly.
Importing PADS library files directly is also supported.

Procedure

Export schematic or PCB as ASCII

1. Open the PADS schematic diagram or PCB on PADS

2, in the top menu - File - Export, choose ASCII format export, schematic suffix TXT, PCB suffix ASC.

3. Open Calico EasyEDA, in the entry: Start page - Import others, select PADS. You are advised to package the schematic diagram and PCB into a zip package and import it.

! Figure 5

Export schematic library or pcb library as ASCII

1. Open the PADS software
2. In the top menu - File - Library, open the pop-up window of library Settings

3, select a library file, click to select encapsulation, symbol display preview

4, click the export button below the pop-up window, the schematic library will export the.c suffix file, PCB packaging library will export the.d suffix file

5. Find the exported file and import it to EasyEDA after packaging zip

Batch to ASCII

1 turn, download batch ASCII script: [] (https://image.lceda.cn/files/PadsScripts_2023.03.09.zip)

2. After decompression, install the script using the REAME.txt file
3. After running the script and selecting the file directory to be converted, all files under the directory, including subfolders, will be automatically converted
4. After the conversion is complete, find the output file in the output directory.

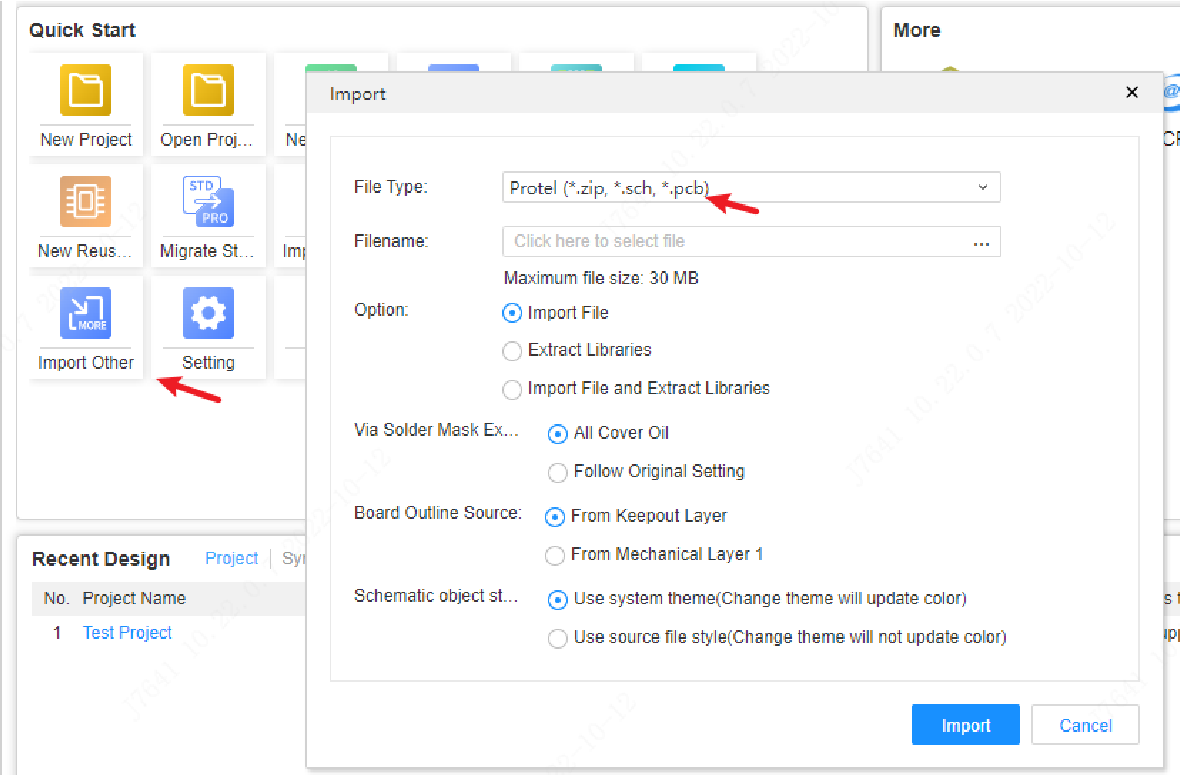
Attention

1. The definition of PCB layers in PADS is not very strict and flexible, so it is possible to import incorrect layers. You can contact us to deal with it.
2. The format of text and copper paving is different from the design, so it needs to be carefully checked and fine-tuned after import.

Import Protel

EasyEDA Pro supports importing Protel.

Entry: Start Page - Import Others, select Import Protel.

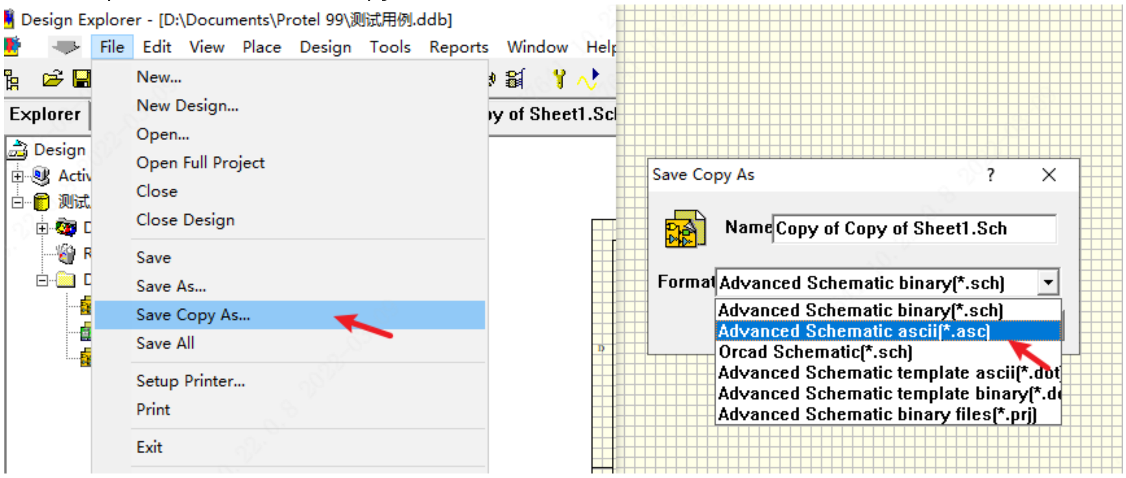


Currently, only Protel 99 SE files in ASCII format can be imported. Please use Protel to export to ASCII format before importing.

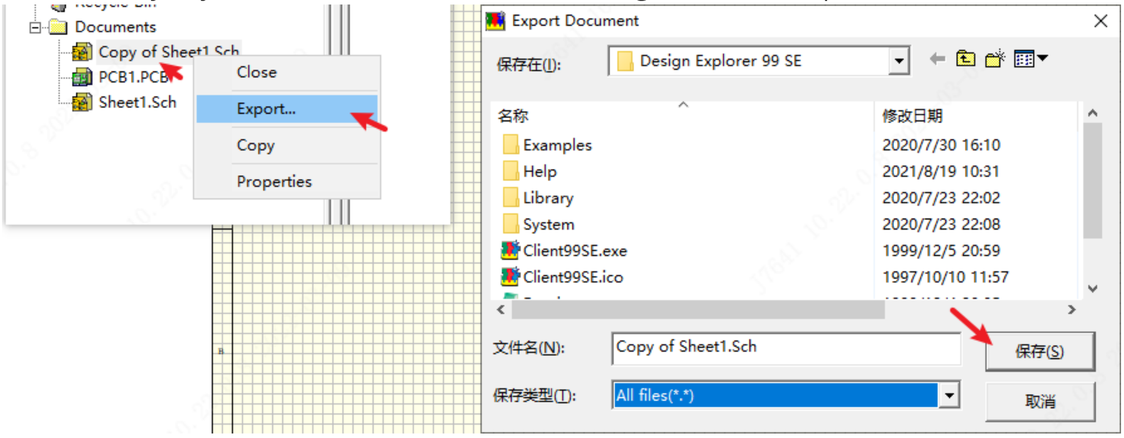
step:

1. Protel 99SE open the project, open the schematic diagram or PCB

2. In the top file menu, save a copy as an ASCII file



3. Find the file you just saved as in the list on the left, right-click and export to a local folder



4. After finding the exported file, compress the schematic and PCB into a zip file, and then import it into EasyEDA Pro.

If it is a Protel file of an earlier version, it may not be imported normally. It is recommended to re-export it in Protel 99 SE first.

Notice:

1. Please compress the schematic diagram and PCB into ZIP and import them together, otherwise there will be no footprint association when importing the schematic diagram alone.
2. There are slight differences in text and other graphic elements, please check carefully.
3. After PCB is imported, it will automatically rebuild and copper area. The result of copper area will be different, please check carefully.

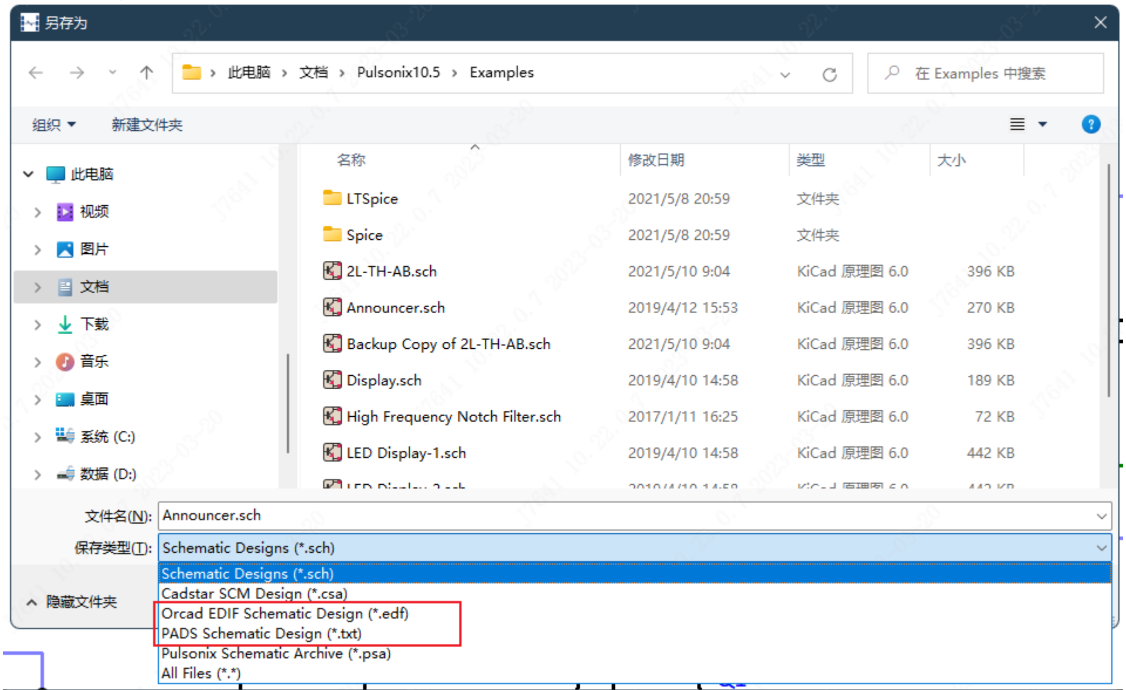
Import Pulsonix

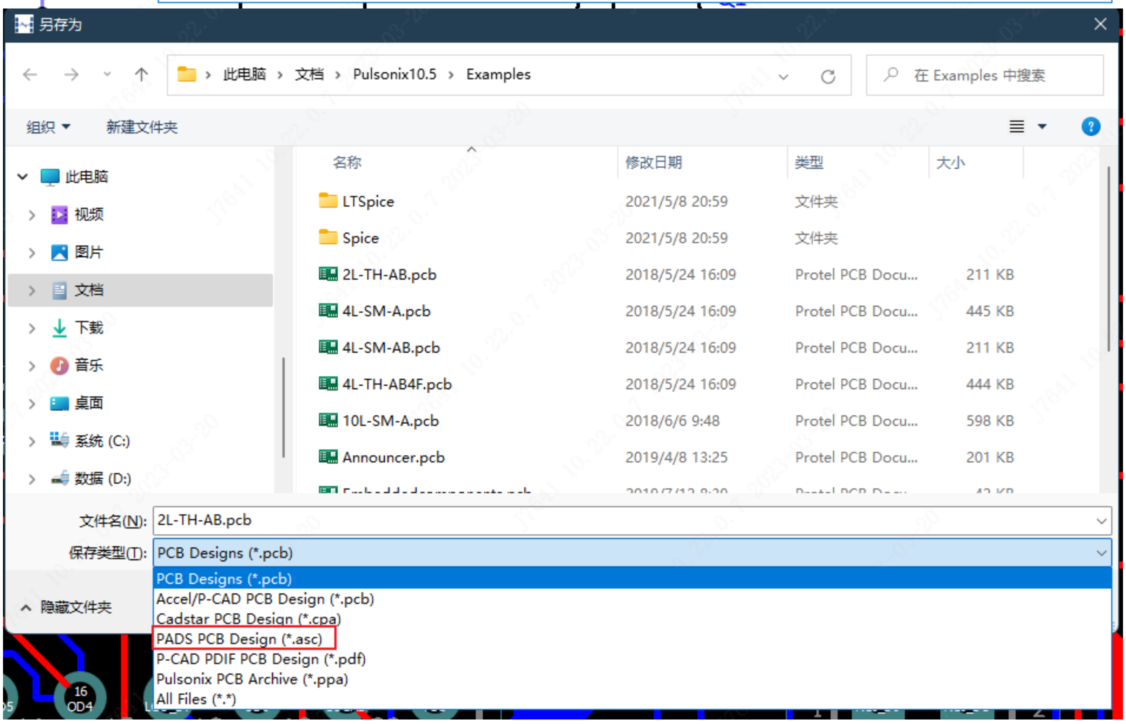
EasyEDA Professional Edition does not support direct import of Pulsonix format files. But you can save it in other formats first and then import it.

method:

1. Use Pulsonix to open the schematic or PCB

2. On the top menu - File - Save As. Select Orcad or PADS format file.





3. Select the corresponding file type in Jiali Chuang EDA to import, Orcad or PADS file import.

Notice:

1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

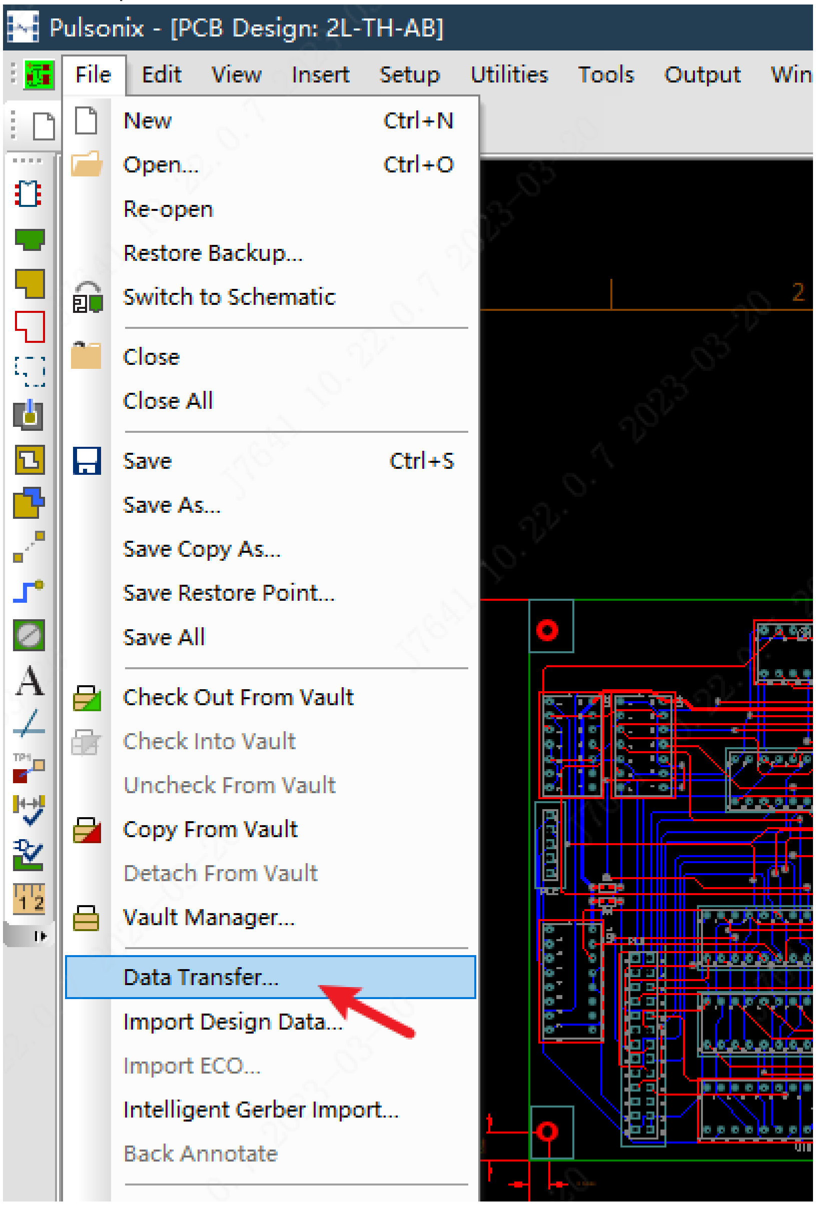
Import PCB Artist

EasyEDA Professional Edition does not support direct import of PCB Artist format files. But you can use other EDA tools to convert it once and then import it.

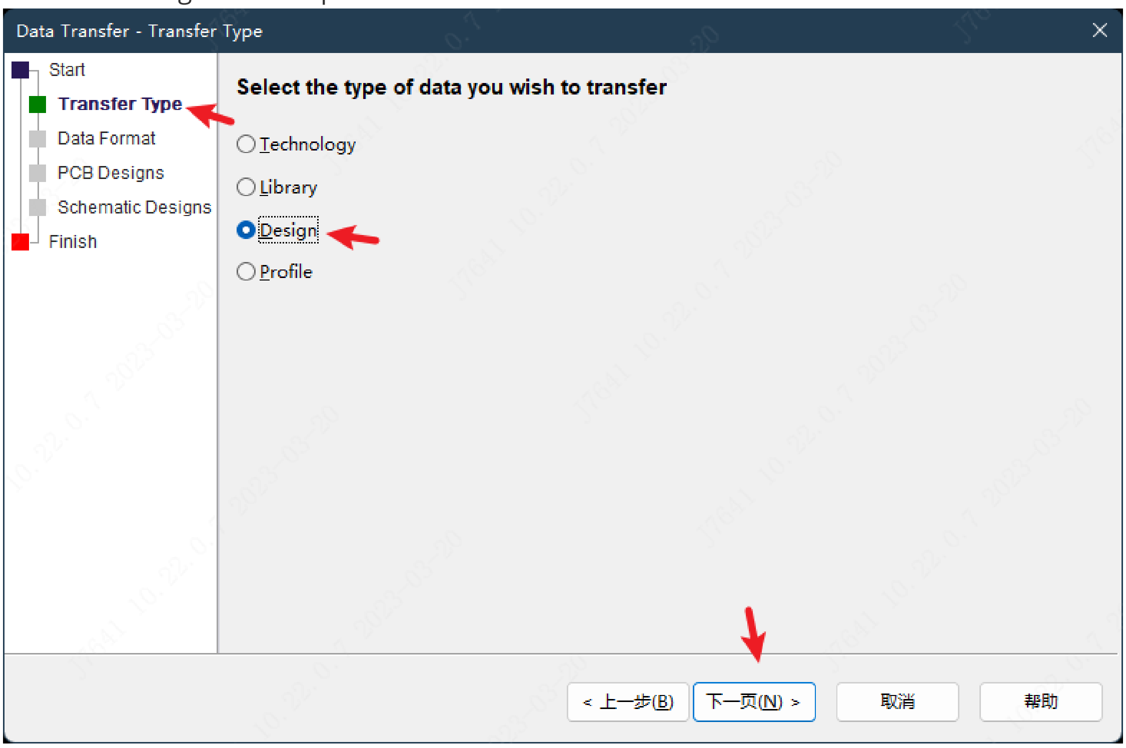
method:

1. Download and install Pulsonix and open it.

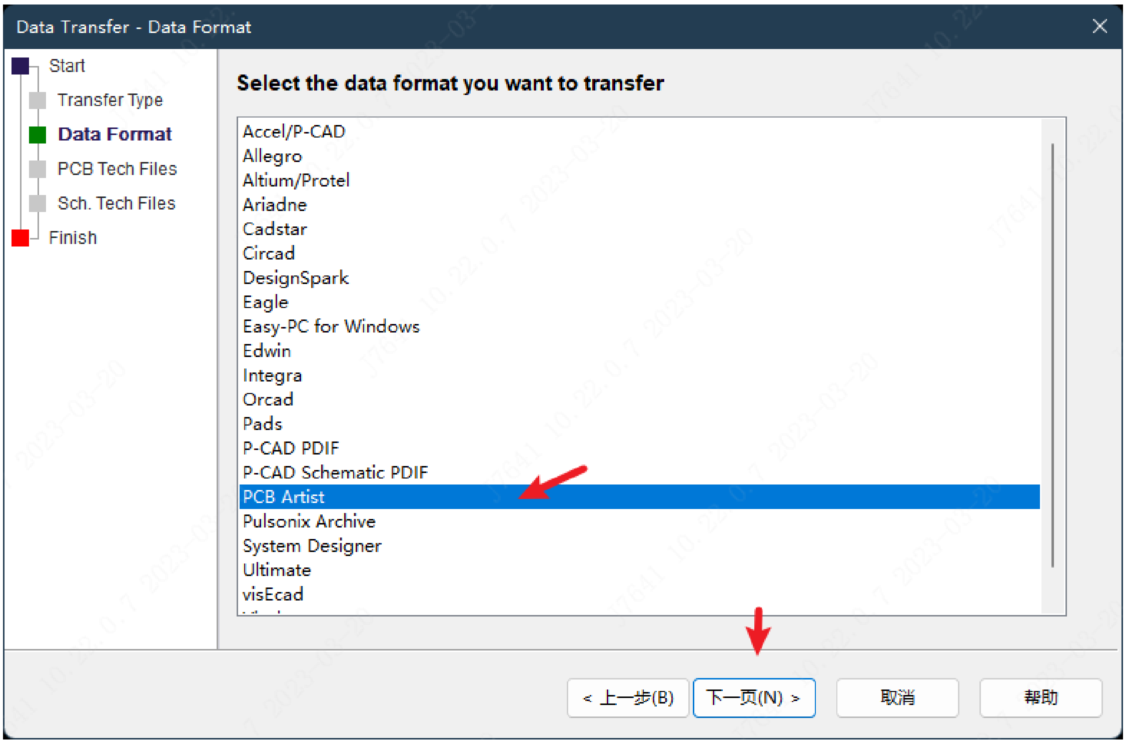
2. On the top menu - File - Data Transfer.



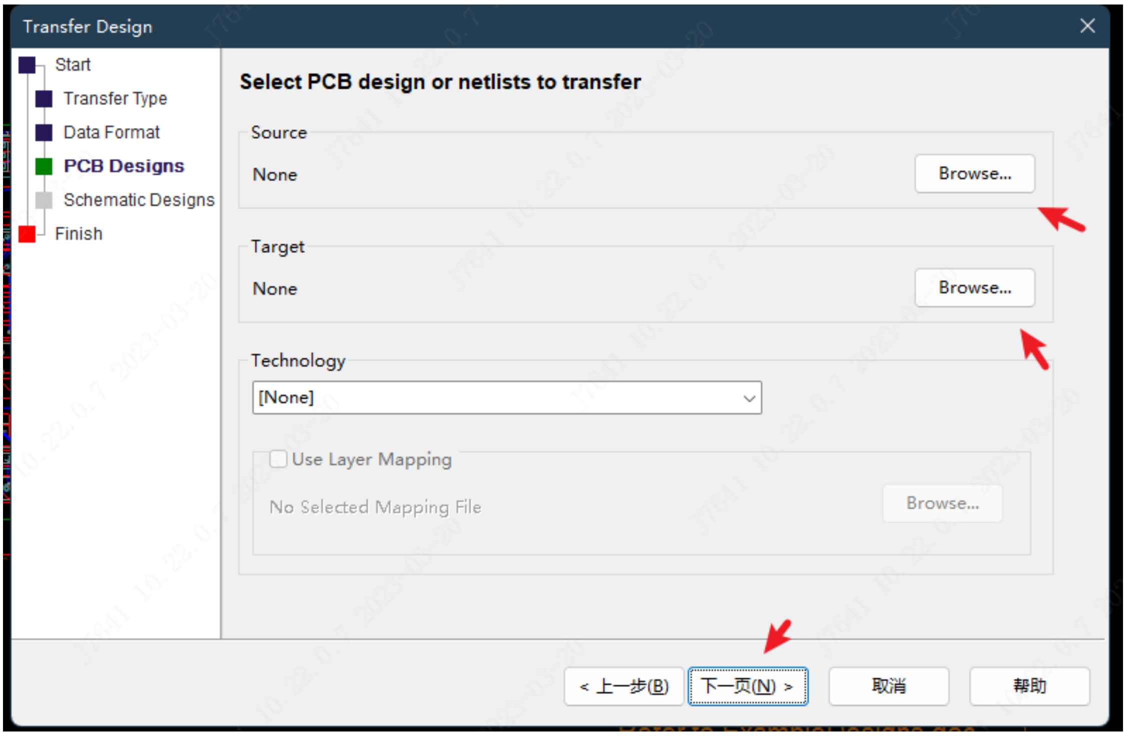
Select the design file to import:



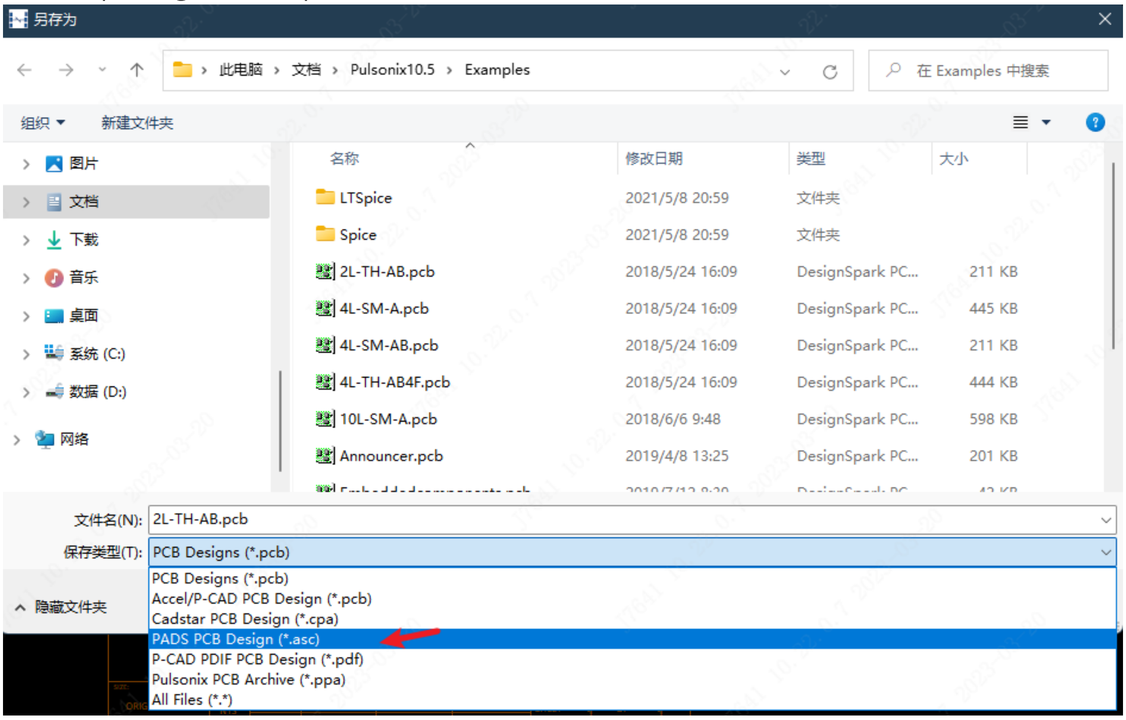
Select the PCB Artist file:



Set the directory where the file needs to be imported, and the directory where the output file is located, and then convert:



3. After importing, in the top file menu - file - save as, select PADS format to save.



4. Select the PADS file type in Jiali Chuang EDA to import it.

Notice:

1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

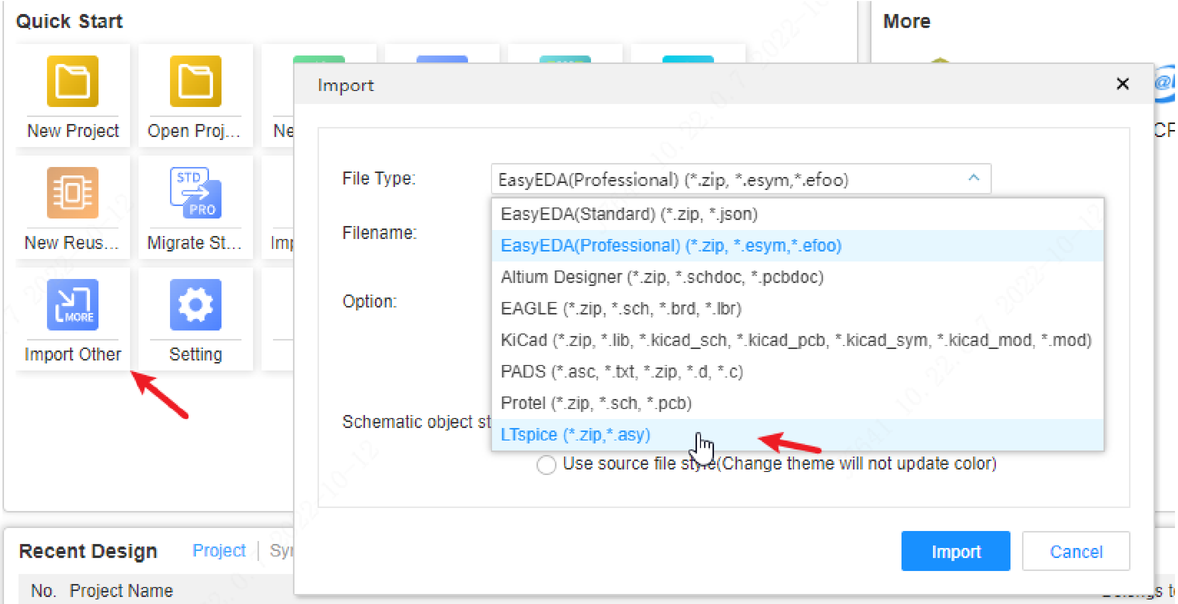
Import LTspice

EasyEDA Pro supports importing LTSpice files, please make sure your LTspice files are saved in a newer version.

Only LTspice files in ASCII format are supported, encrypted files cannot be imported.

Before importing, please pack the schematic asc file of LTspice and the corresponding symbol file asy file into a ZIP archive and import. Because the schematic diagram of LTspice does not contain symbol content, the schematic file cannot be directly imported.

Entry: Start Page - Quick Start - Import Other - Import LTspice.
Or after opening the file: Top Menu - File - Import - LTspice

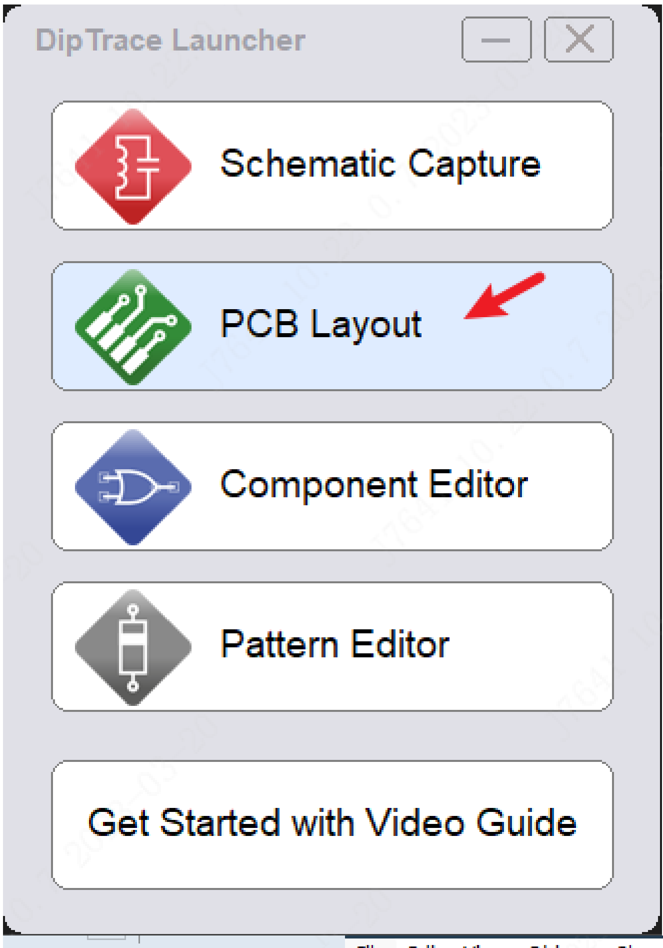


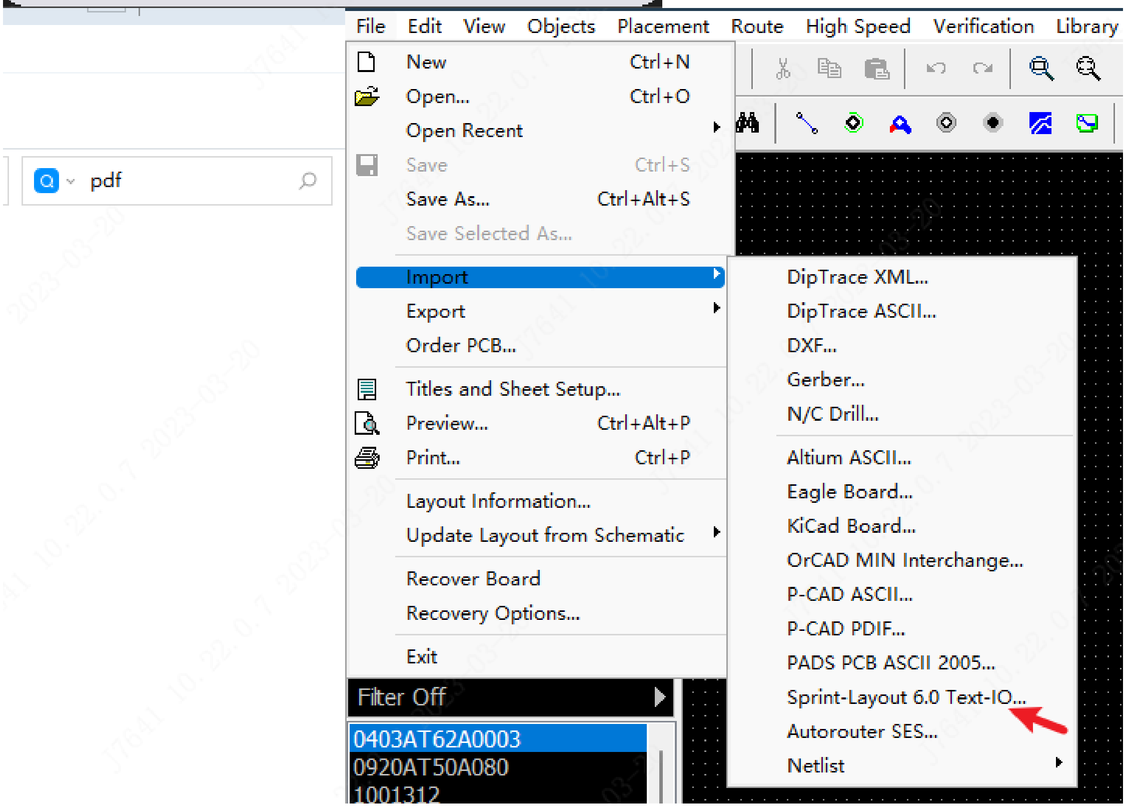
Import Sprint Layout

EasyEDA Professional Edition does not currently support the direct import of Sprint Layout format files. But it can be imported again through other EDA tools.

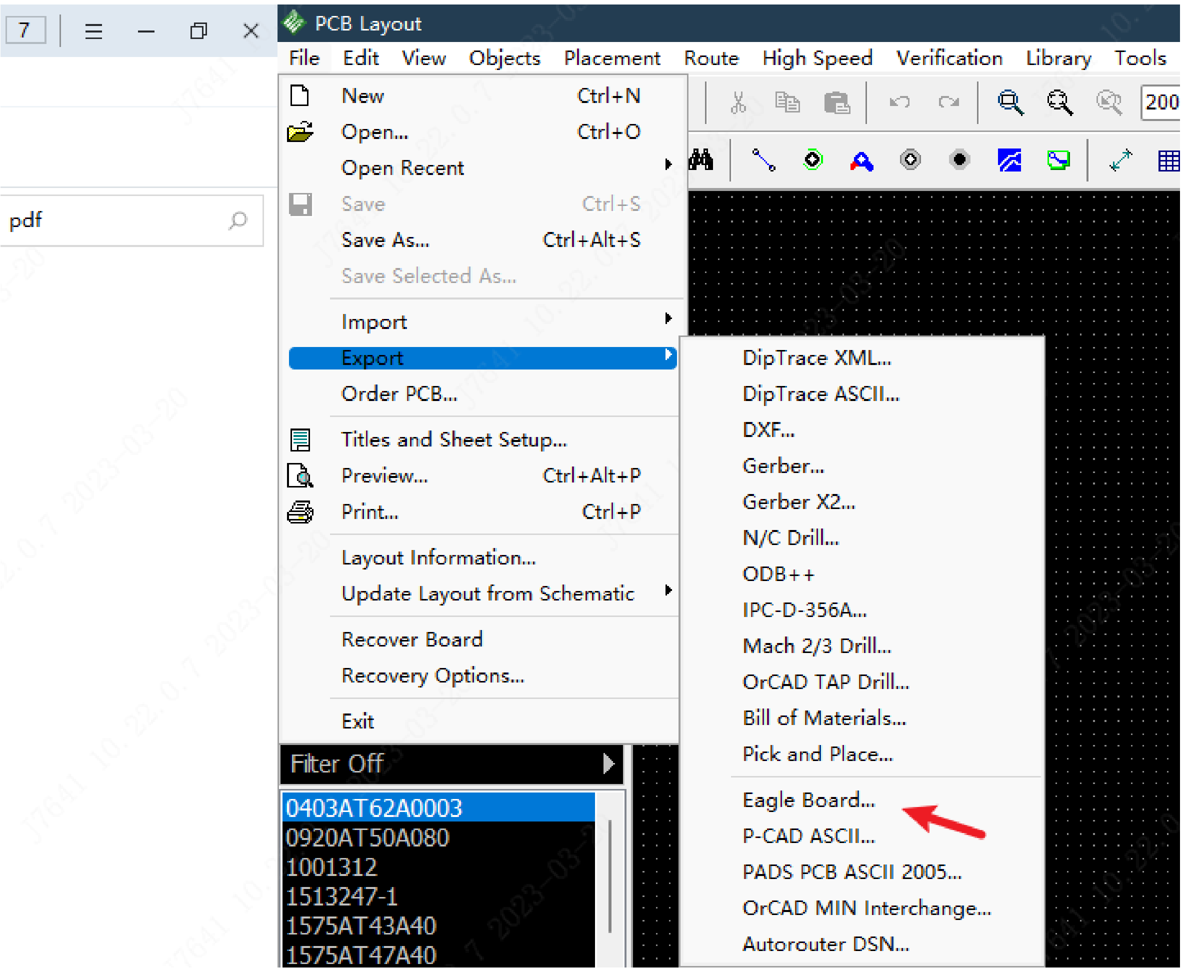
method:

1. Use Diptrace to import the Sprint Layout file
2. Open the PCB module, select the PCB file import of Sprint Layout, and select the import menu in the top menu.





3. After importing, go to File menu - Export - Eagle Board file



4. Then import the Eagle file into Jiali Chuang EDA

Notice:

1. Because the function and design of the graphic elements are different, it is impossible to ensure that they are completely consistent before and after importing. Please check carefully.

Export Altium Designer

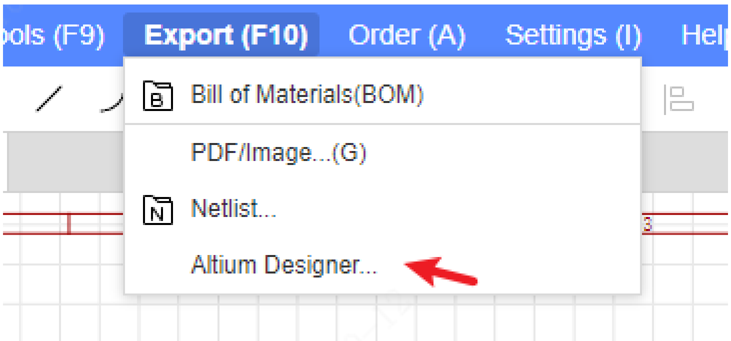
Disclaimer

1. Because the format design and graphics elements are different, the exported Altium Designer files cannot be guaranteed to be completely consistent (especially the text size and position). Please be sure to check carefully after opening Altium Designer, and do not place an order without checking. EasyEDA is not responsible for any problems or losses caused by differences in the export.
2. If you check the exported file and the original design is very different, please contact us, we will try our best to correct it.

export project

EasyEDA supports exporting Altium Designer files in ASCII 5.0 format.

Entry: Top Menu - File - Export - Altium Designer. Or: Top Menu - Export - Altium Designer



The file format of Altium Designer will be exported based on all documents of the current project and footprintd in a compressed footprint.

Known issues:

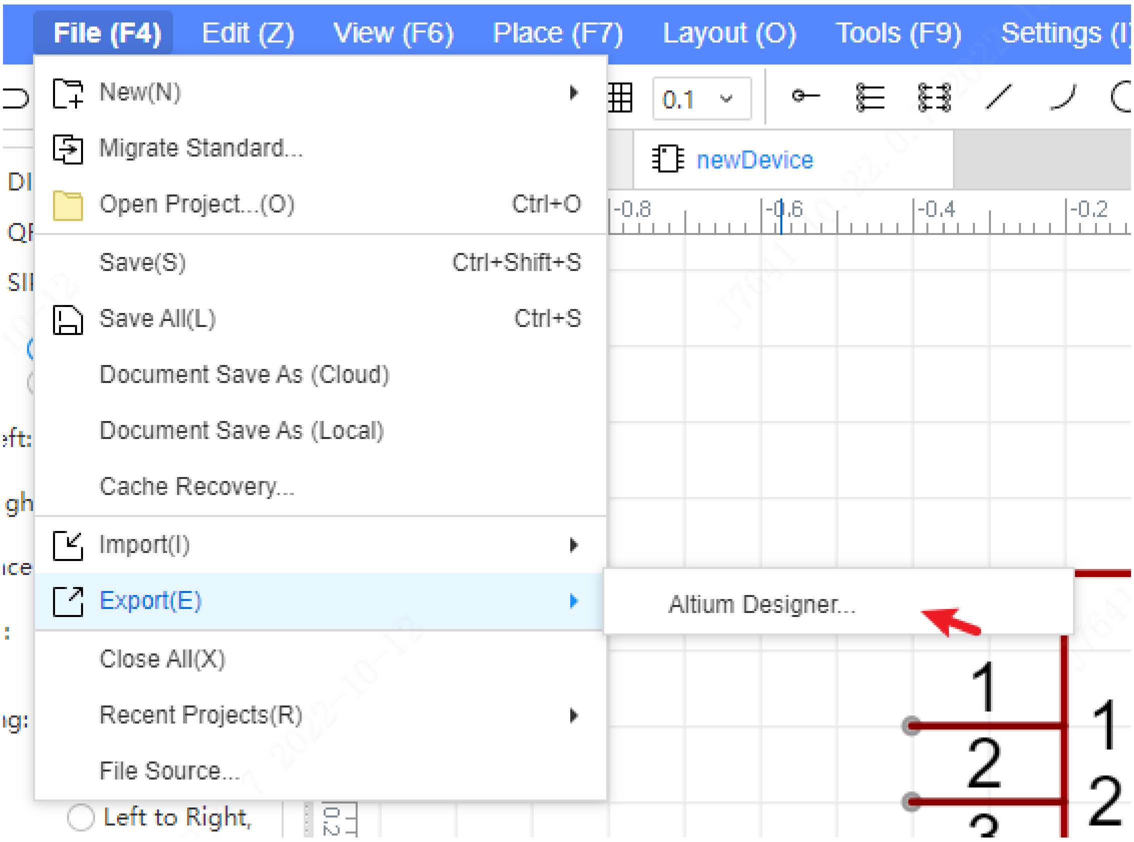
1. The image of the schematic diagram does not support export.
2. The image of PCB does not support export.
3. The rules of PCB do not support exporting.
4. The copper filling of the PCB does not support export. After exporting, you need to manually rebuild all the copper in Altium Designer.
5. The export of the inner electrical layer area of the PCB cannot be completely consistent, and it needs to be manually adjusted and reconstructed.
6. Some layers of the PCB cannot accurately find the corresponding layer of Altium, and will be exported in more mechanical layers.

Export library file

EasyEDA Pro support direct export to Altium Designer format from symbol library and footprint library. Or you can export by exporting the project.

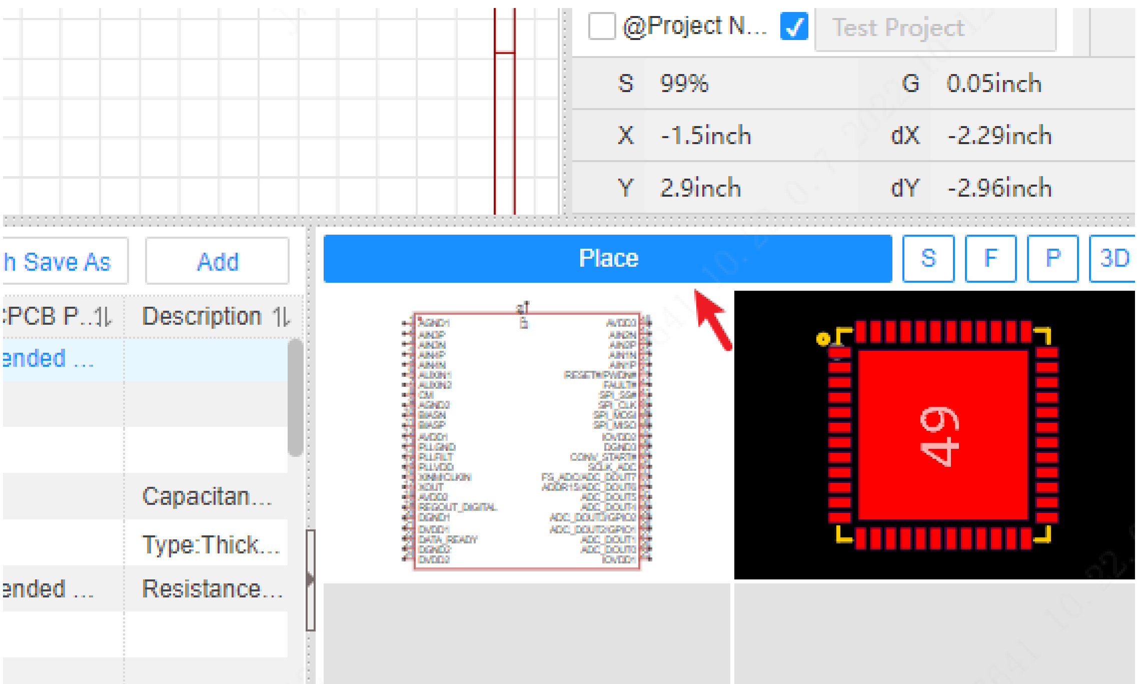
Method 1:

1. Edit the symbol or footprint graphic
2. Export as Altium file via: Top Menu - File - Export - Altium Designer



Method 2:

1. Create a new project
2. Find the corresponding device according to the library name or product number you want to export, and place it on the schematic or PCB. Shortcut SHIFT + F, or bottom bank, shortcut S.



3. After placing the device on the schematic and PCB, save it. Export menu at the top - export Altium Designer.
4. Export the Alitum Designer zip footprint will contain the schematic and PCB, and open it in Altium Designer after decompression.
5. After Altium Designer opens the schematic diagram or PCB, go to: Top Menu - Design - Extract Library File to complete the library extraction.

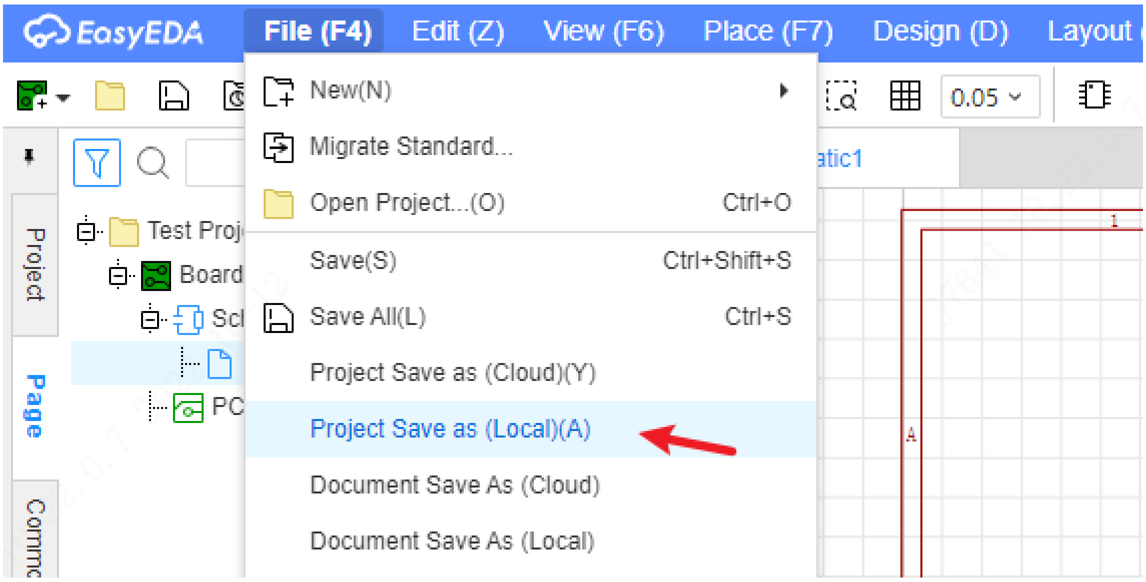
Export EasyEDA Pro

EasyEDA Pro supports saving project files to local files. The created project is saved in the cloud by default, and it needs to exist locally and needs to be operated by the user.

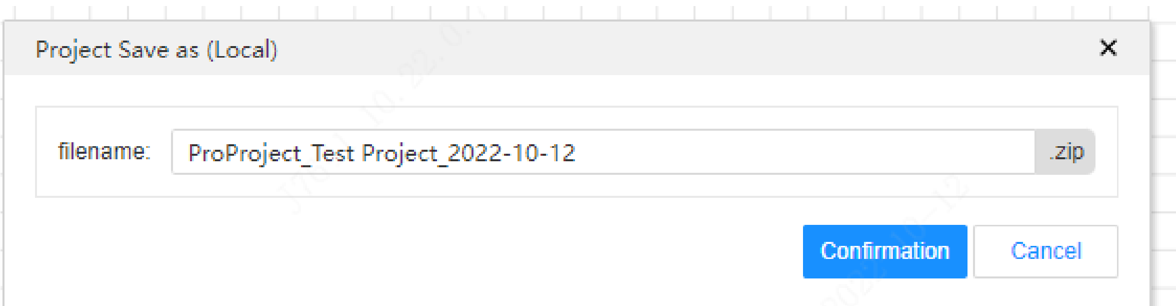
Export to EasyEDA Standard format is not supported.

Export process:

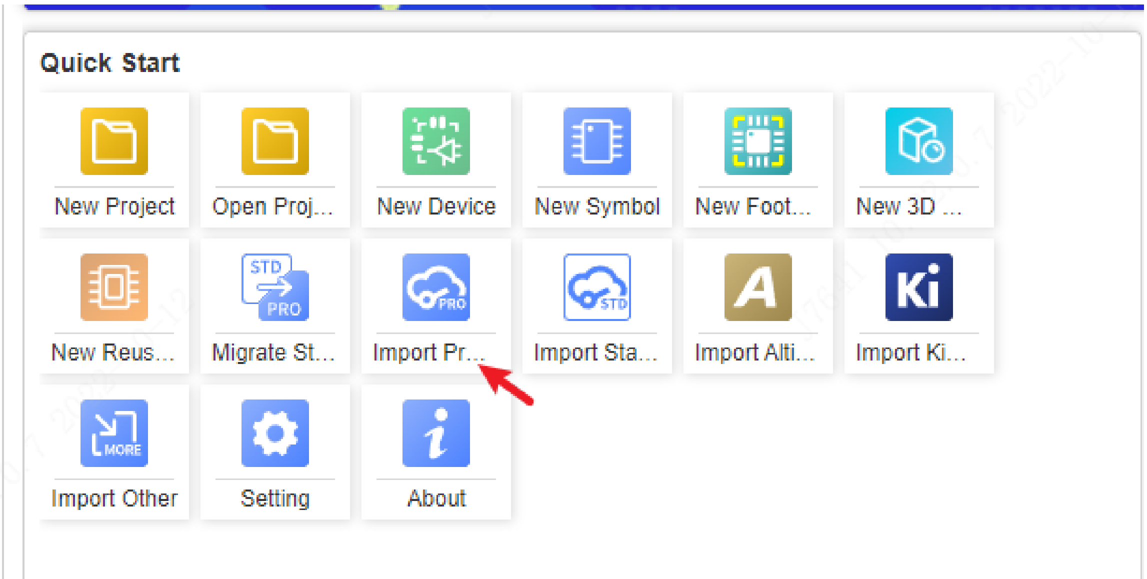
- Top Menu - File - Project Save as (local)



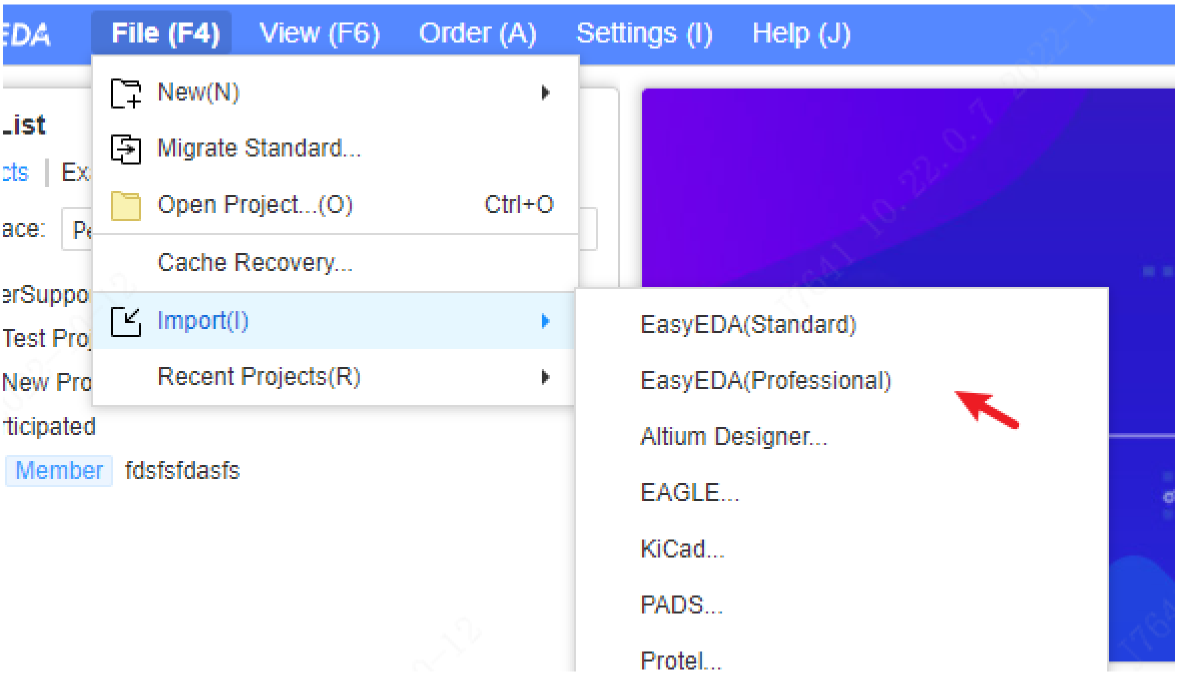
After clicking, the files in the project can be compressed to the local, and the compressed footprint includes the device library and footprint placed in the engineering schematic diagram.



On the editor start page, you can import the project archive zip file which is saved locally.



or at Top Menu - File - Import - EasyEDA Professional



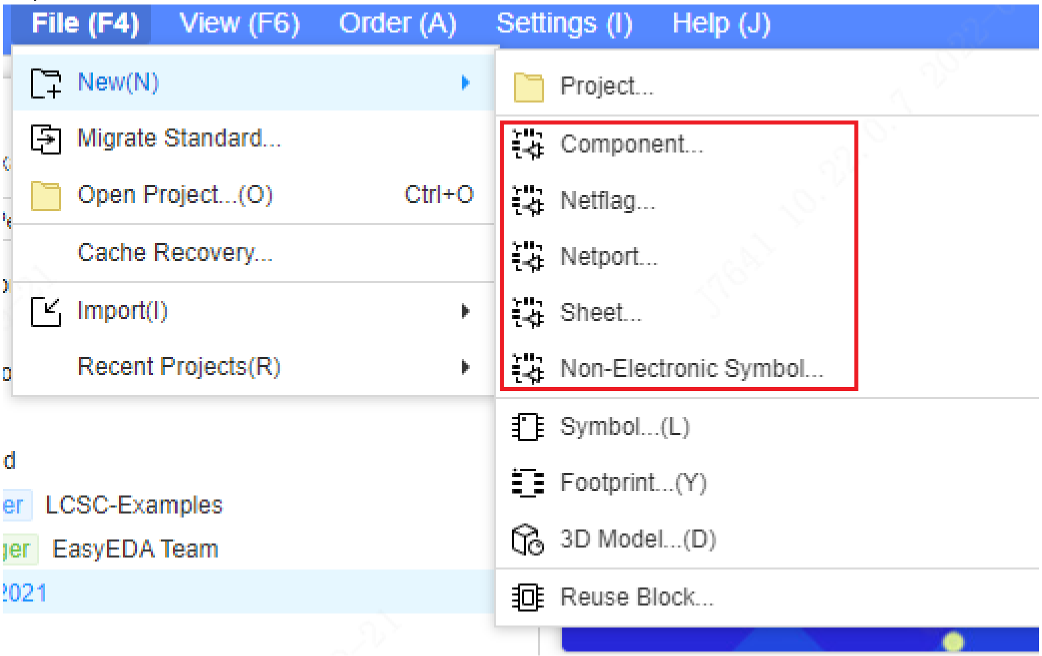
Device Library

Create Device

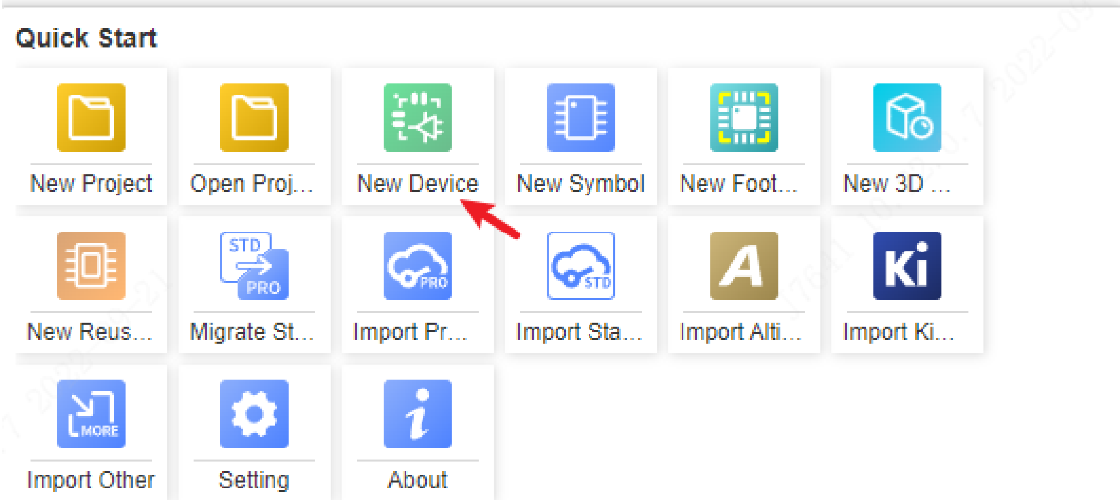
The logical order of creating a device in EasyEDA Pro is: device = symbol + footprint + 3D model + properties

New Device Steps

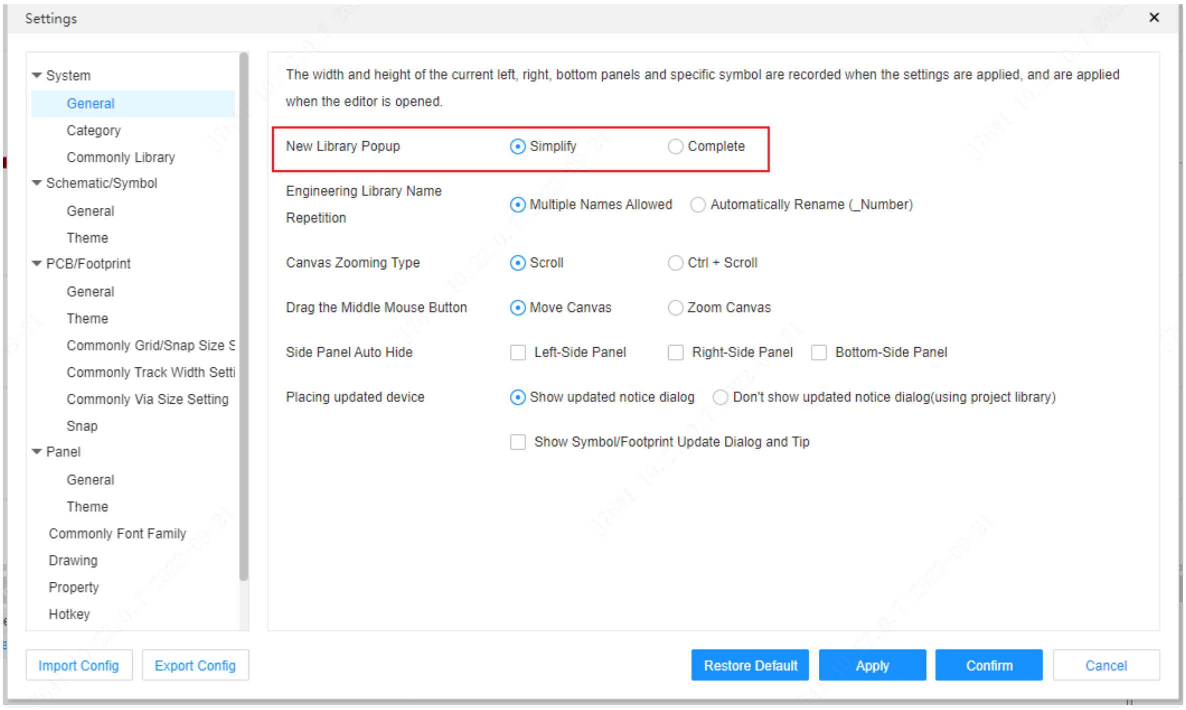
- Top Menu - New - Device



- Start Page - Quick Start - New Device

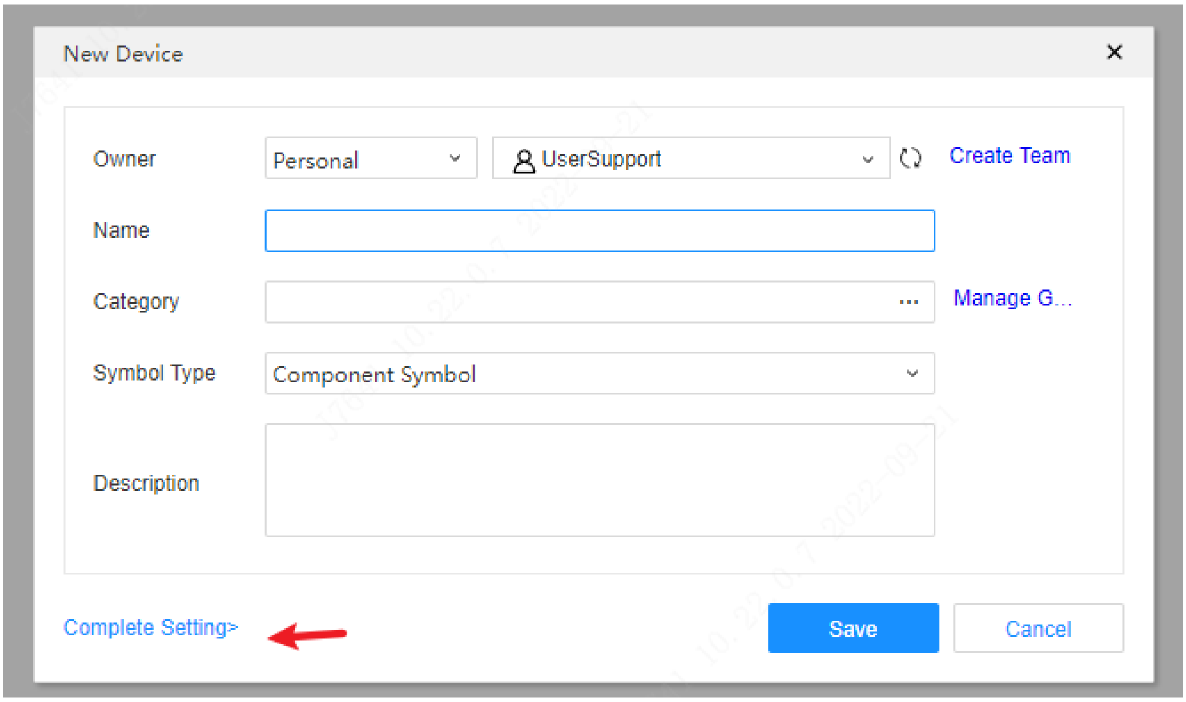


The new device dialog has two setting: Simplify and Complete Setting, you can use it as you want, and you can set it as default at Setting - System - General:

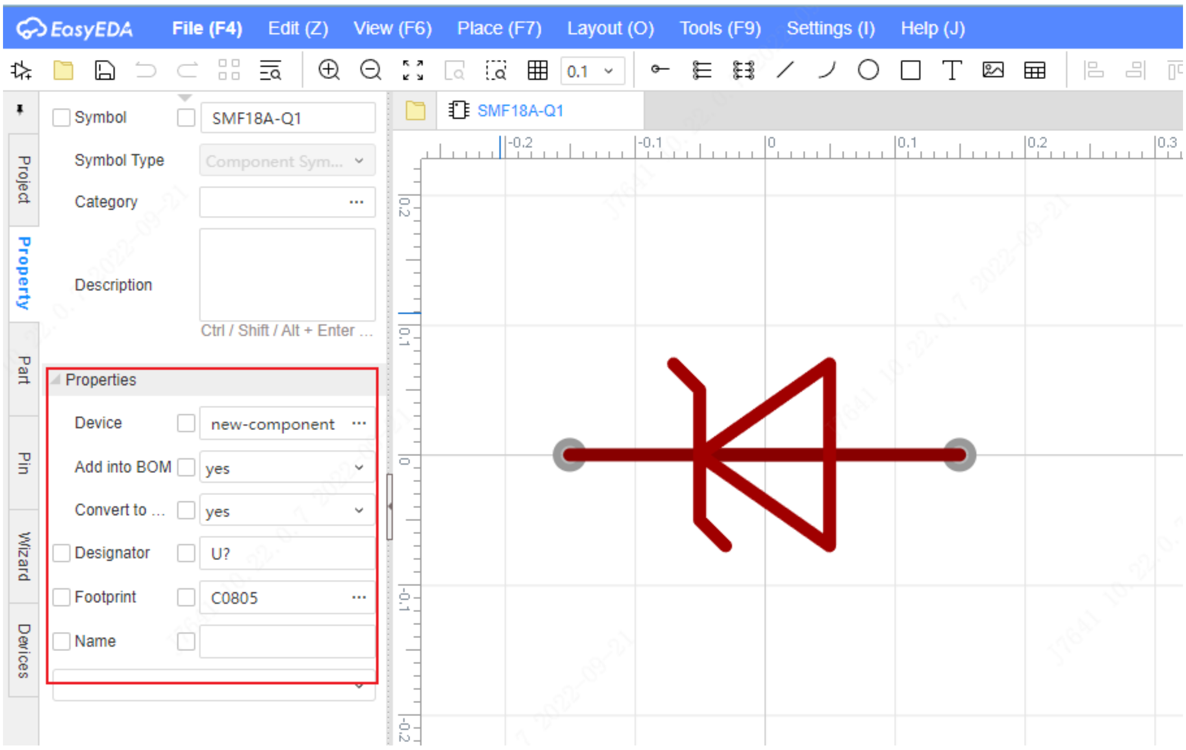


Simplify Setting Dialog

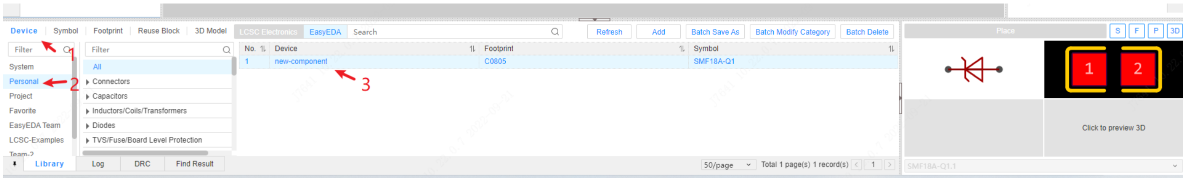
The simplify setting dialog just need to set some important properties.



when you click save button, the editor will open a new symbol, you can draw the symbol directly, and set the device properties at left-side panel



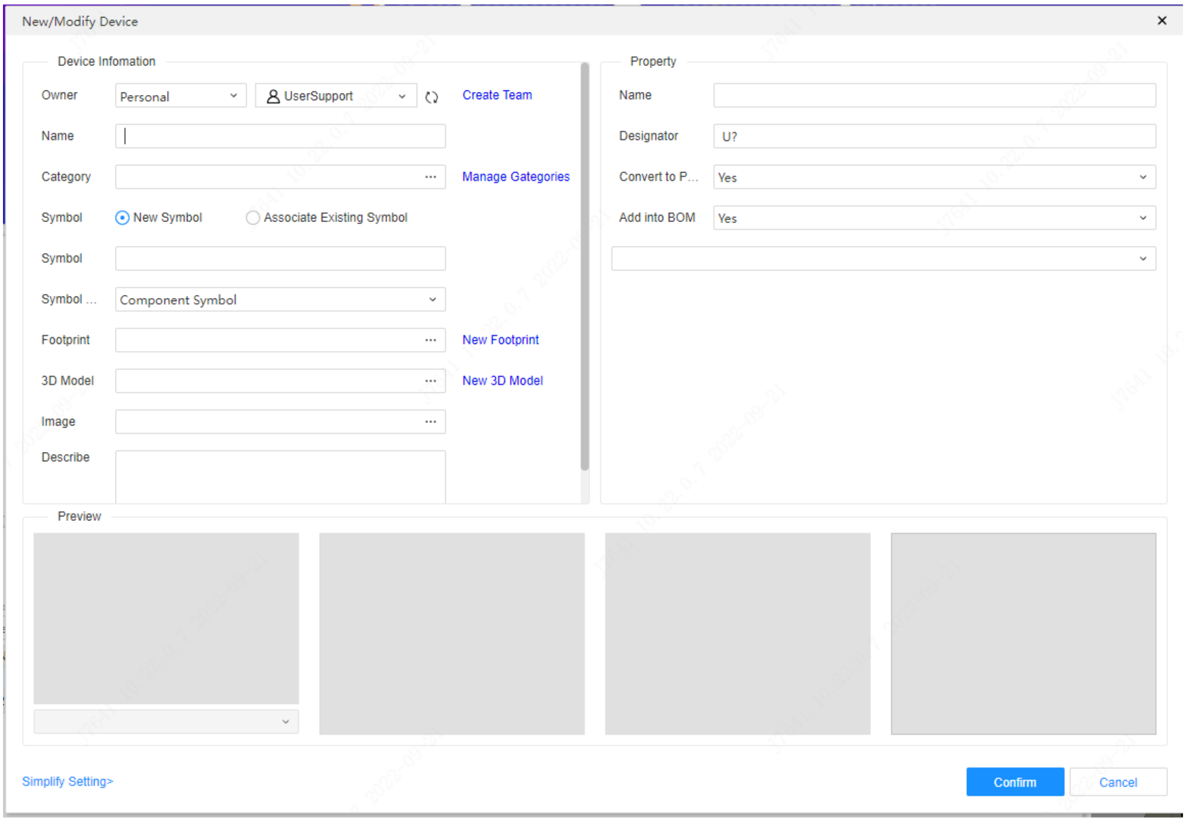
when you finish, you can find your new device at bottom library panel, at the device list, you can right-click it and modify the device information



Complete Setting Dialog

Click Create to pop up the device properties pop-up window, use the complete setting to fill in the name of the device and the team or individual to which the device belongs.

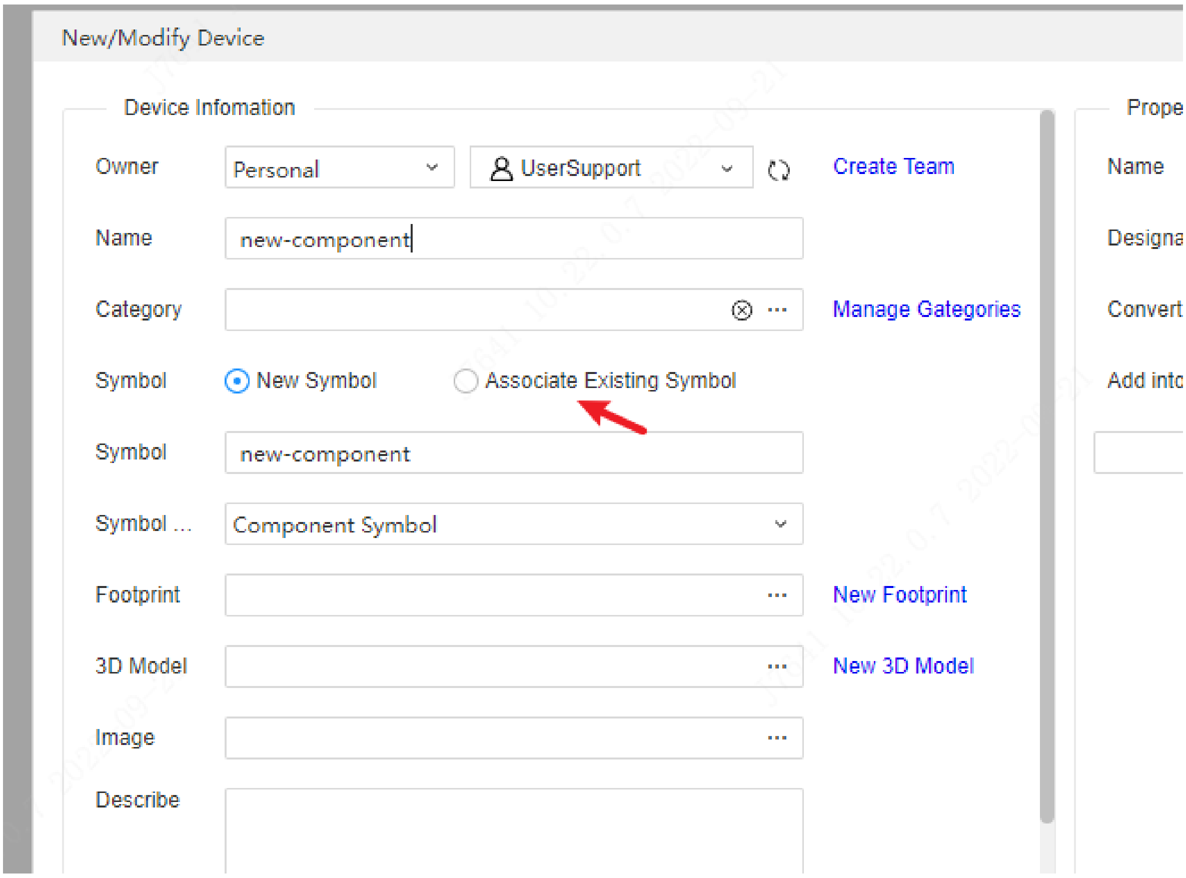
- **Owner**: The device is owned by the user or team.
- **Name**: The name of the device.
- **Symbol**: Click to bind the device.
- **footprint**: Click to bind the footprint to the device.
- **3D Model**: Click to select the model bound to the device.
- **Picture**: Click to bind a real picture to the device.
- **Catagory**: Click to classify the currently bound device.
- **Description**: A description of the device.



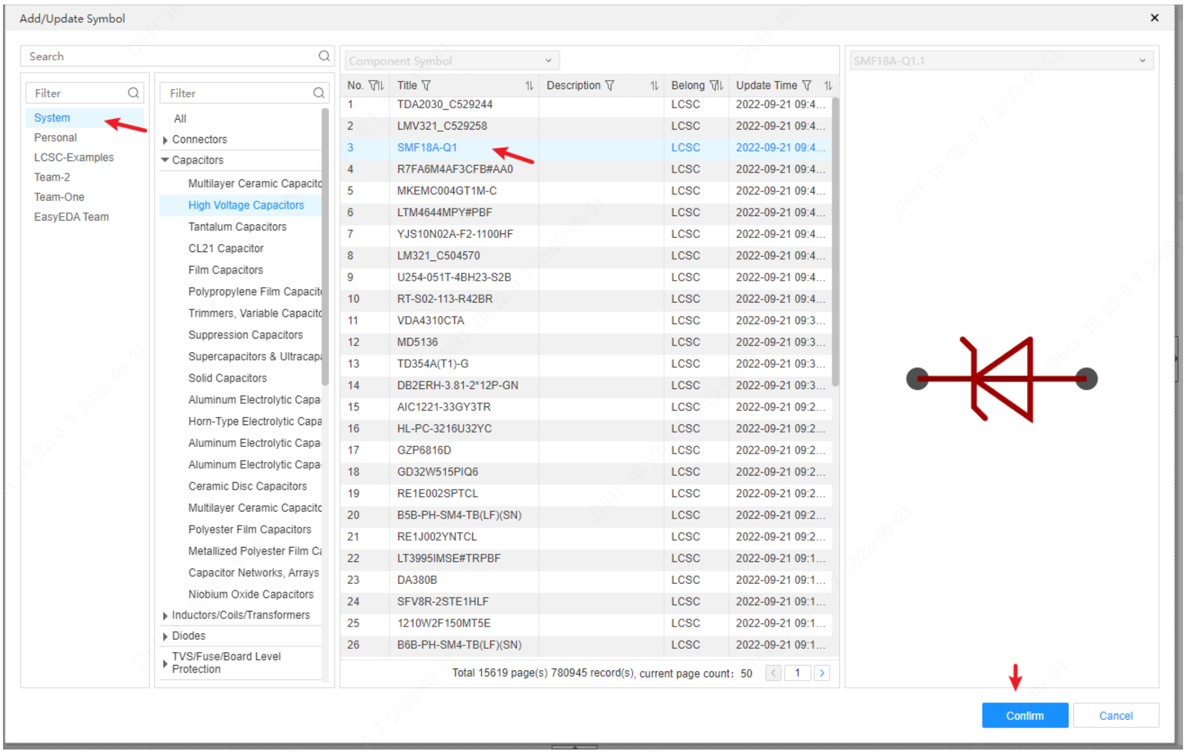
Add Symbols

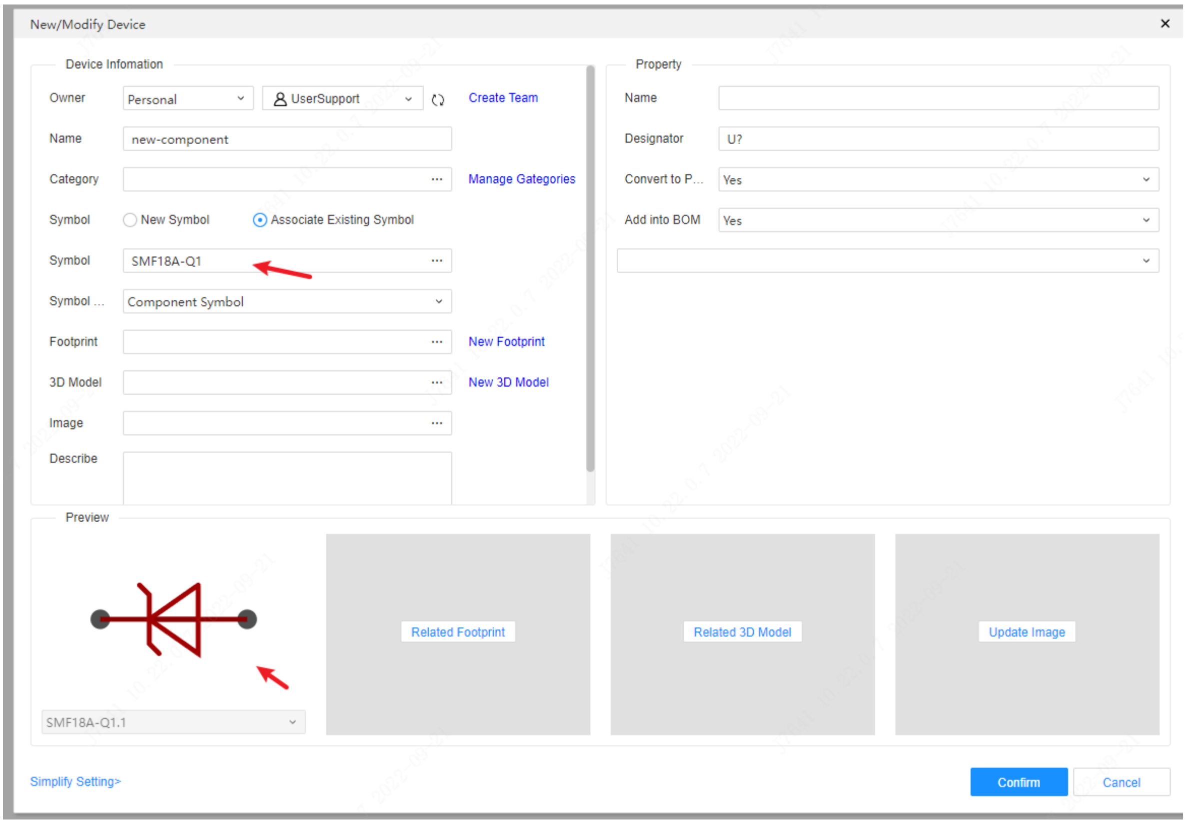
if you use the "Simple Setting" dialog, create the device will create the symbol automatically, if you use "Complete Setting", you can create or binding existed symbol.

Click the symbol's checkbox or the preview area below to add a symbol library to the device.



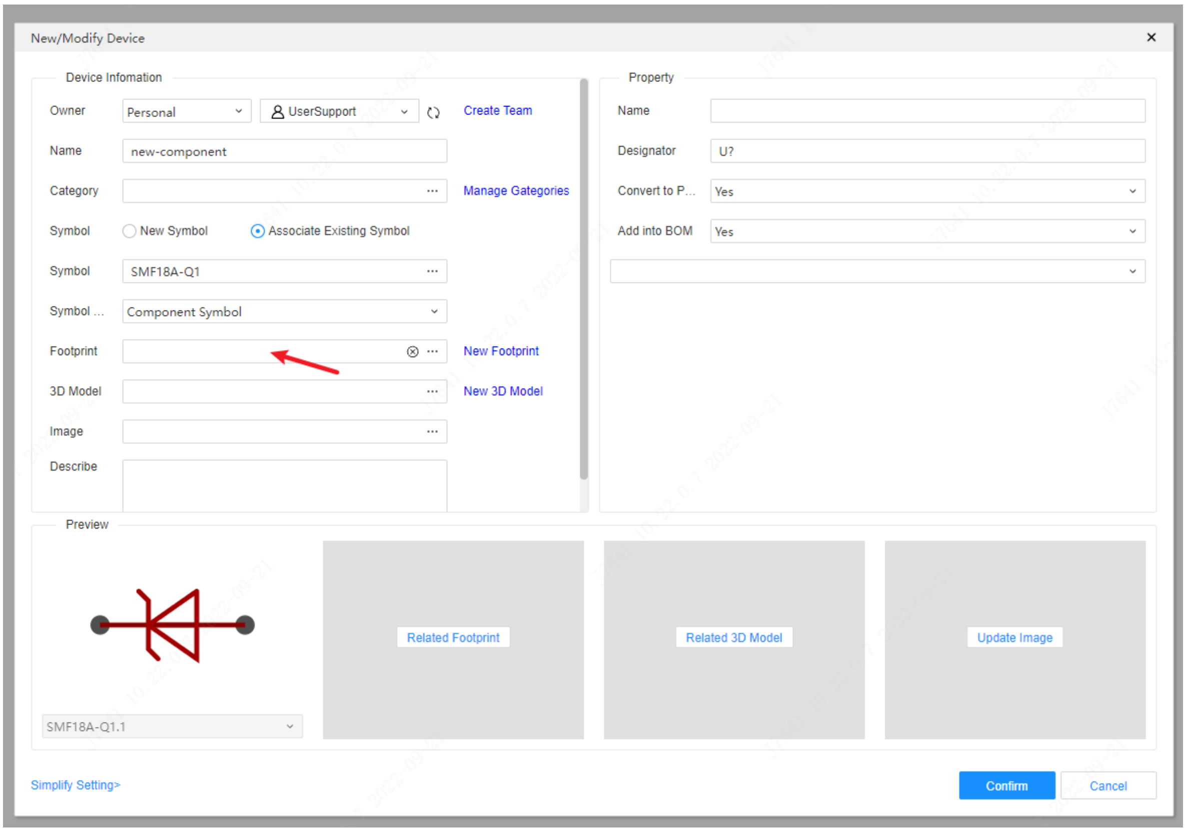
After clicking, a pop-up window will pop up, find the symbol of the device that needs to be bound, and click OK to complete the binding of the symbol.



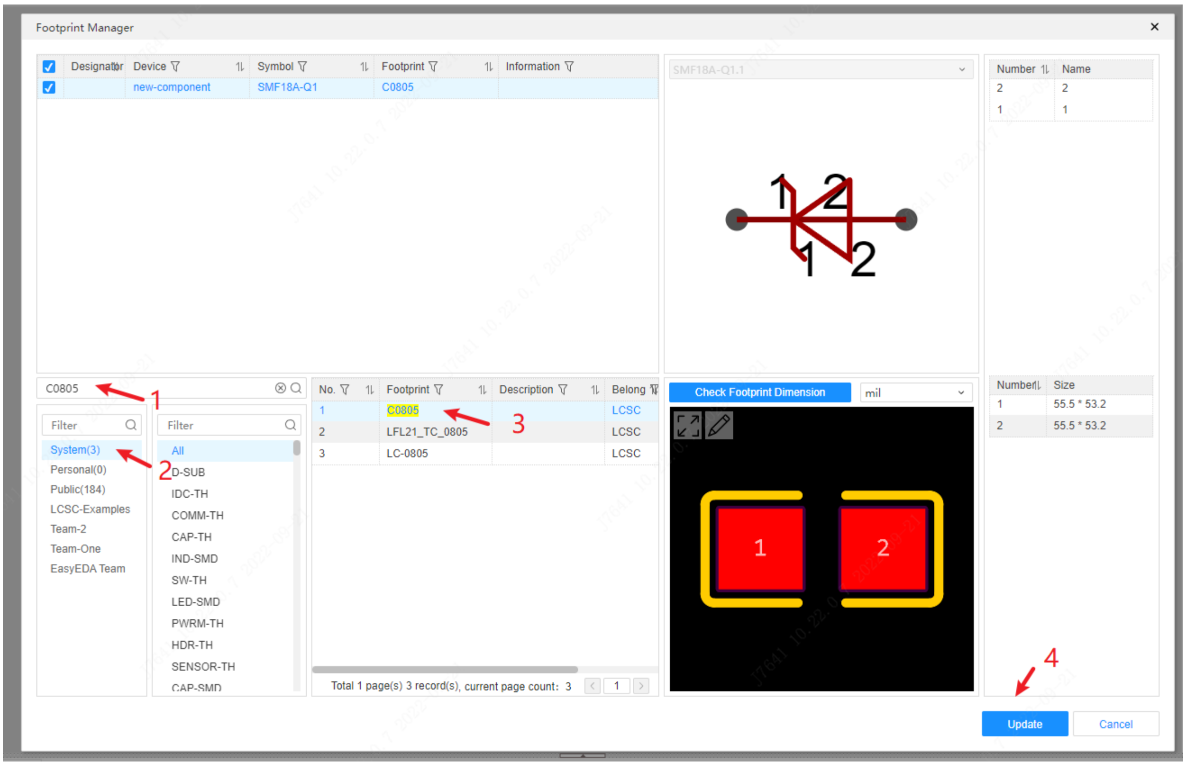


Add footprint

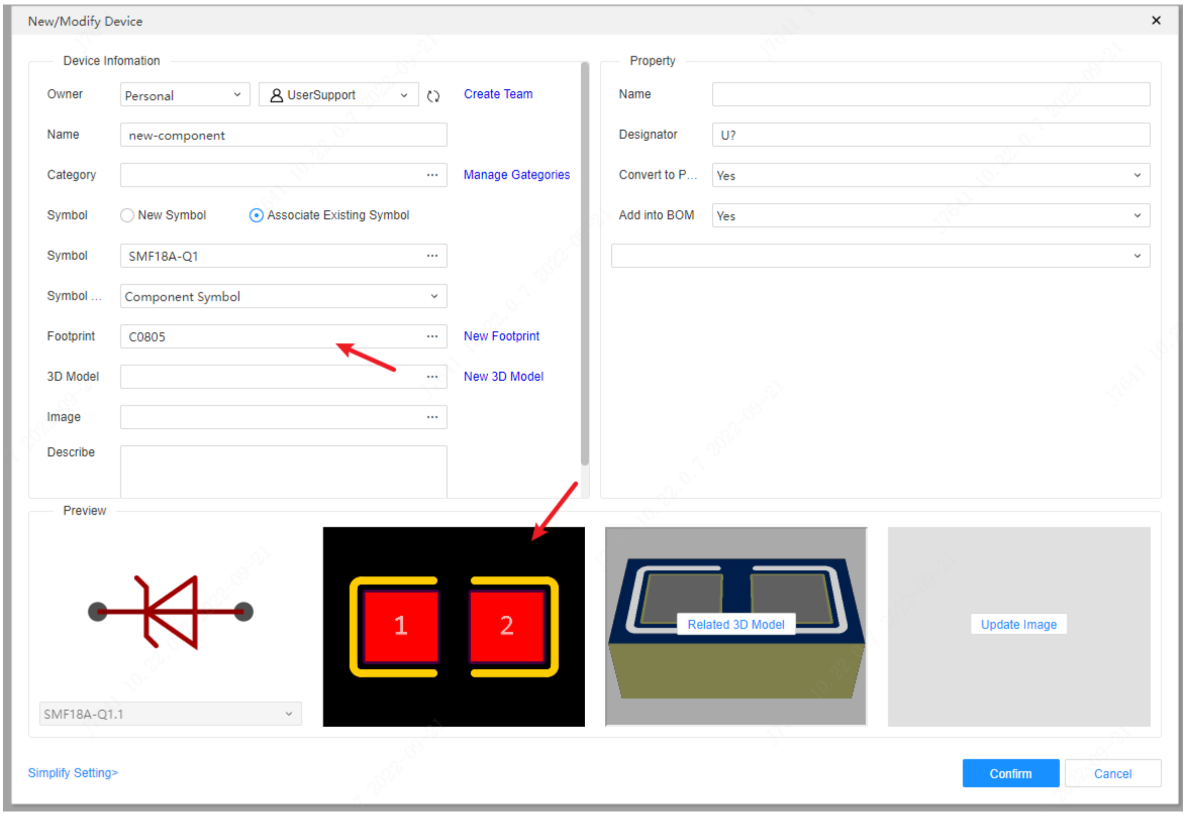
Click the footprint check box or the footprint preview area.



After clicking, a pop-up window will pop up, where you can find the footprint library created by the user and the footprint library that comes with the system, find the corresponding footprint, and click Update to add it to the device library.

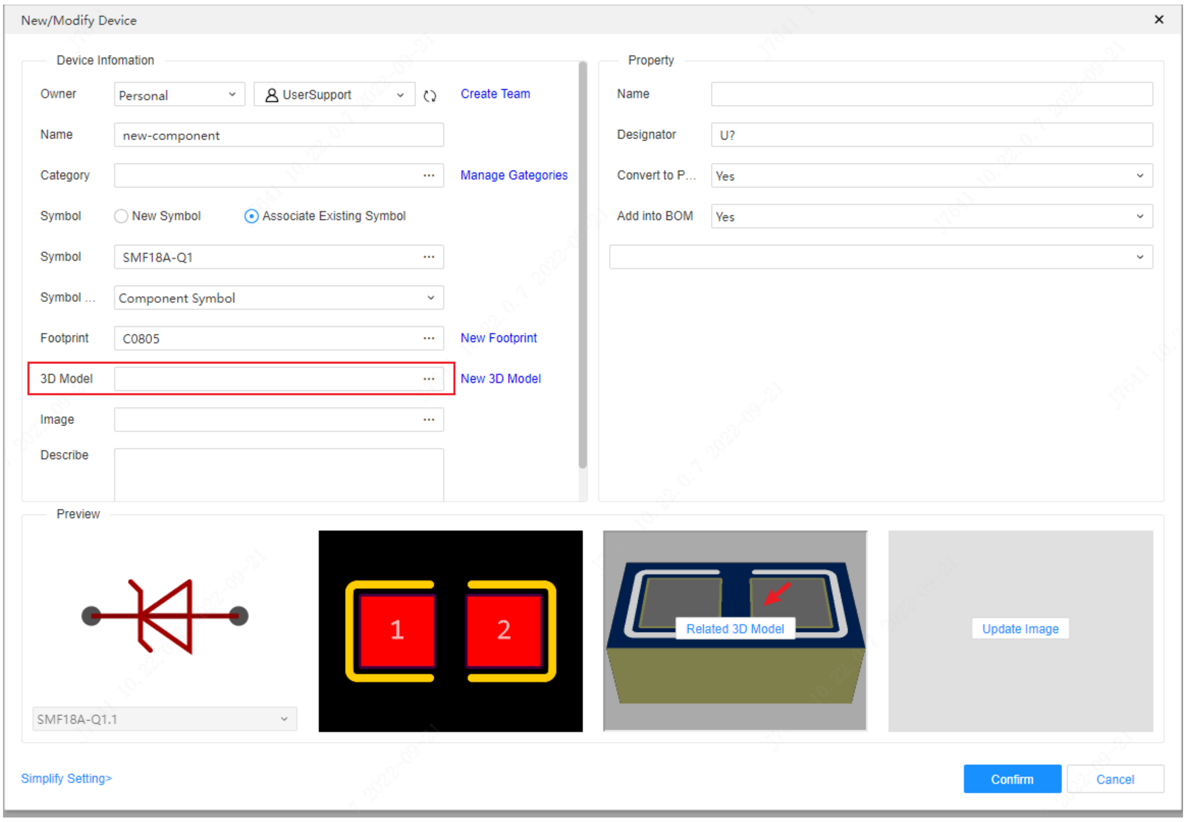


binding footprint:

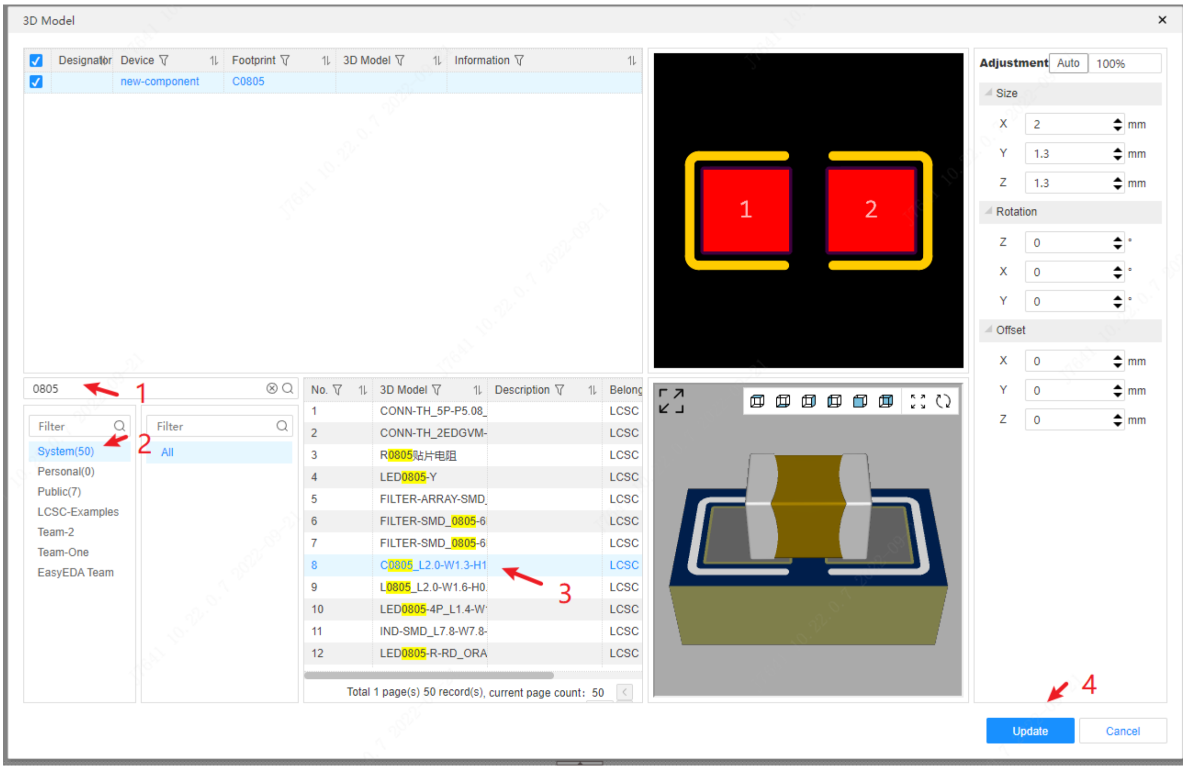


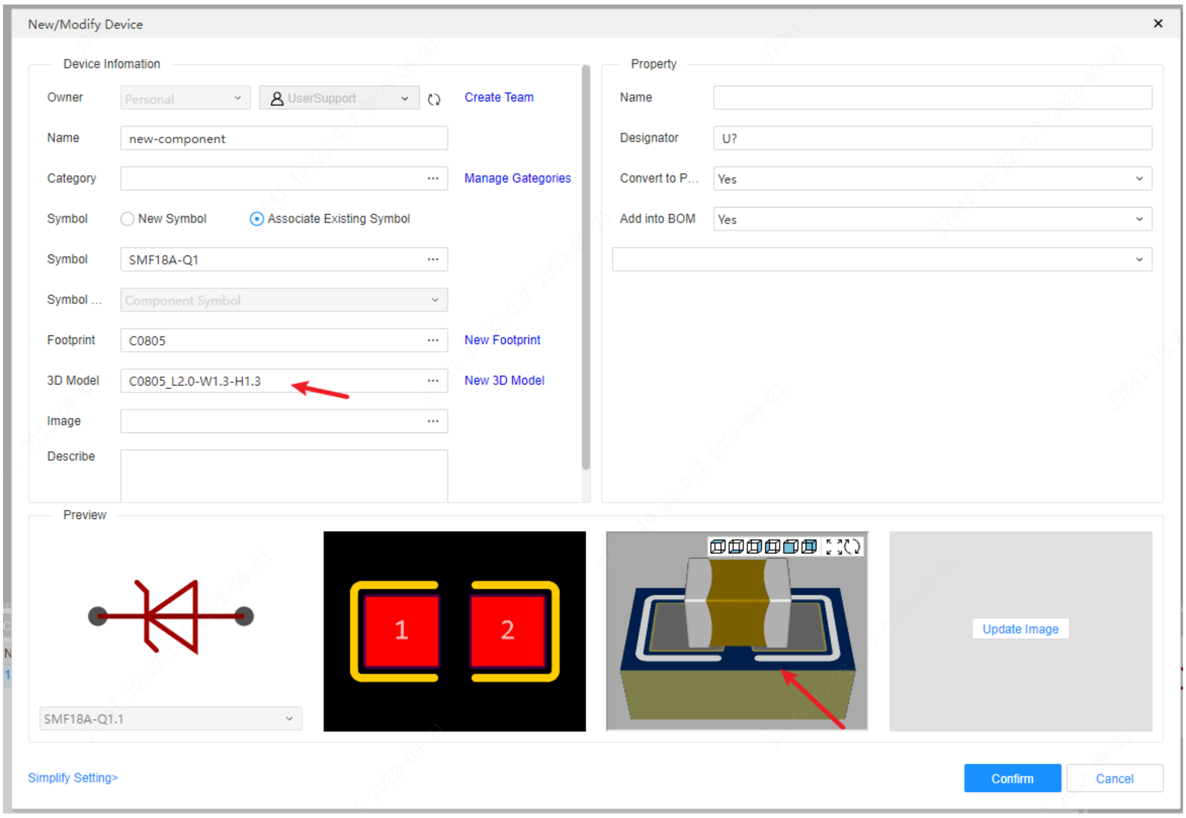
Add 3D Model

Click the 3D model check box or the 3D preview area,



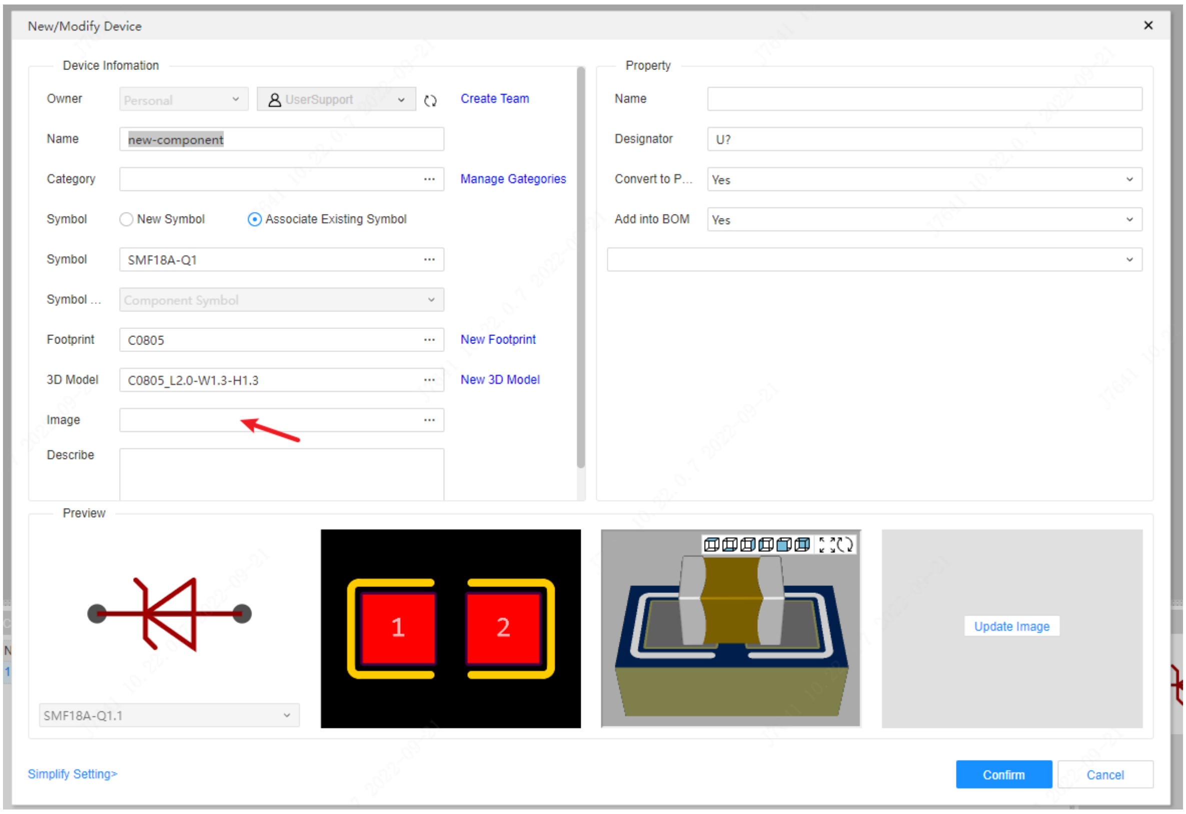
find the corresponding 3D model, adjust the position of the model and click Update to complete the addition of the 3D model.



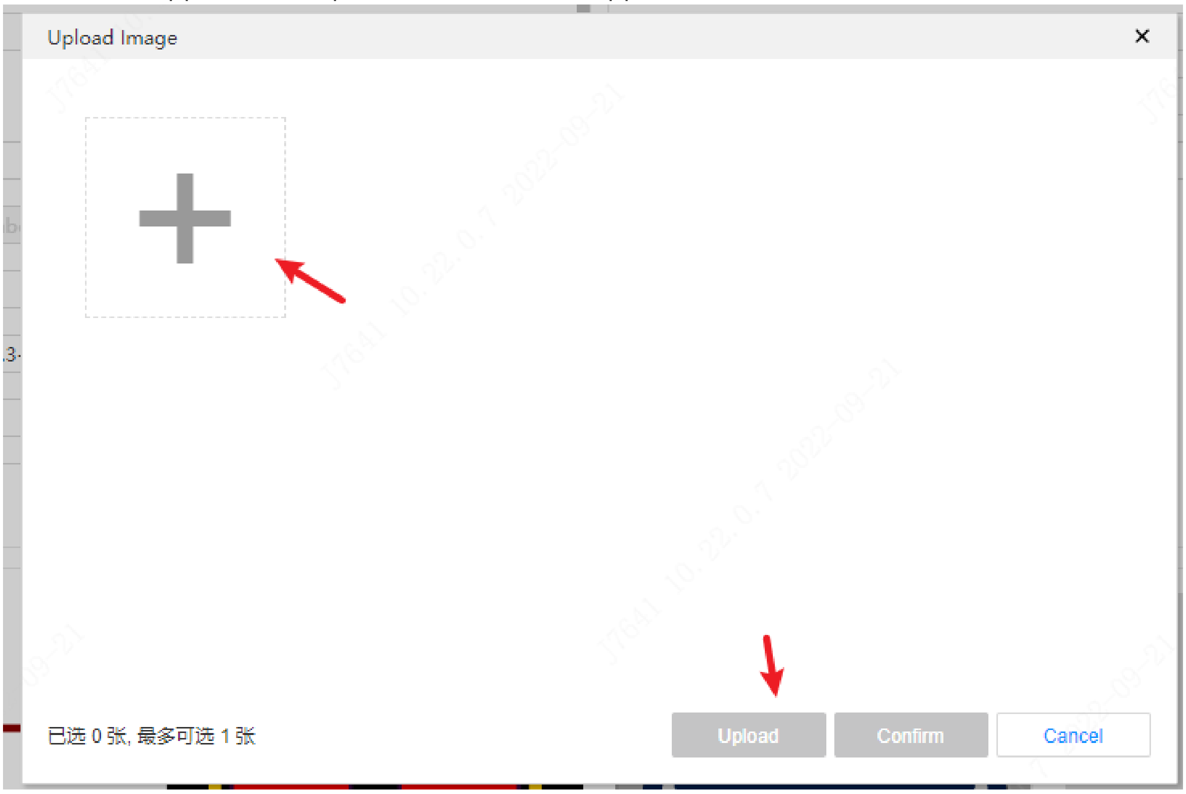


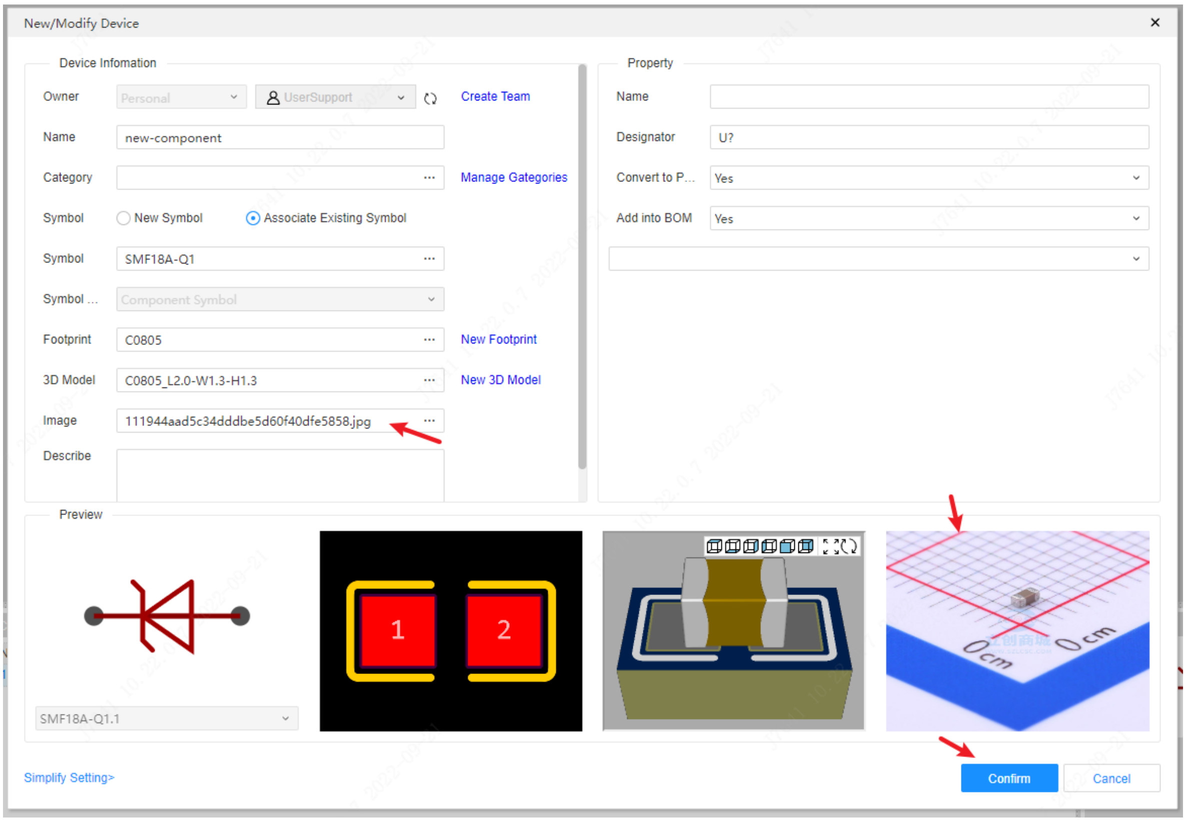
Add Pictures

Click the image's checkbox or the image preview area.



After the pop-up window pops up, click "+" to select the picture to be added. Only files in picture format are supported, and picture links are not supported.

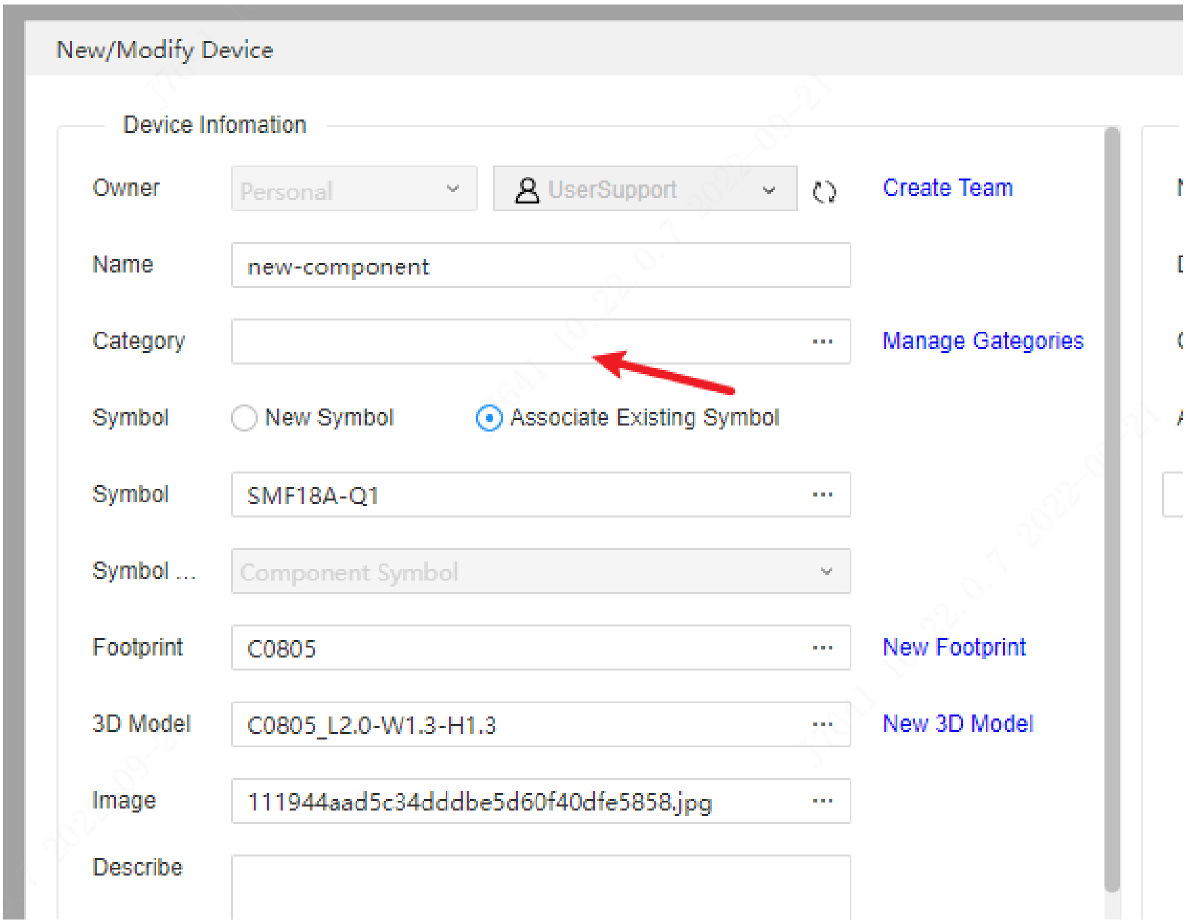




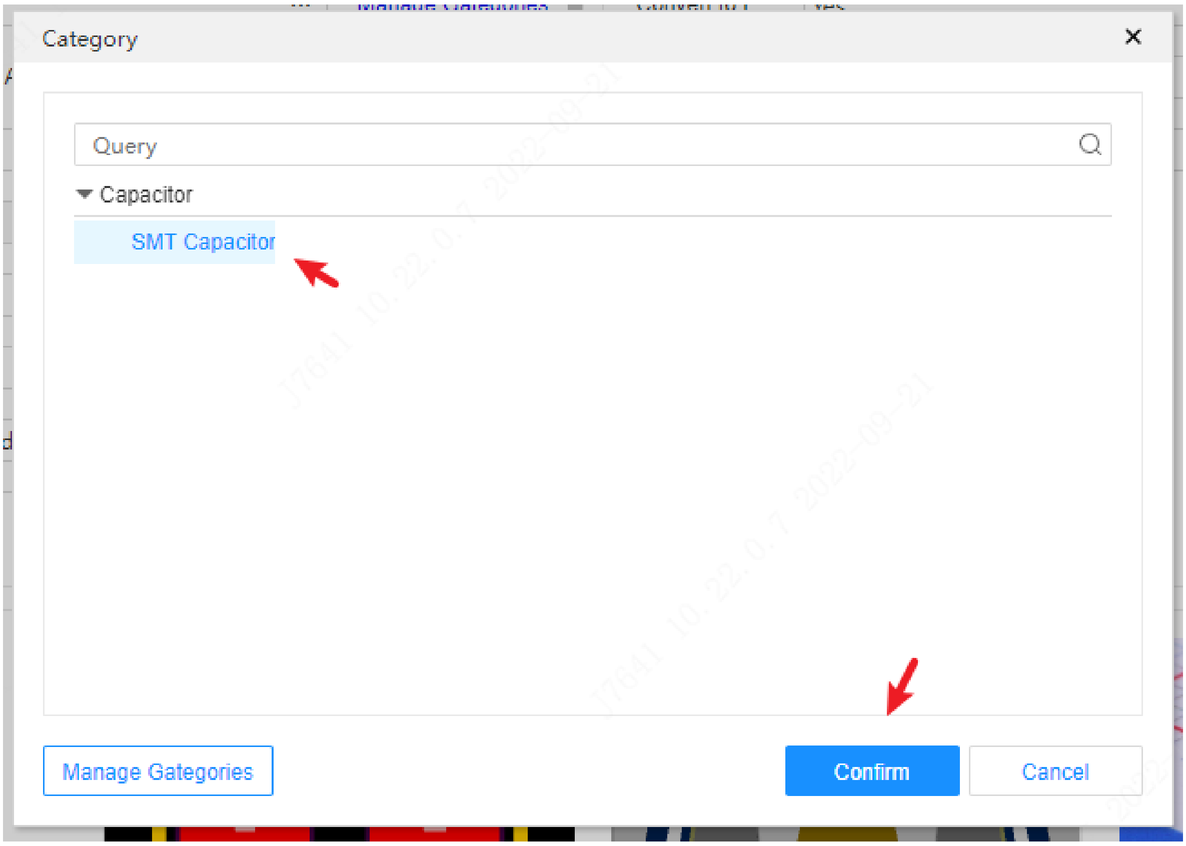
Add Category

The device is classified and set to facilitate management and maintenance.

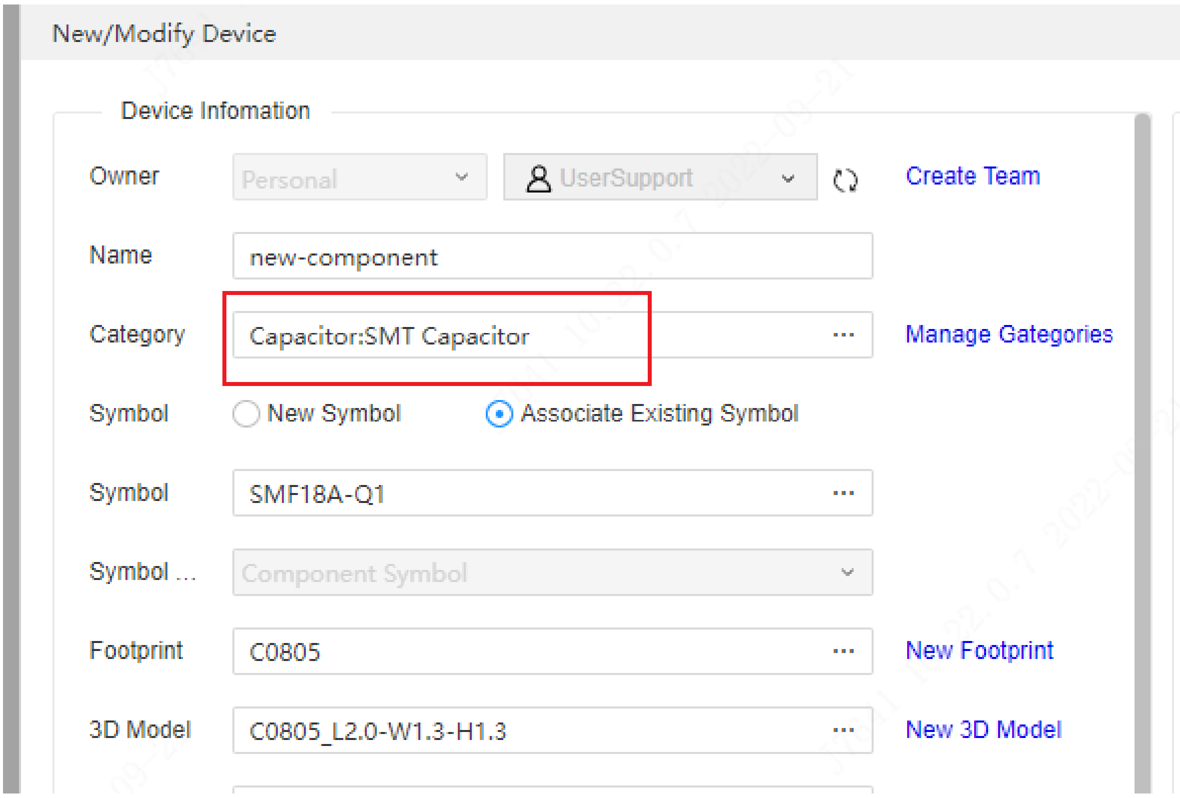
Click the checkbox for the category



Select the corresponding attribute and click OK to add it to the corresponding category.

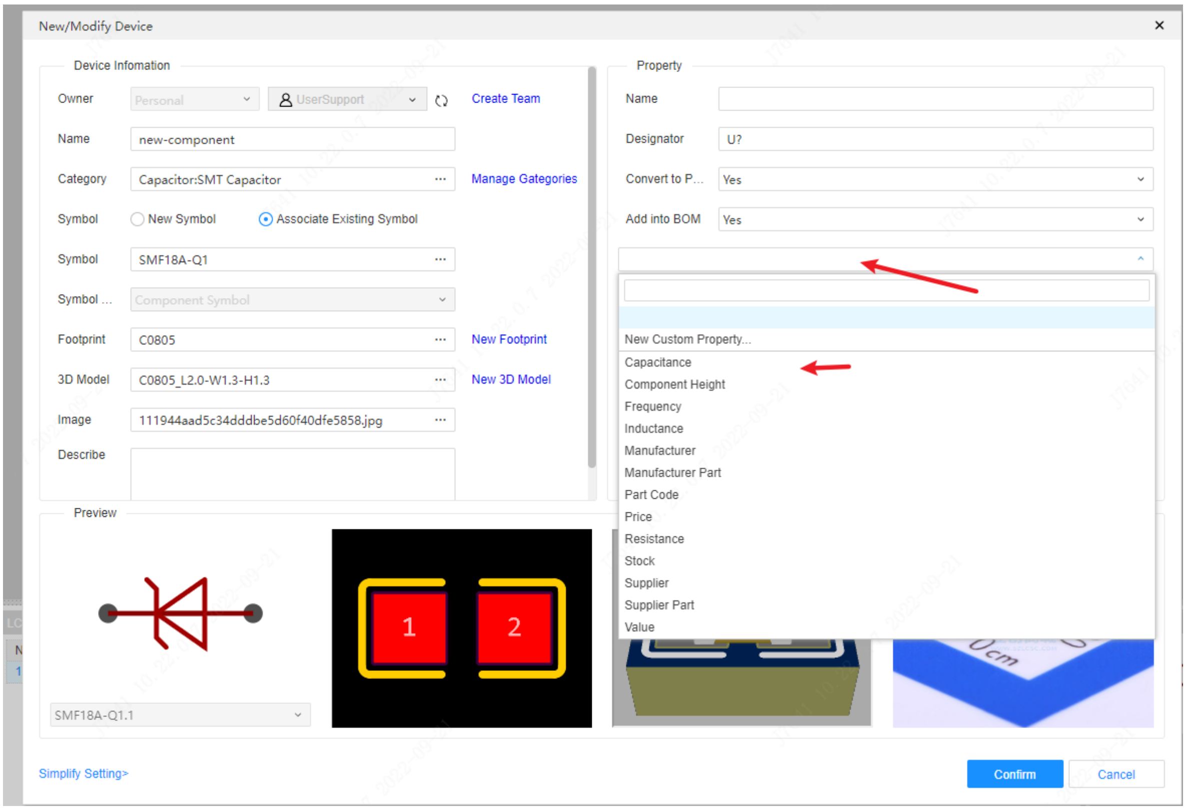


Added catagory:

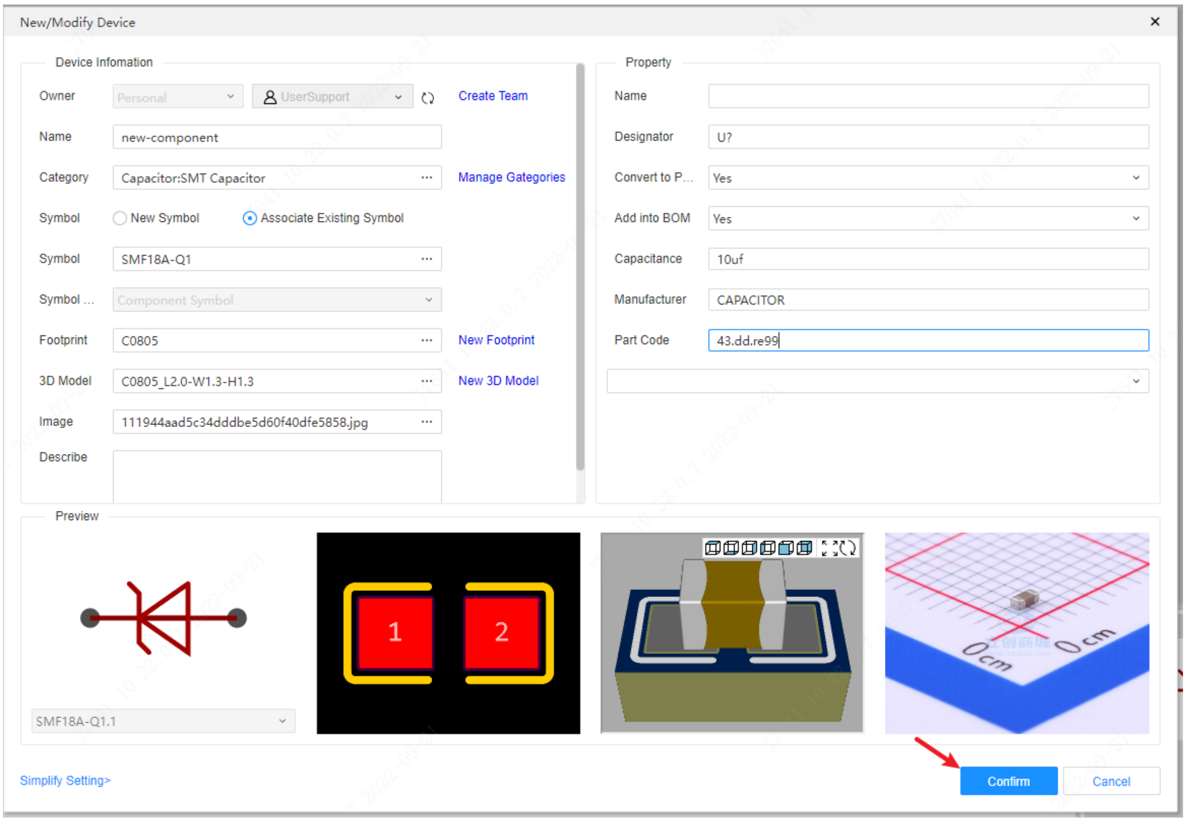


Add Properties

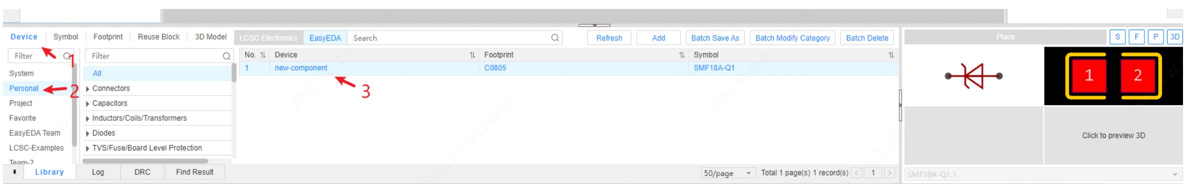
Add the properties of the device, select the properties to be added, and then add the device.



After adding symbols, footprints and corresponding properties, click OK to complete the creation of the device.



The created device can be viewed in the device library list in the bottom panel.



Edit Device

Perform secondary editing and modification on the created device.

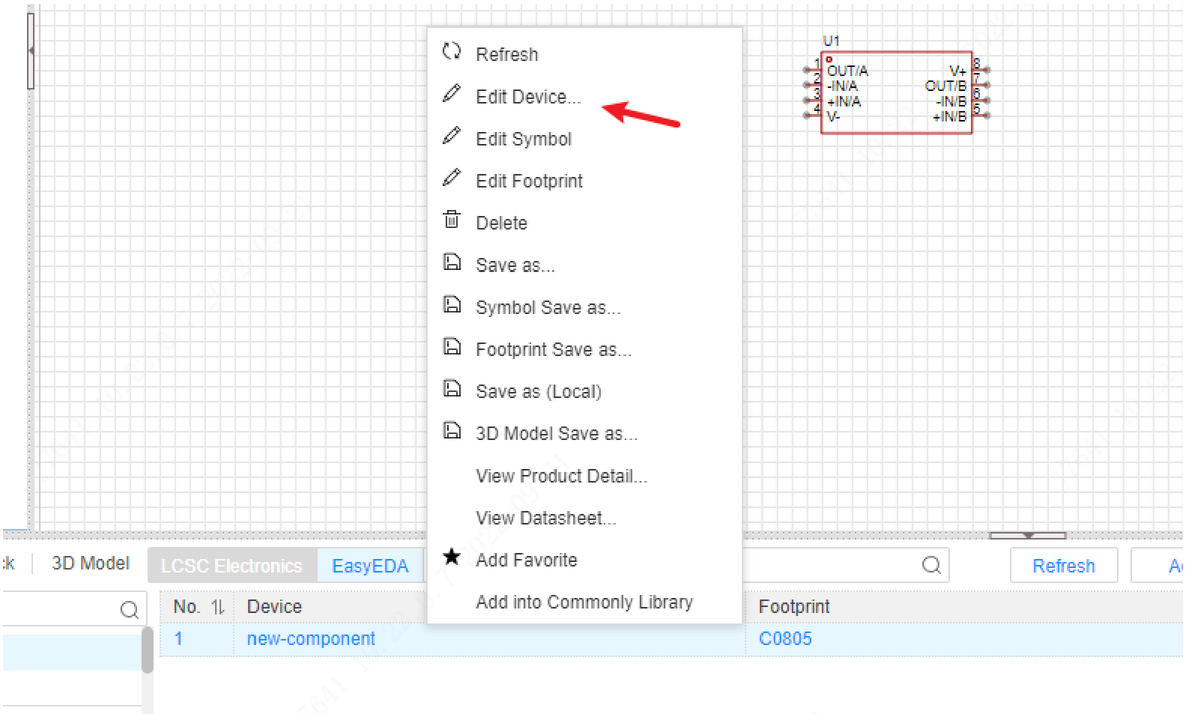
Edit Device Steps:

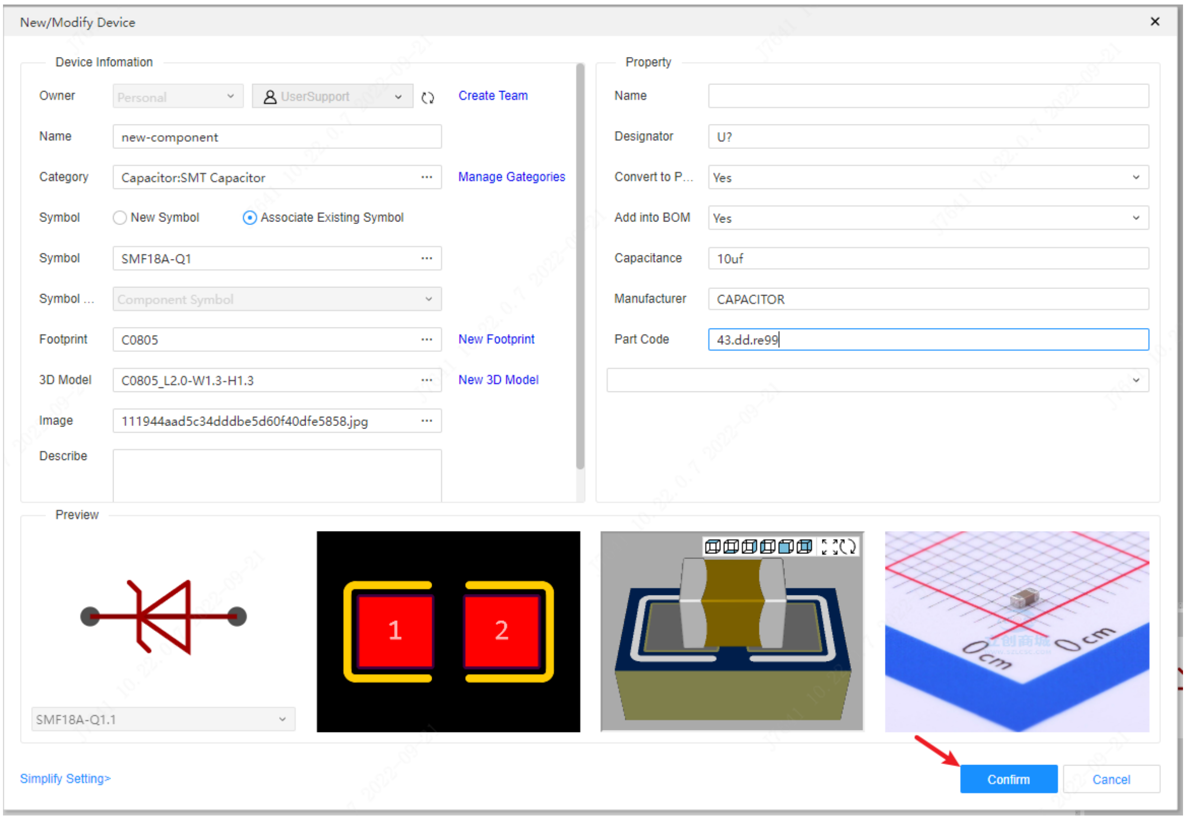
- Open the bottom panel, find the device that needs to be edited twice, `right mouse button` - Edit the device.

illustrate

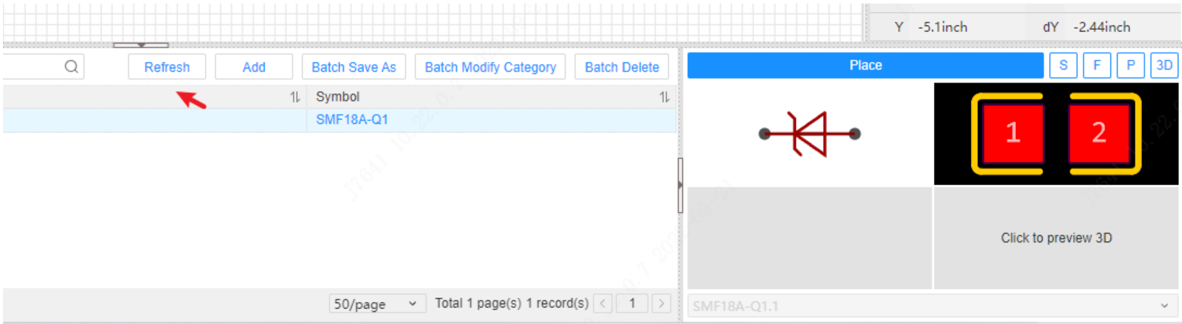
- The device in the system library cannot be edited. If you need to edit it, please save it as personal and then edit it.
- The device panel opened by `Shift` + `F` cannot edit the device, only place the device.

The editing pop-up window of the device will pop up. The pop-up window is the same as the new device pop-up window. After the modification is completed, click OK to update the revision information.





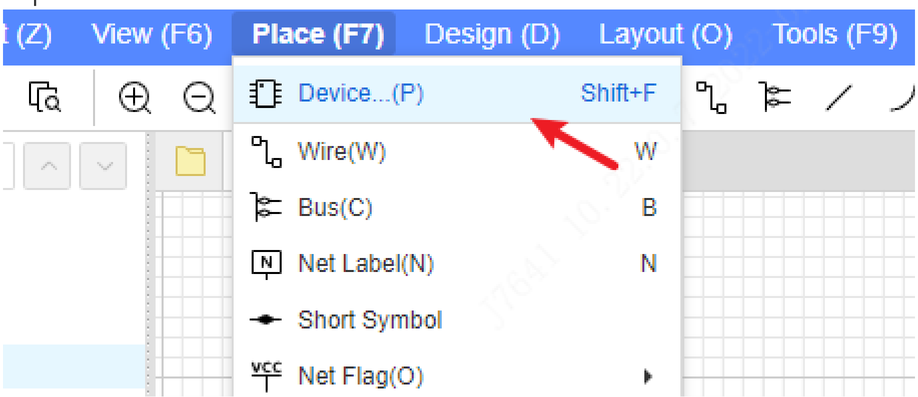
After modifying the device, you need to click the `refresh` button of the device library to see the latest modified information

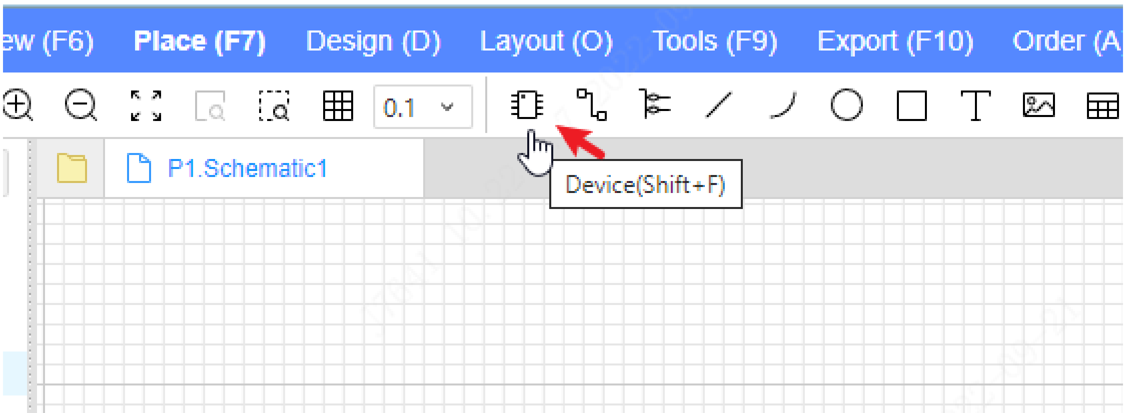


Place Device

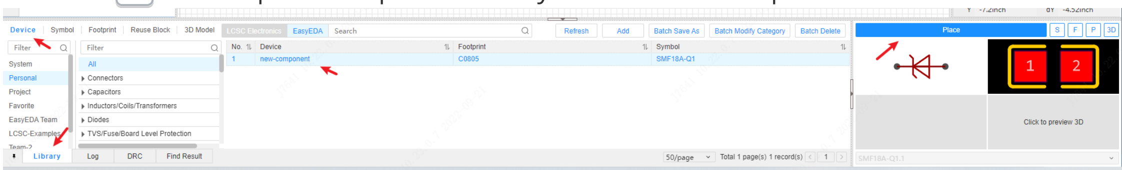
How to operate:

- Top Menu - Place - Device

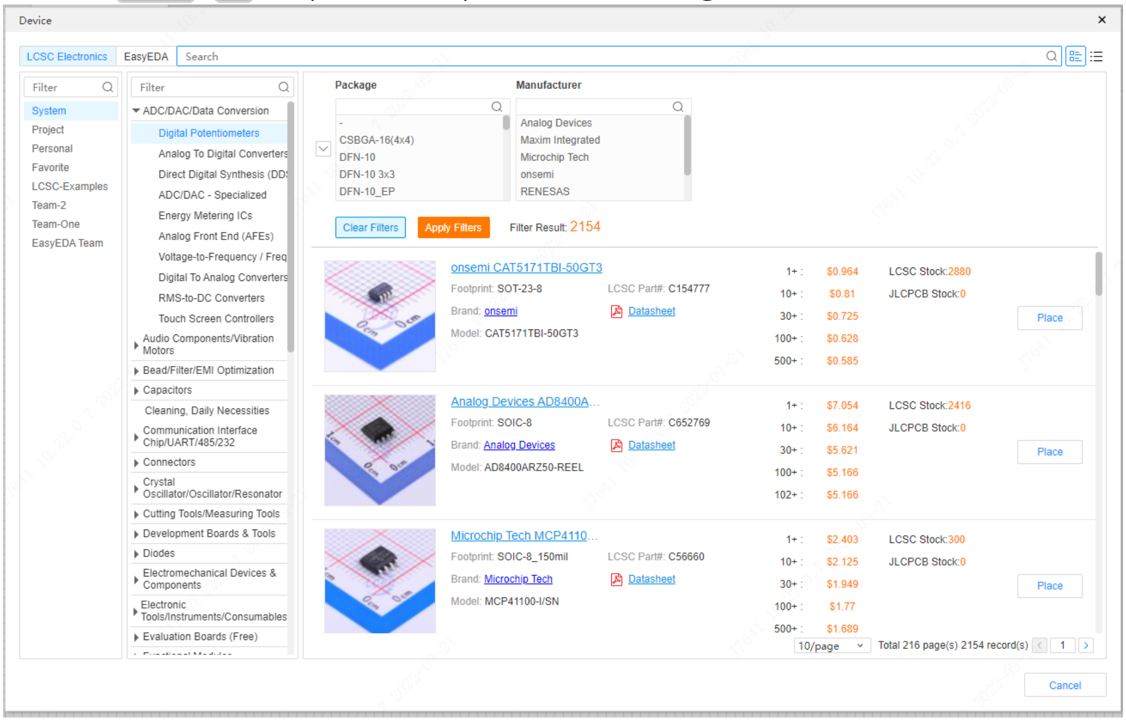




- Shortcut `S` to call up the component library list in the bottom panel



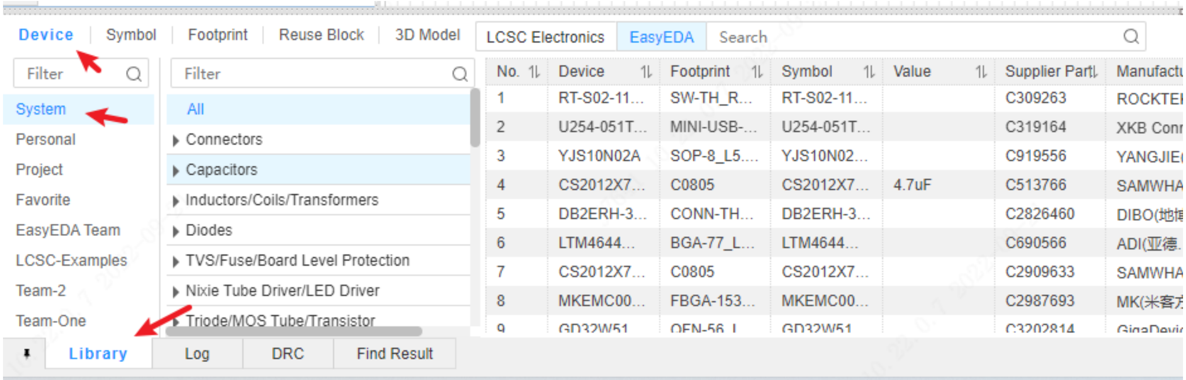
- Shortcut `Shifi`+`F` to open the independent Place dialog



Select the device, click to select the desired device, and click the Place button on the right side. The mouse cursor turns into a cross state to place the device. The device does not support drag and drop.

Device List

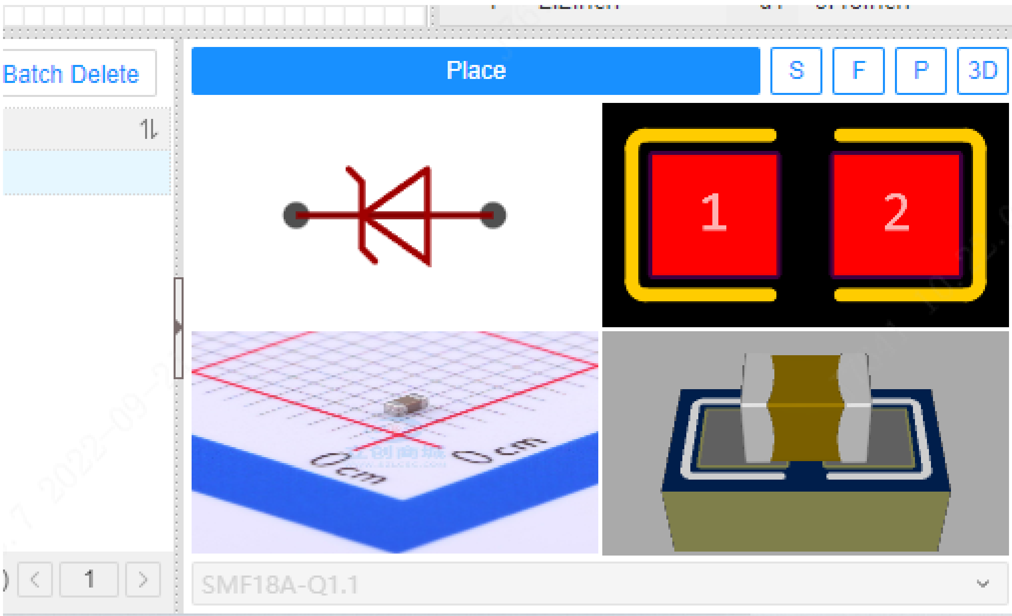
Device List In the bottom panel of the editor, the shortcut key `F` or `shift`+`F` opens the display device library. In the device list, system devices, personal created devices, team created devices and favorite devices will be displayed in the list.



Preview Area

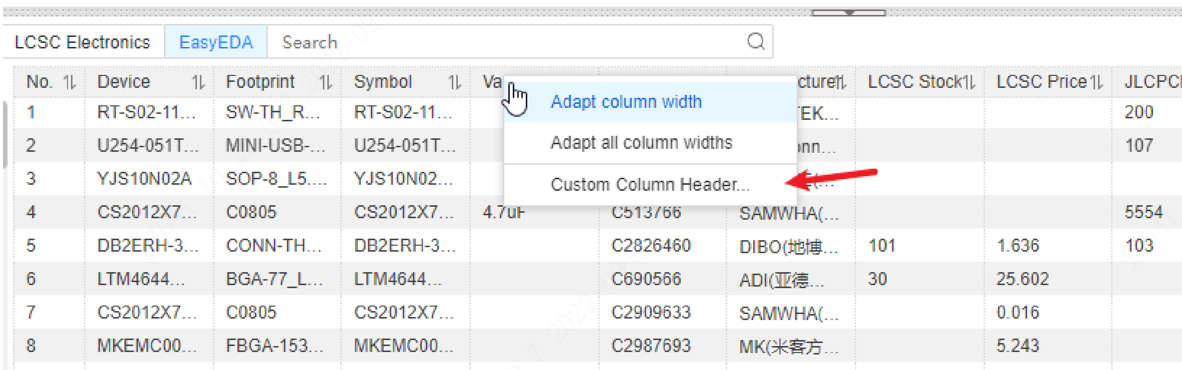
On the right side of the device list is the preview area of the device. Click the button above to display or hide the corresponding preview image.

- **S**: Symbol, symbol map
- **F**: Footprint, footprint
- **P**: Product, product image, uploaded image
- **3D**: 3D Model, 3D model drawing

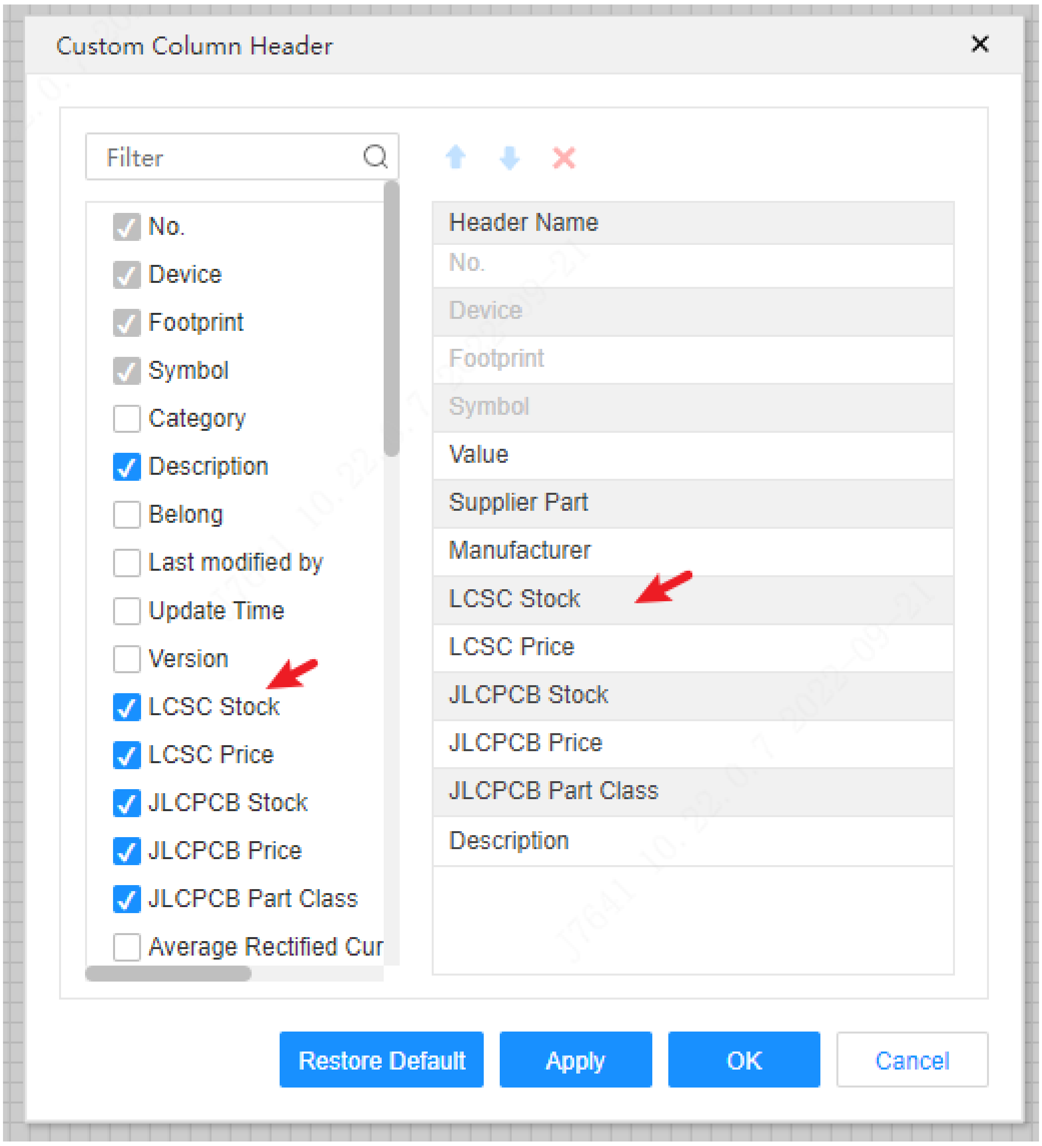


List Header

The device library list can be customized by the user. Put the mouse on the header `right-click mouse` select custom header,



On the left are properties that have not been added to the header, and on the right are properties that have been added to the list header. The set custom header parameters will be saved to personal preferences.



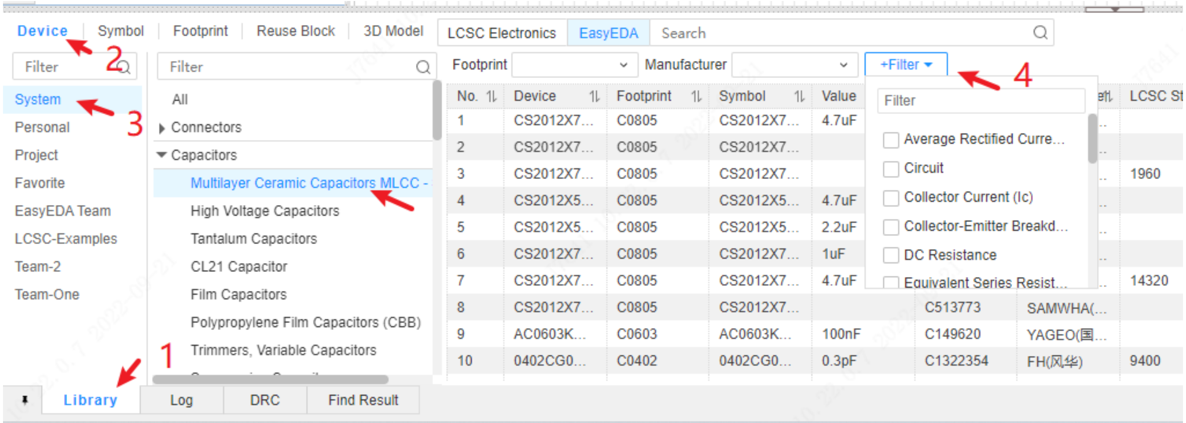
Notice:

- The corresponding column name will only appear when the current list has the corresponding parameter name. For example, the resistance value column, if the device in the current list does not have this resistance value attribute, the resistance value column will not be displayed in the list.

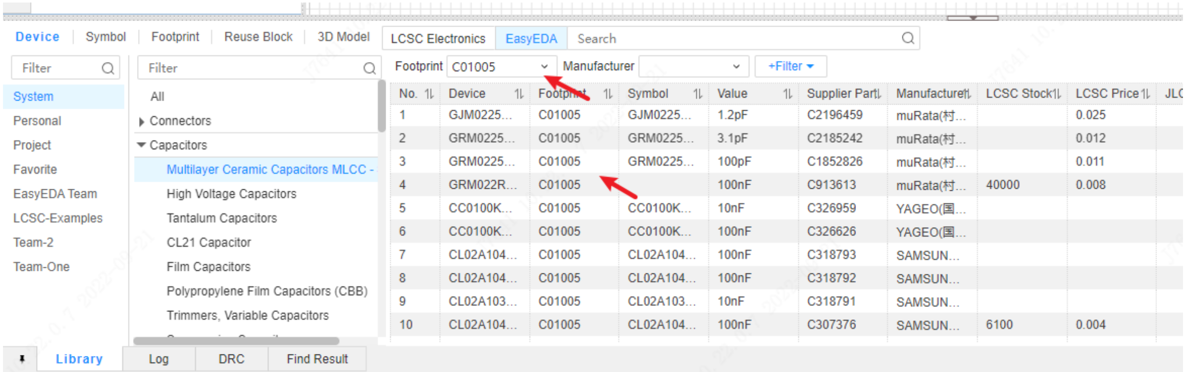
Parameter Filter Sorting

When in the device library of the system, click on the specific first-level or second-level category to make parameter options.

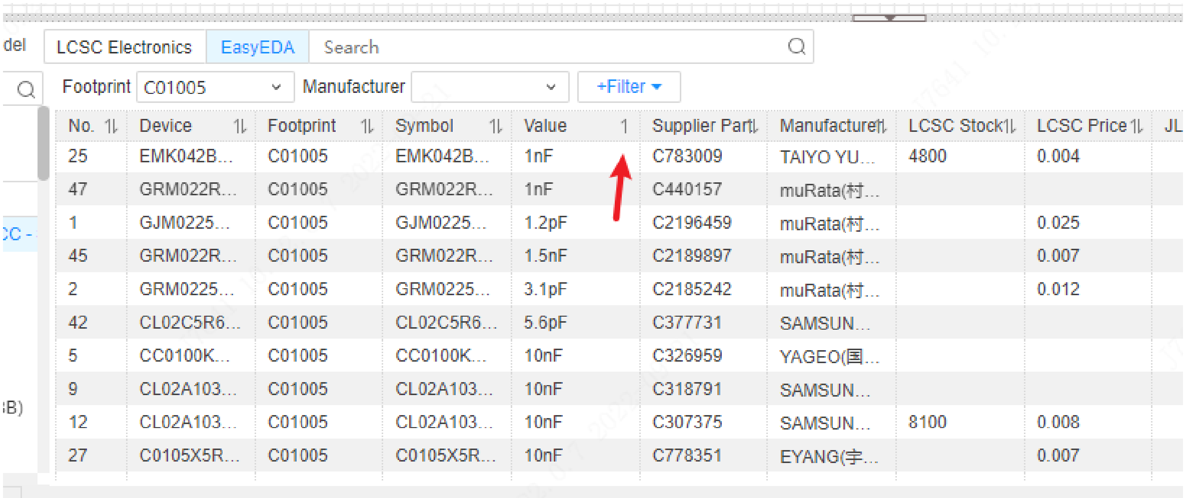
Notes: The filter sorting function can only be used by selecting devices in the primary or secondary category.



Click the "**Filter**" button at the top to select the parameter name to be filtered, and click the parameter value input box to filter the specific value, which is basically similar to the selection directly in the Create Mall.



Click the sort icon in the header to sort: default, ascending, descending, it only sorting current page.

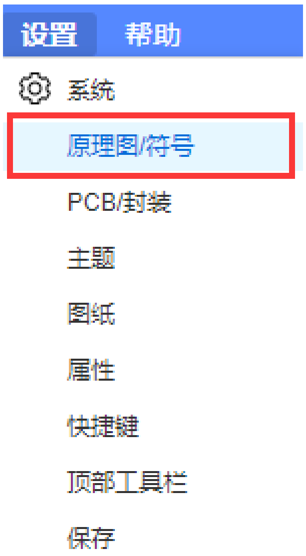


Symbol Library

Symbol Settings

Steps:

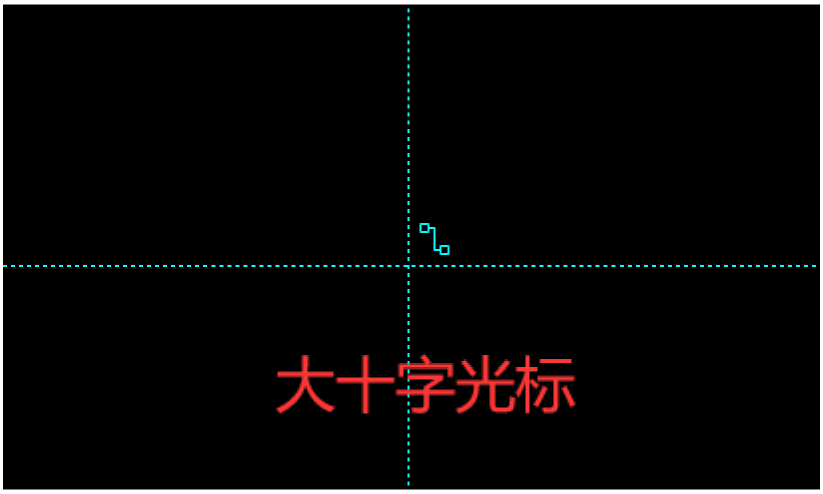
- Top Menu - Settings - Schematic/Symbol

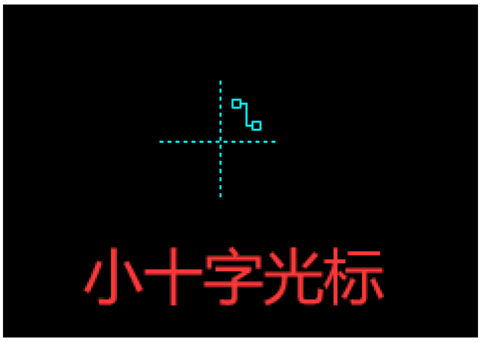


grid type

Set the default canvas! image-20201219153627002 There are only three grid types, namely dot, grid, and none.

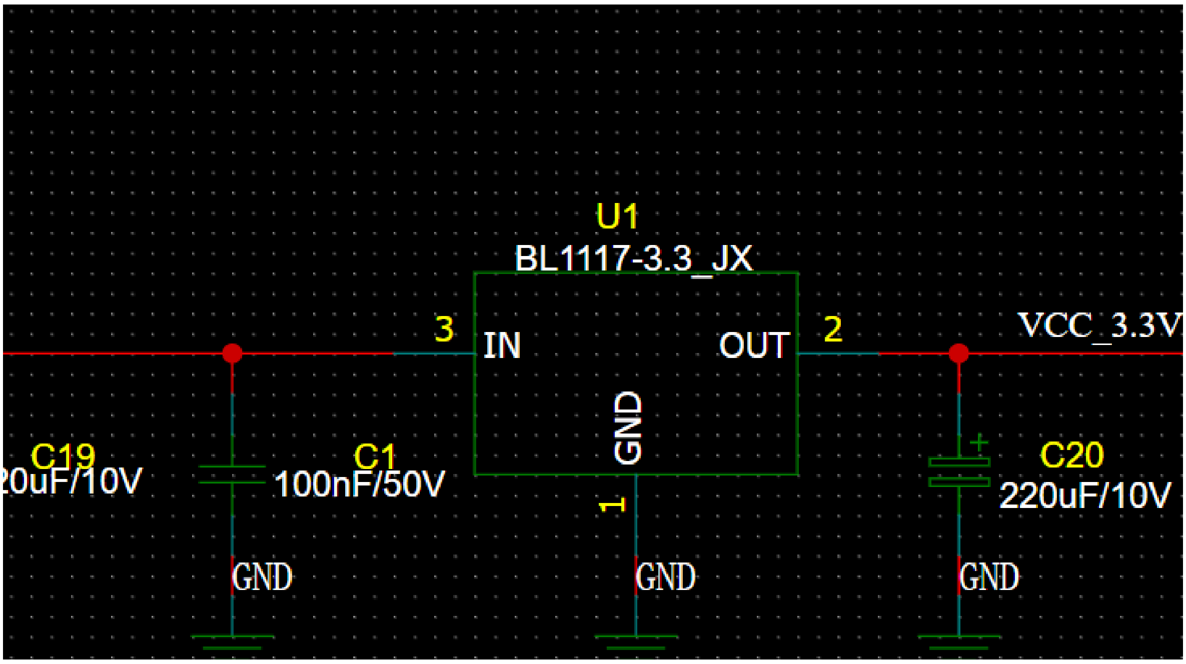
crosshair





line width display

Sets the size of the line width of the overall device line width.



The line effect that follows zoom changes will become thicker.

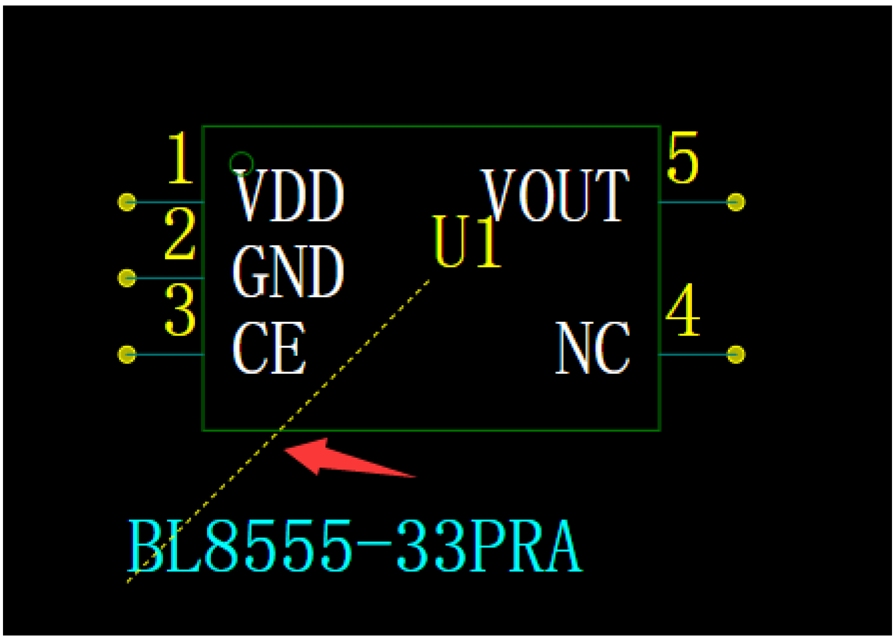
Default grid size

Modify the schematic page and default grid size to match the drawing page.

indicator line

Modifying the property indication of the schematic device can be modified into the following three.

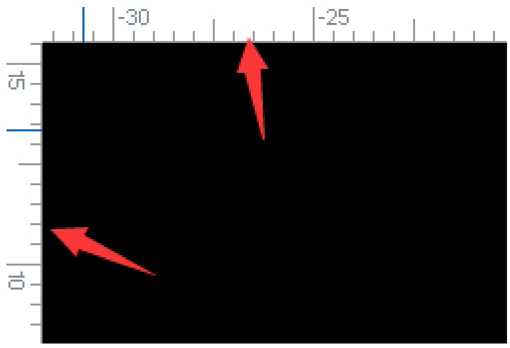
- **Show when selecting a single device and selecting an attribute**: Display the indicator line when a single device or attribute is selected.
- **Show when selected**: Display the indicator line when the attribute is single-selected.
- **DO NOT DISPLAY**: Do not display the indicator line.



Move symbol, wire follow mode

- Follow by default, press and hold Alt to disconnect before the movement starts: When the symbol moves, the wire will move with the symbol. Press and hold the Alt key to move the symbol, and the wire will not move with the symbol.
- No follow by default, press and hold Alt to keep connected before moving: When moving the symbol, the wire will not move with the device, you need to hold down the Alt key to move the wire with the device.

show symbol ruler

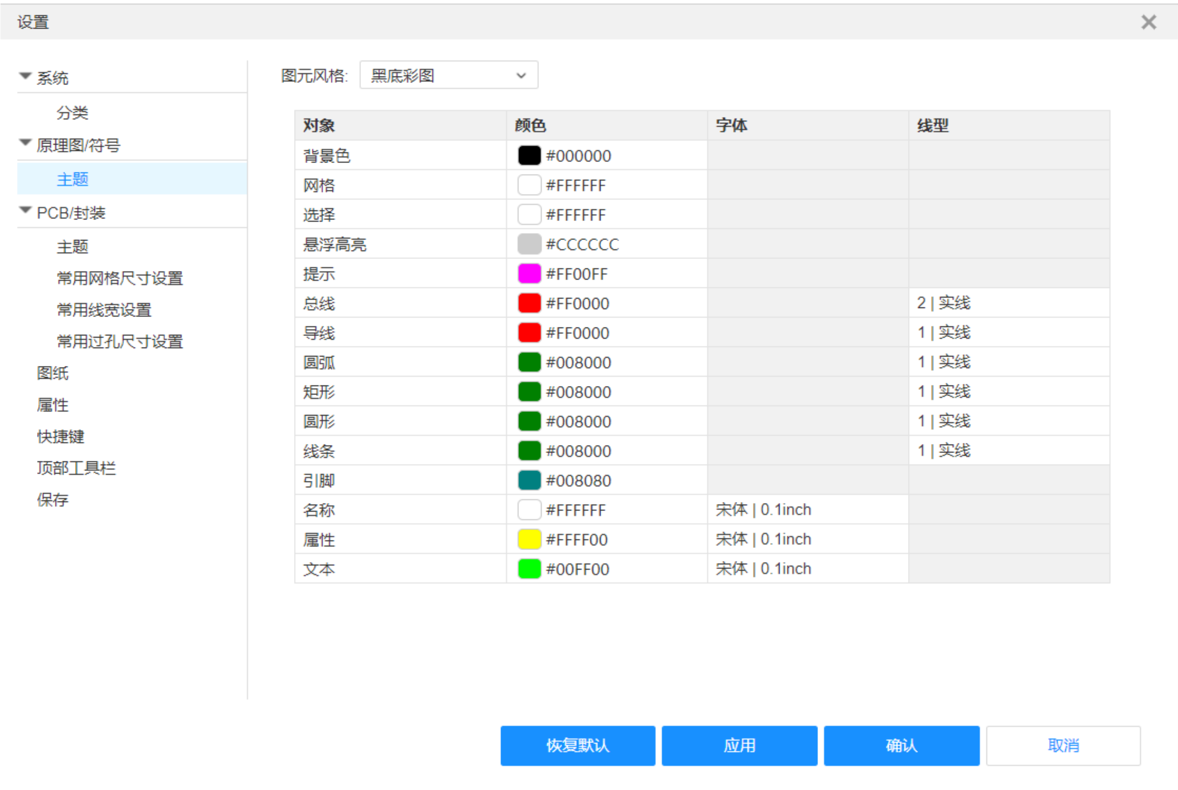


Automatically assign tags when placing or pasting devices

Tag numbers are automatically assigned when components are pasted or placed.

Schematic theme settings

Here you can modify the background color of the schematic page or the color of the text, the default font, and the default line type.

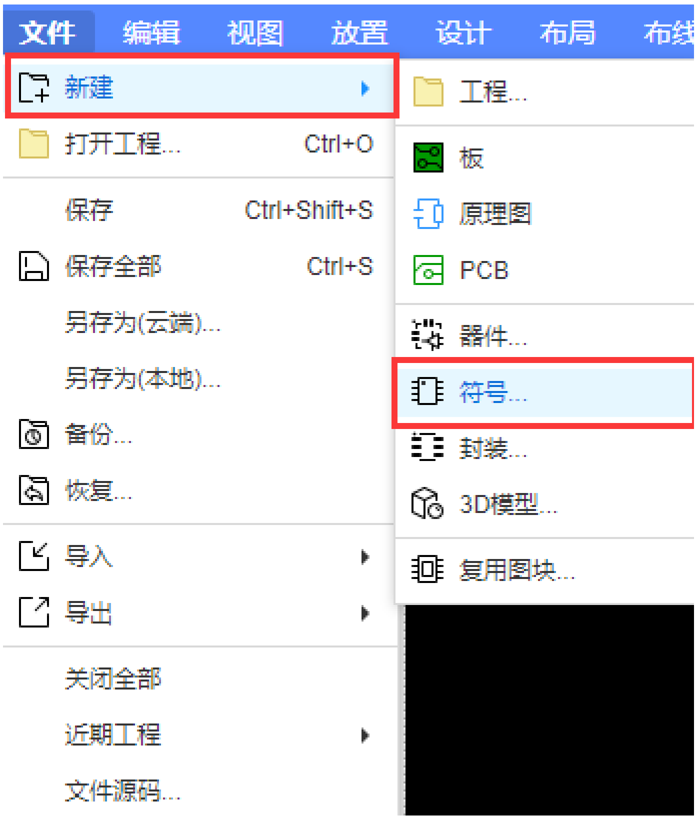


Create Symbol

create symbol library

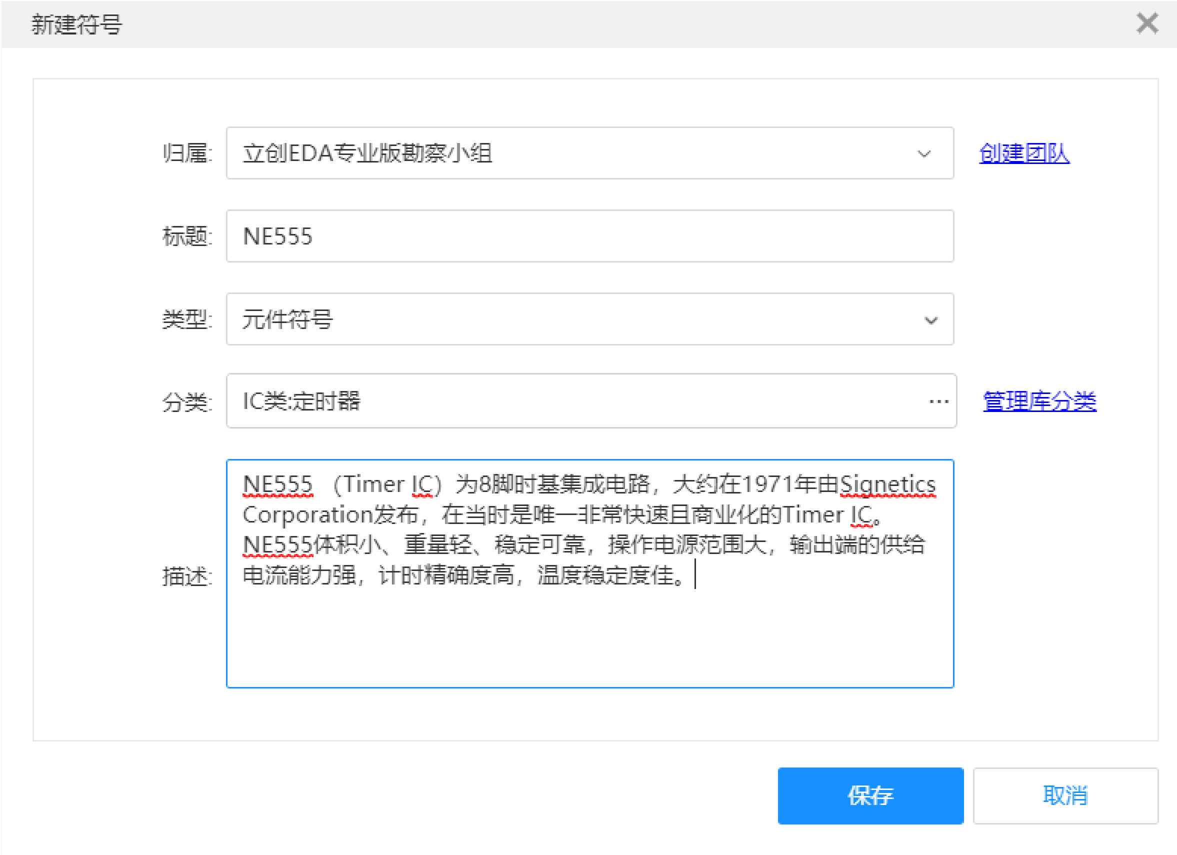
Create steps:

- Top Menu - File - New - Symbol



- **Attribution**: Select the user or creator of the symbol.
- **Title**: The name of the created symbol.
- **Type**: Other types of symbols can be created from, for example: Component Symbols, Net Identifier Symbols, Net Port Symbols, Drawing Symbols, No Electrical Symbols. Please refer to the next chapter for the creation of specific symbol types.
- **Classification**: Selecting a good classification makes it easier to distinguish devices and better manage
- **description**: some detailed description of the symbol.

Click OK to create the symbol



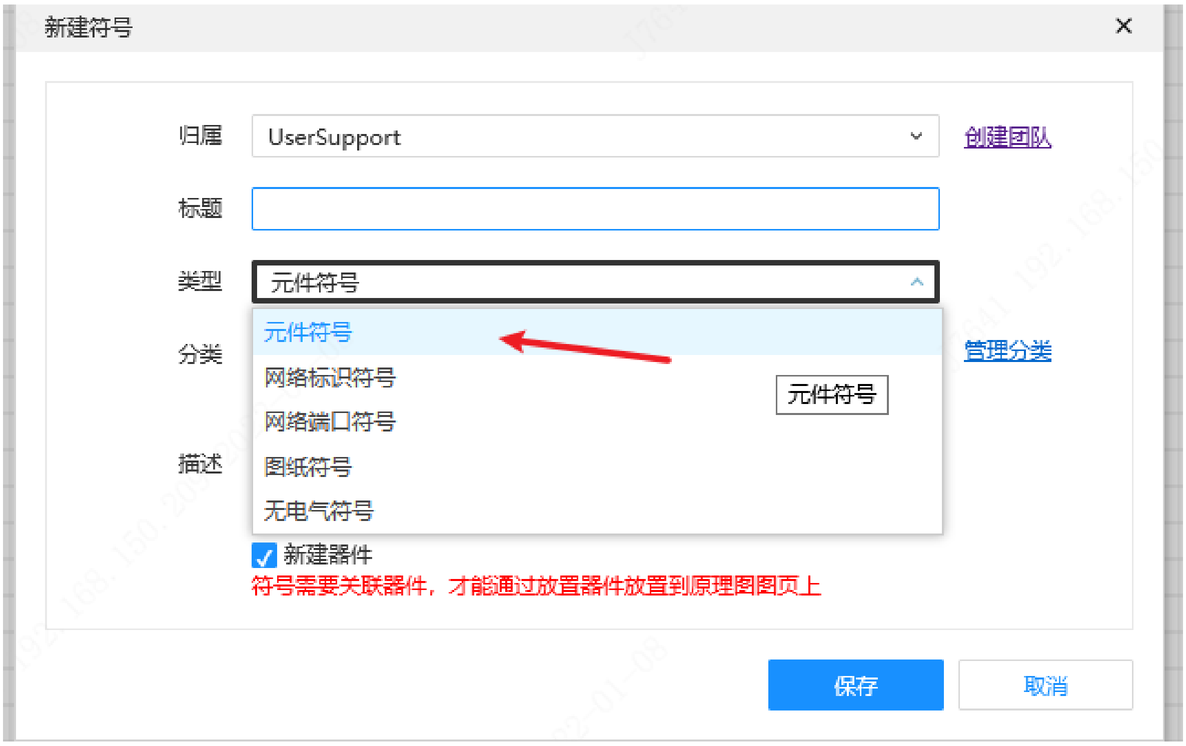
All created symbols must be device associated before they can be placed in the schematic. Therefore, after creating the symbol, you need to create a device for association.

component symbol

create component symbols

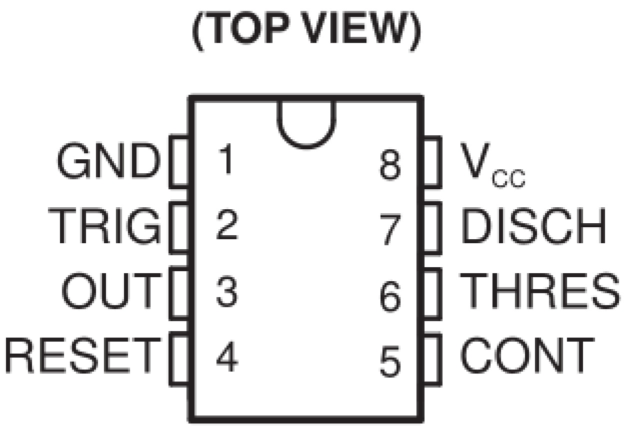
Component symbols refer to the components placed in the schematic diagram, which need to connect pins, set pin names, etc.

Entry: Top menu - New - Symbol, type select Symbol.

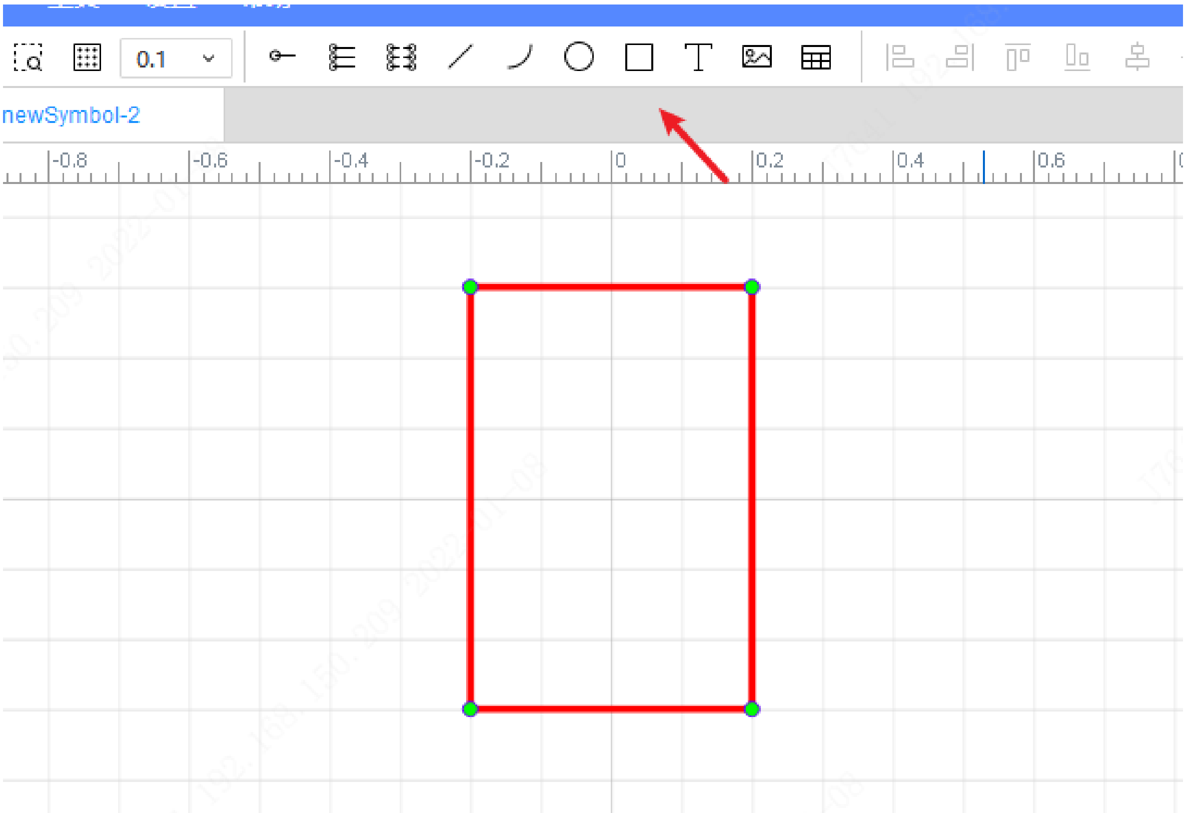


Steps to create a key symbol:

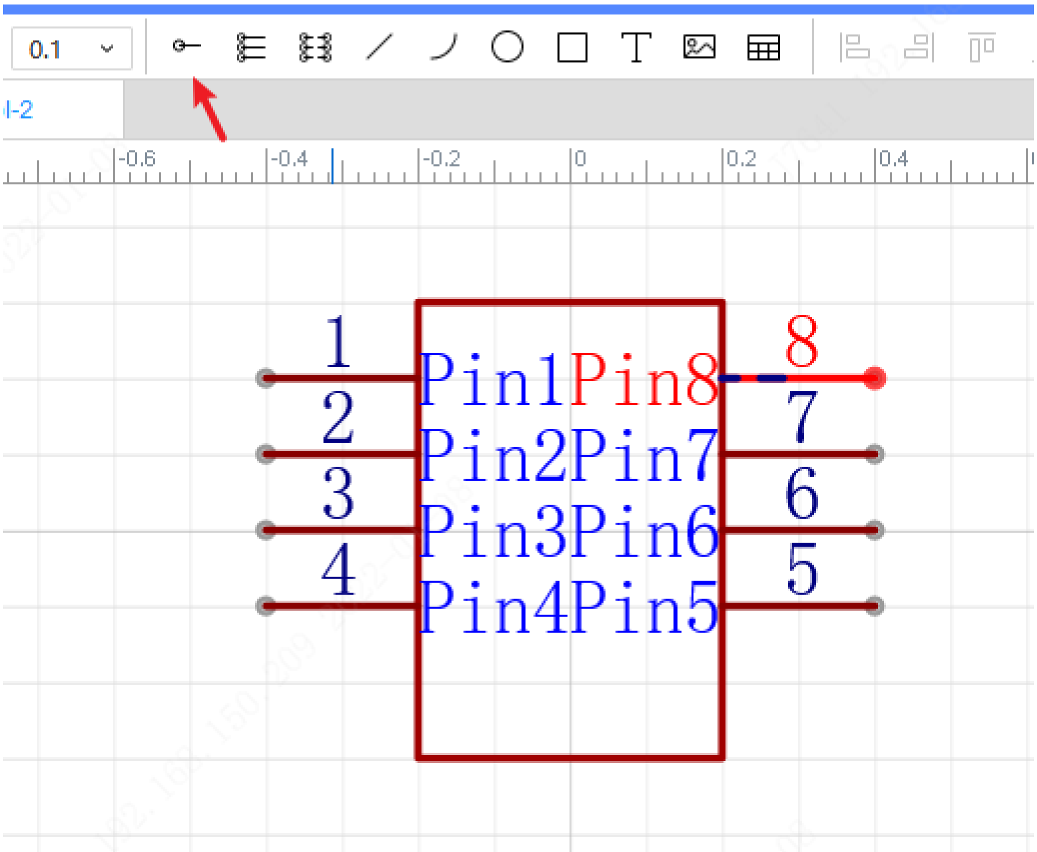
1. Obtain the component specification and get the symbol style and pin information. For example: https://item.szlcsc.com/8070.html



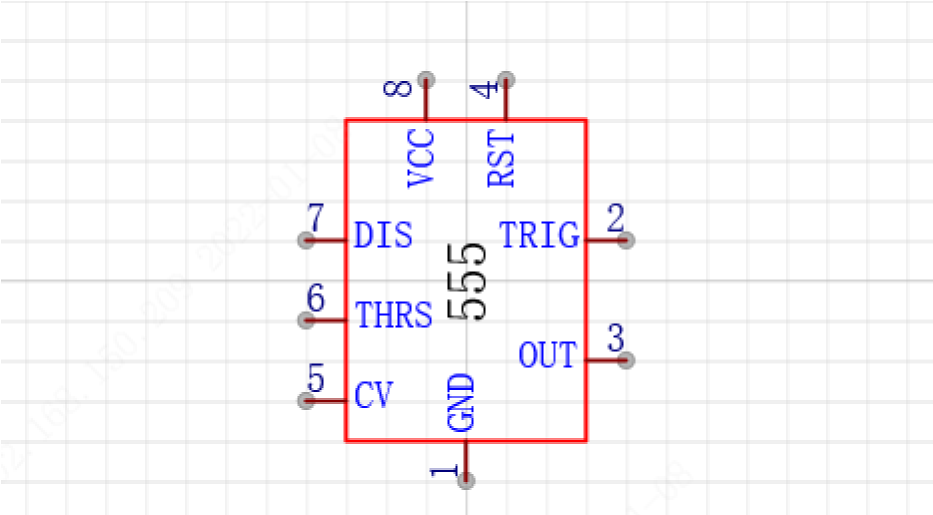
2. Draw a rectangle as a component graphic.



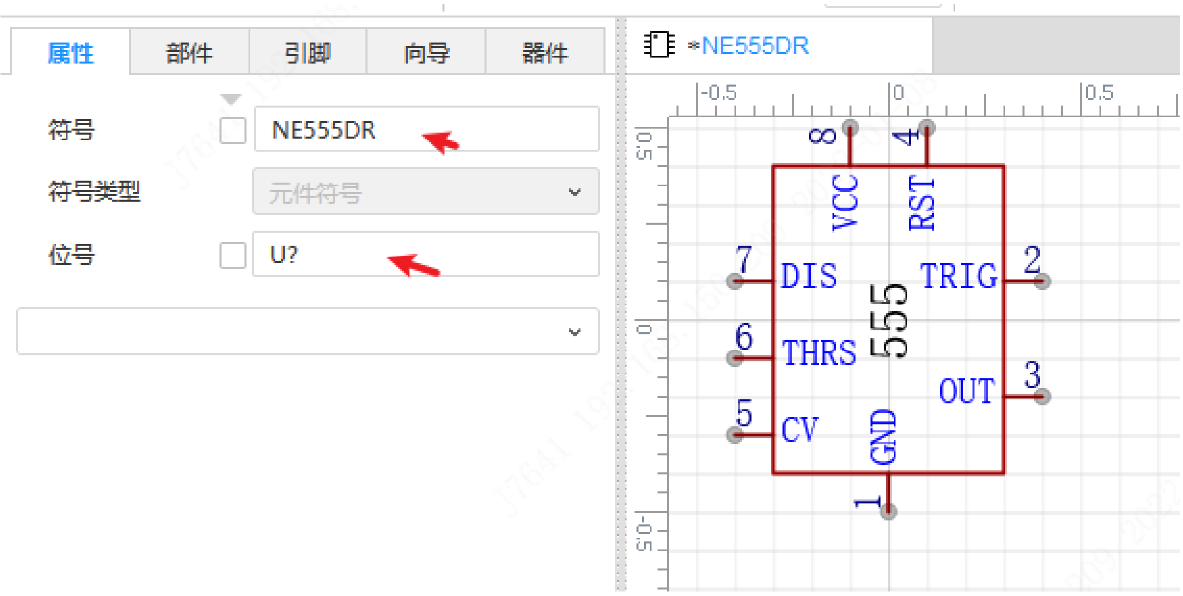
3. Place the pins. Shortcut key P. Entry: Top Menu - Place - Pins.



4. Set the pin number and name. Double-click the pin number or name to modify, you can also click on the pin list on the left, and modify it in the property panel on the right.
   Adjust the orientation of the pins according to the actual situation to facilitate the drawing connection.



5. Set the name and bit number value of the symbol in the property panel on the left, such as U?



6. Save and complete the creation.
   After saving, it can be found in the component library in the bottom panel: Bottom Panel - Component Library - Symbol - Personal.

If you need to create a sub-library, or a multi-part symbol library, please refer to the following.

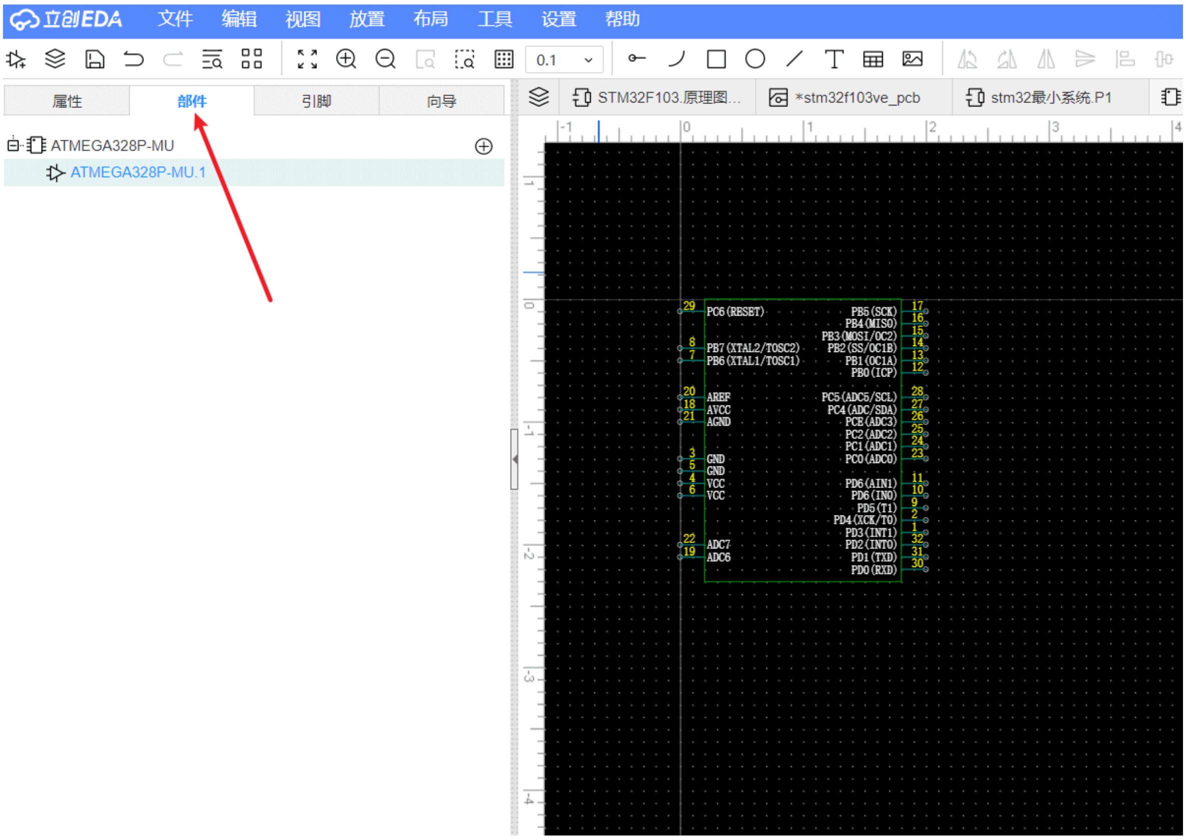
Note: You must create a good symbol created by binding a device with the same name or a different name before it can be placed and used in the schematic.

create sublibraries/multiparts

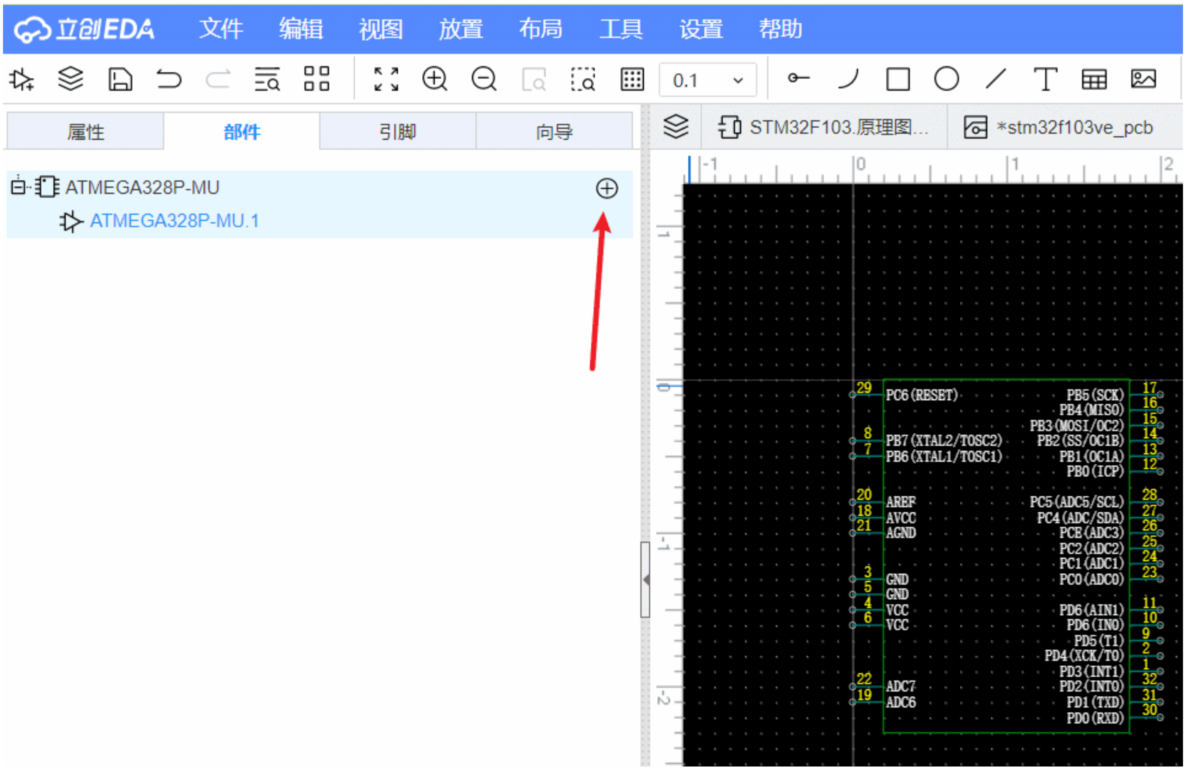
When a component has too many pins, it will be very space-consuming to draw it all in one library file, so it can be done by creating sub-libraries (also known as multi-part devices, sub-libraries, multi-part symbols, etc.) Multiple symbols combine the element. All sub-libraries together form a complete component. Each sub-library corresponds to the same footprint, only the corresponding pad numbers are different.

Steps:

1. Create a symbol library file and select the component in the project panel on the left.



2. Click + on the right to add a sub-library



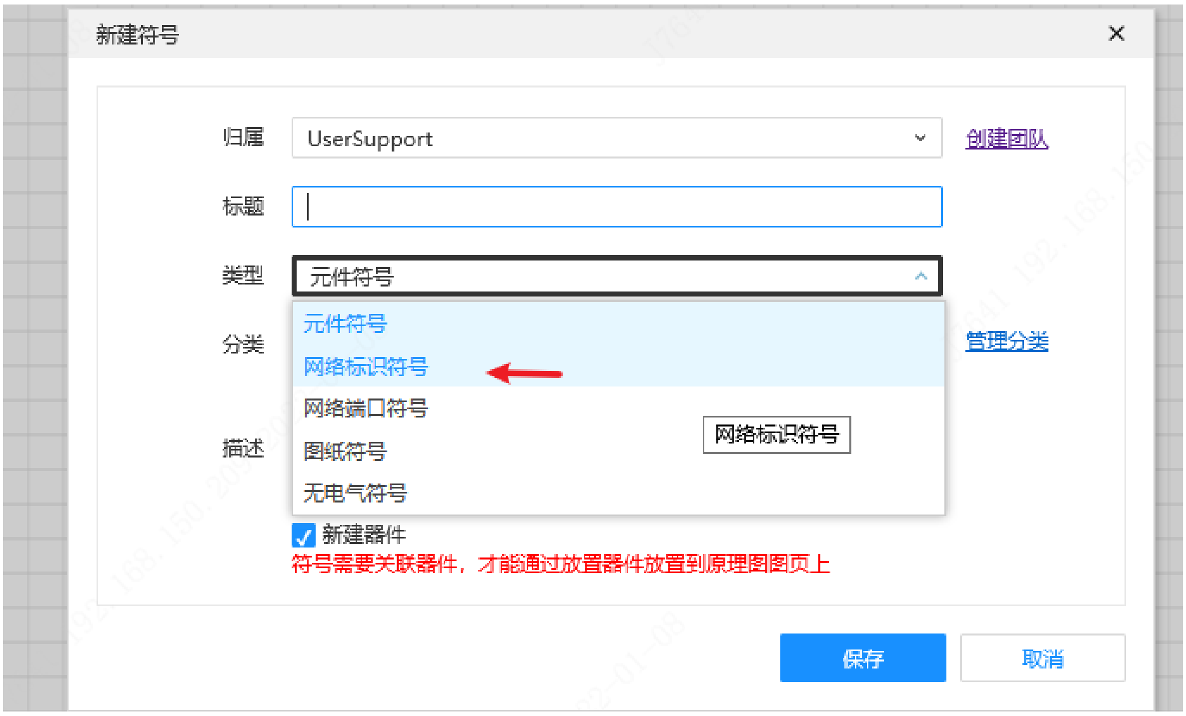
3. Each time you add a sub-library, a sub-library file with names such as .1, .2, .3, etc. will be generated.

- For components with sub-libraries, only one footprint can be specified in the parent library. If you specify different footprints for each sub-library, EasyEDA Pro will keep the last specified footprint as the component's footprint*
- The numbers of the sub-libraries must all be the same as the parent library.

Net Flag Symbol

If several network identifiers provided by the system do not meet your needs, you can create your own commonly used network identifier styles for easy reuse later.

Entrance: Top menu - New - Symbol, select Net Flag Symbol for type.



After drawing the style, place a pin. Cannot place more than already pins.

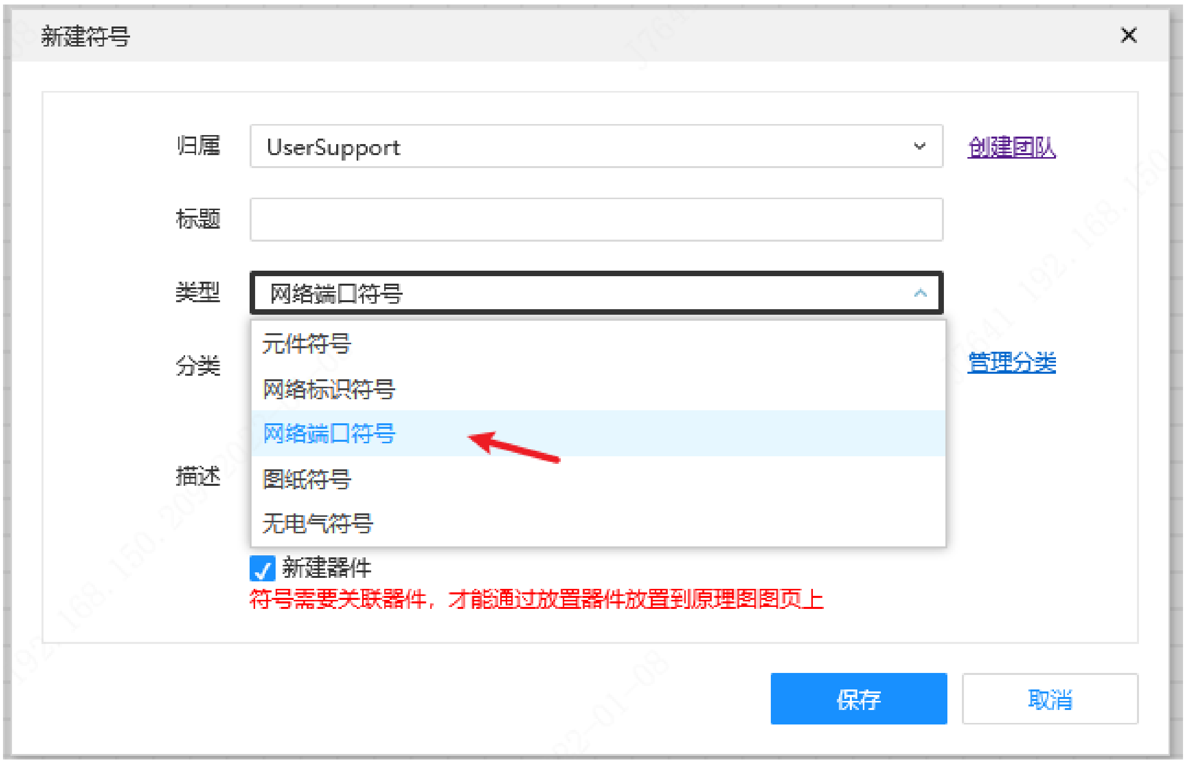
After saving, it can be found in the component library in the bottom panel: Bottom Panel - Component Library - Symbol - Personal.

Note: You must create a good symbol created by binding a device with the same name or a different name before placing the device in the schematic for use.

Net Port Symbol

If several network port symbols provided by the system do not meet your needs, you can create your own commonly used network port symbol styles for easy reuse later.

Entrance: Top menu - New - Symbol, type select Net Port Symbol.



After drawing the style, place a pin. Cannot place more than already pins.

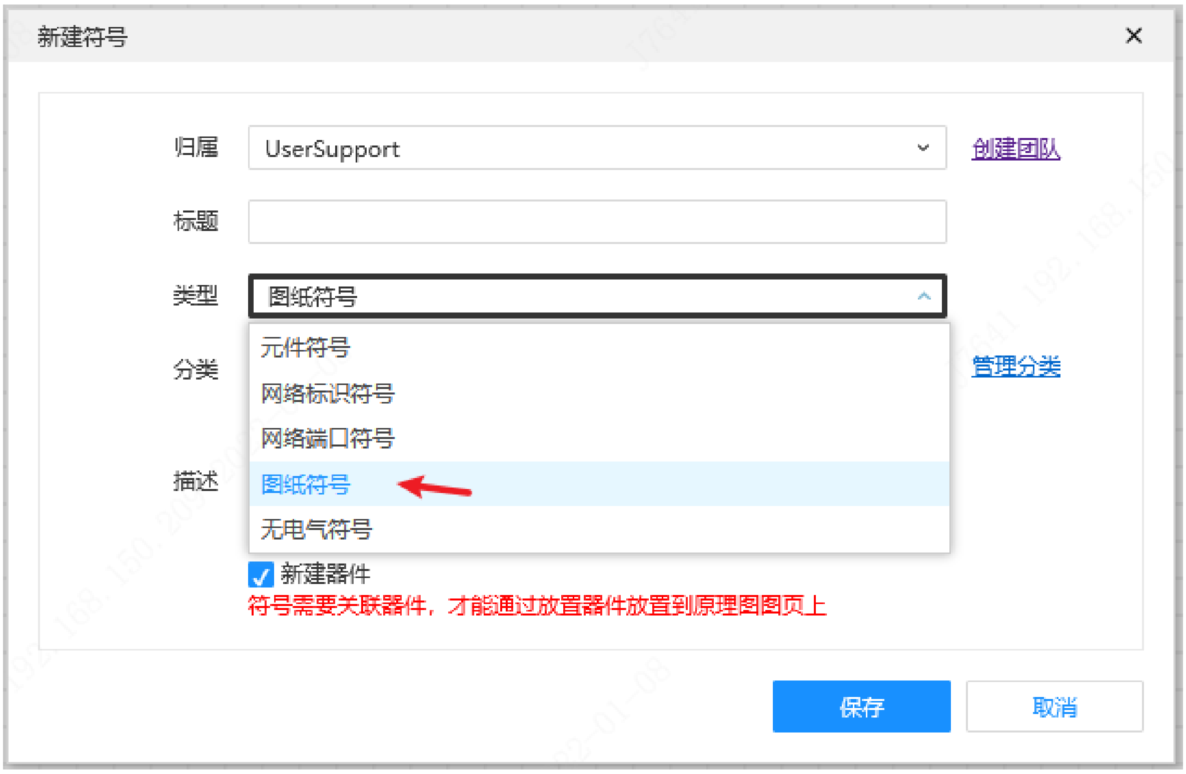
After saving, it can be found in the component library in the bottom panel: Bottom Panel - Component Library - Symbol - Personal.

Note: You must create a good symbol created by binding a device with the same name or a different name before placing the device in the schematic for use.

Drawing Symbols

If several drawing symbols provided by the system do not meet your needs, you can create your own commonly used drawing symbol styles for easy reuse later.

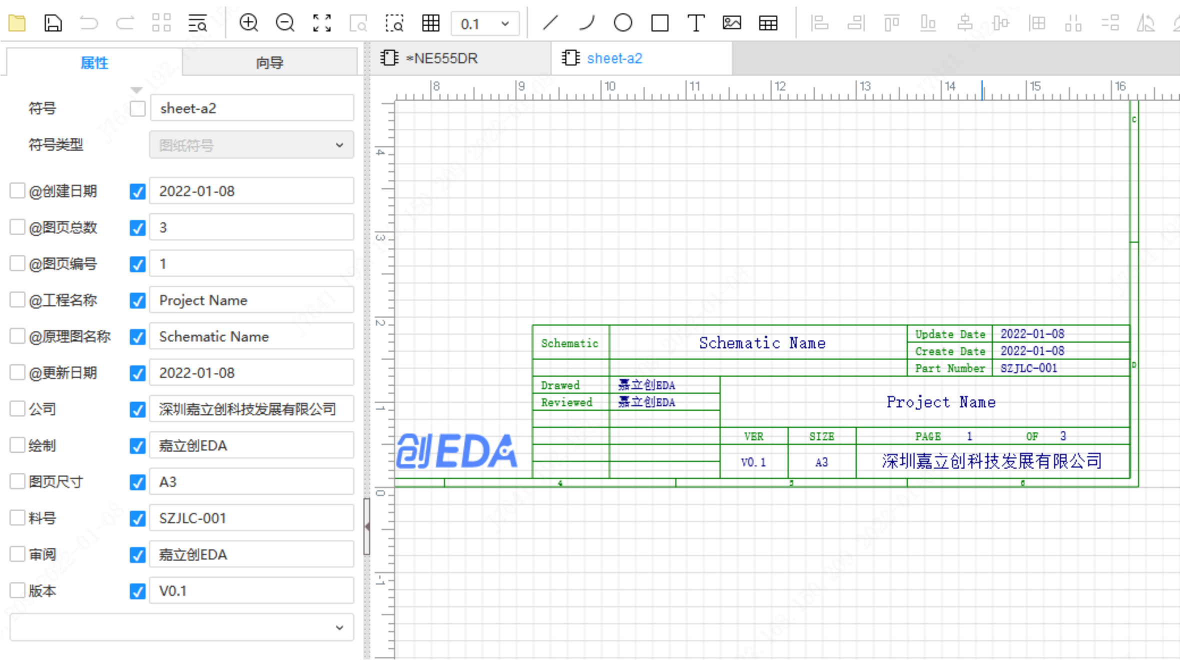
Entrance: Top Menu - New - Symbol, Type select Sheet Symbol.



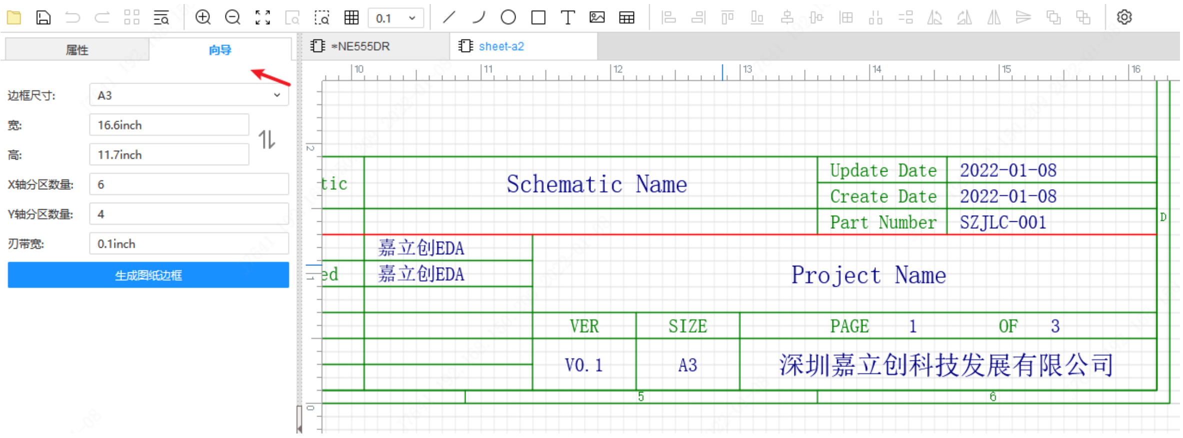
Drawing symbols cannot place pins.

Create steps:

1. After creating a new drawing symbol, set the properties required by the drawing in the properties on the left, and check the values that need to be displayed on the canvas. Attributes starting with @ are built-in attributes in the system. After they are placed on the page, these attributes will be updated automatically, and there is no need to set values in advance.



2. You can manually adjust the drawing boundaries, the size and position of the lines in the lower right corner of the header, etc., or you can use the wizard on the left to create



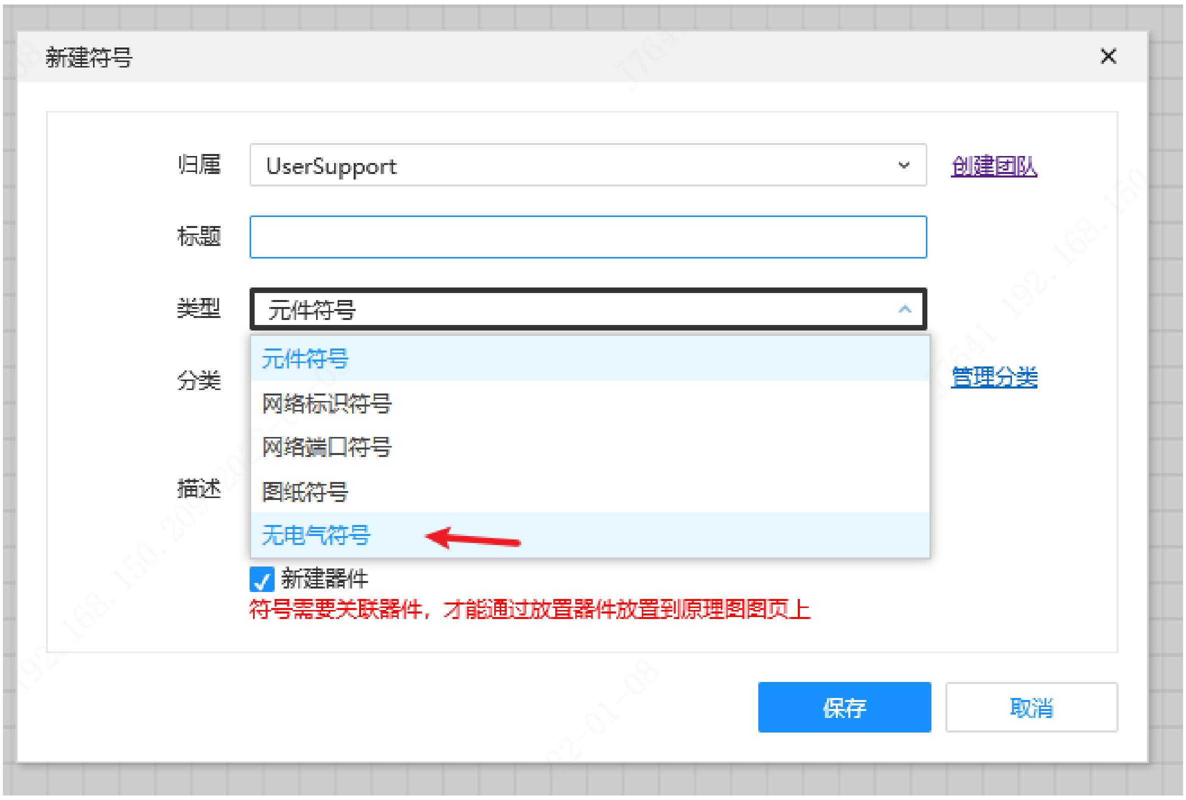
3. After the adjustment is completed, save it. After saving, it can be found in the component library in the bottom panel: Bottom Panel - Component Library - Symbol - Personal.

Note: You must create a good symbol created by binding a device with the same name or a different name before placing the device in the schematic for use.

No Electrical Symbols

If you want to create a logo library or other markup library for easy reuse later, you can create no electrical symbols.

Entrance: Top menu - New - Symbol, type select No Electrical Symbol.



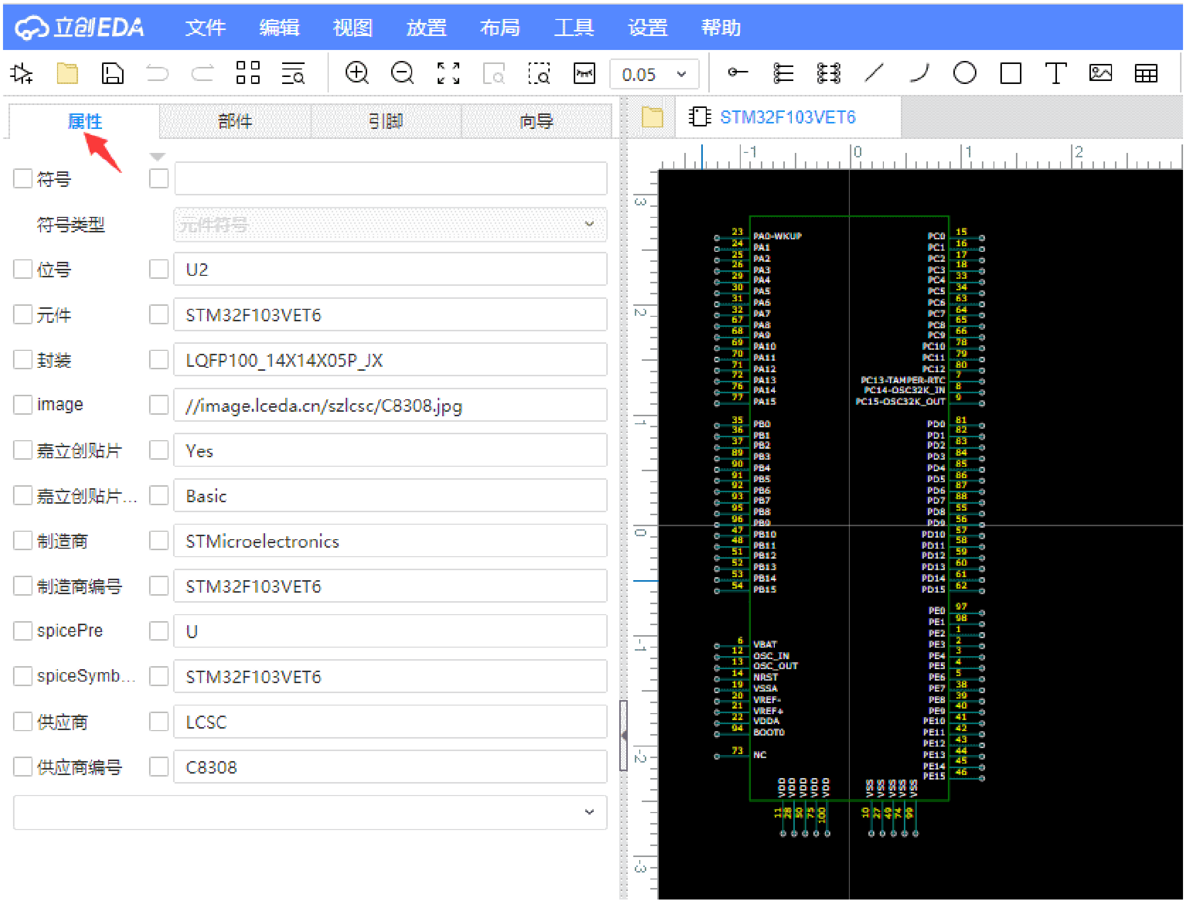
Pins cannot be placed without electrical symbols.

After saving, it can be found in the component library in the bottom panel: Bottom Panel - Component Library - Symbol - Personal.

Note: You must create a good symbol created by binding a device with the same name or a different name before it can be placed and used in the schematic.

symbol attribute

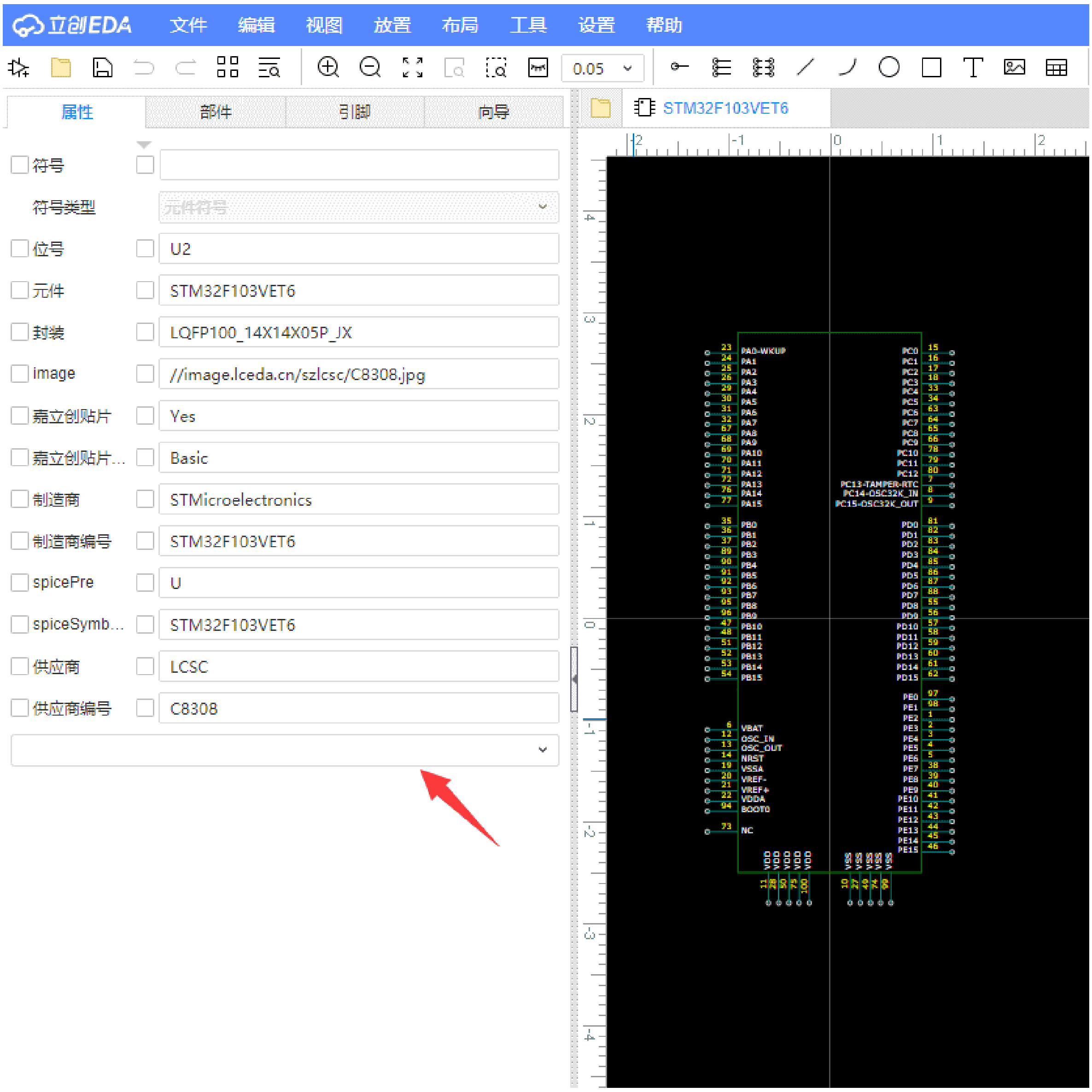
The property panel of the symbol is on the left side of the symbol editing interface, the property panel of the primitive is on the right, and the property panel of the symbol on the left.



- **symbol**: The name of the symbol.
- **Symbol Type**: Selected at creation and cannot be changed.
- **Tag**: The tag of the symbol is related to the tag placed on the schematic.

add properties

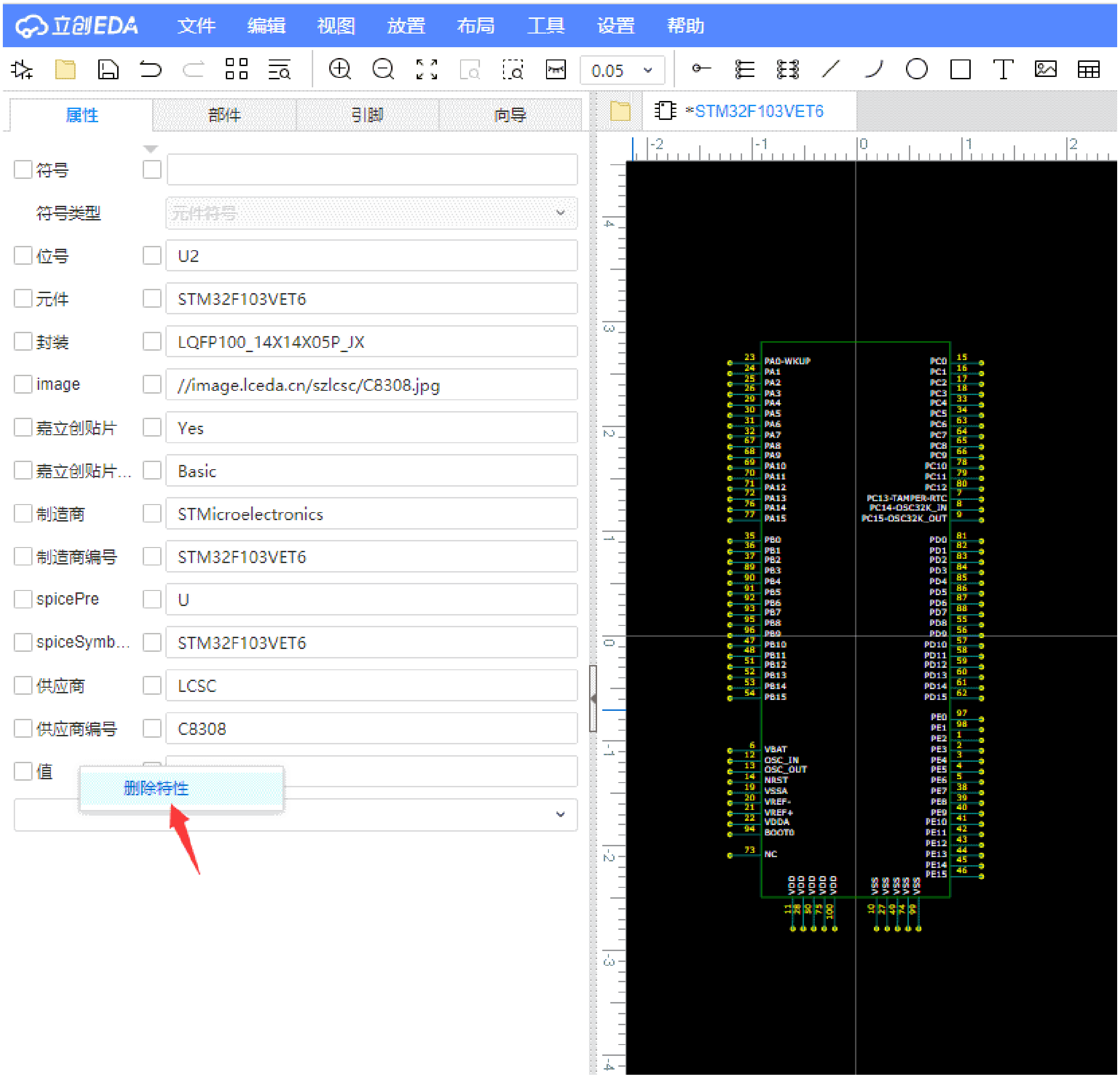
Click the drop-down selection box, select the attribute to be added, and the addition is successful.



delete attribute

Select the added attribute - right mouse button - delete attribute

Only user-added attributes can be deleted, and system default attributes cannot be deleted.

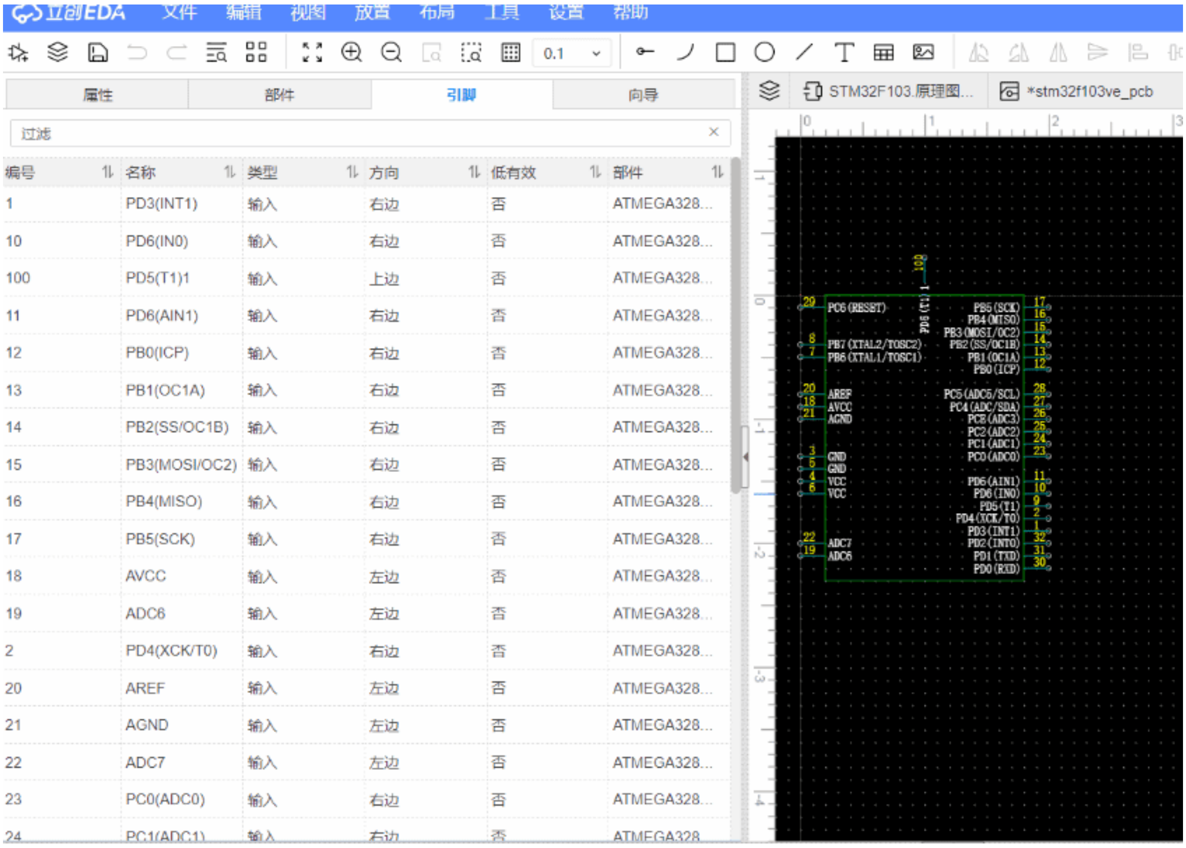


Pin Manager

The pin manager is to display all the pins you place in the symbol editor in the pin editor, including the sub-symbol library, and you can quickly modify the pins.

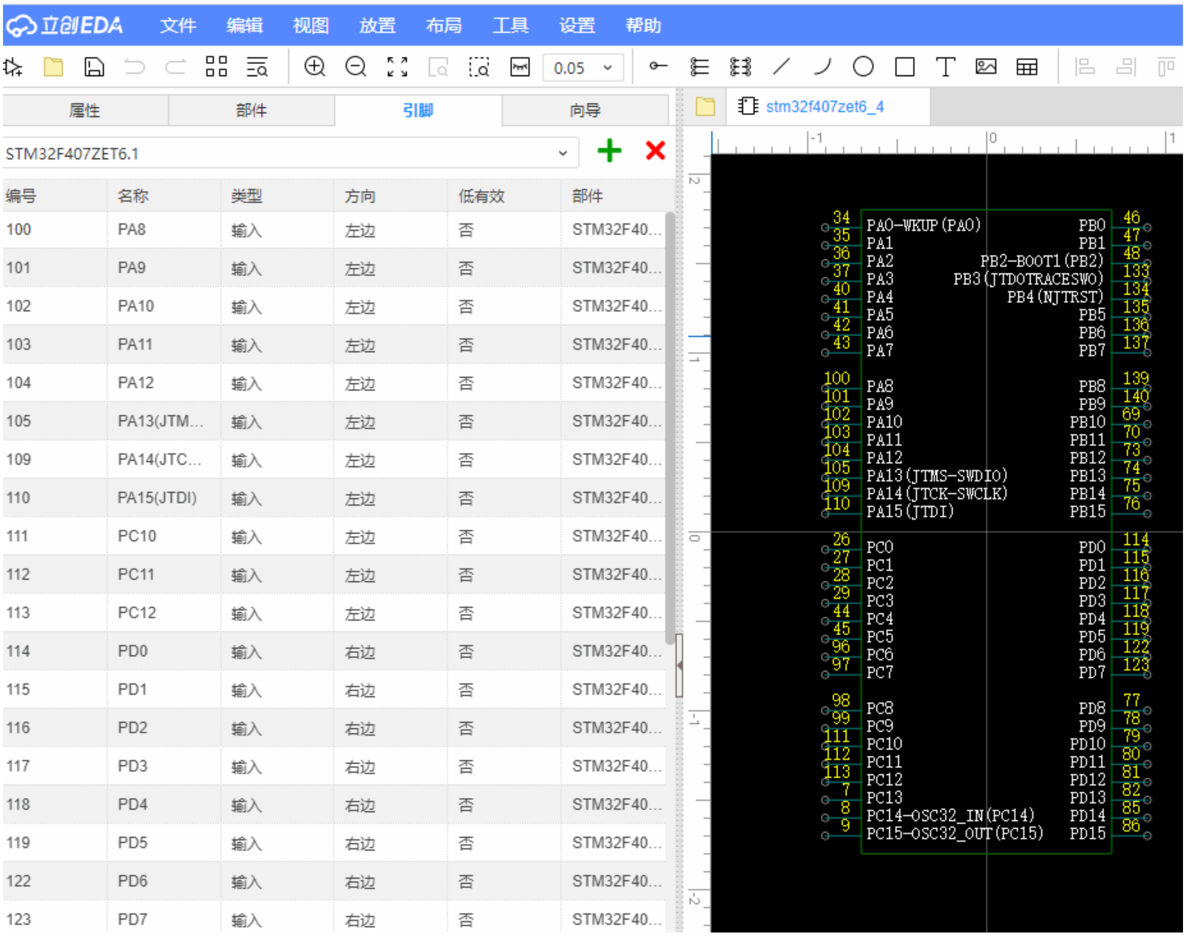
Select the pin in the left view of the symbol editor interface

- **Number**: Pin number.
- **Name**: The name of the pin.
- **Type**: Pin input, output, bidirectional type selection.
- **Orientation**: The orientation of the pin is selected.
- **Active low**: Select whether the pin is active low or not.
- **Parts**: Displays the name of the current symbol and which page of which sub-library the symbol is in.



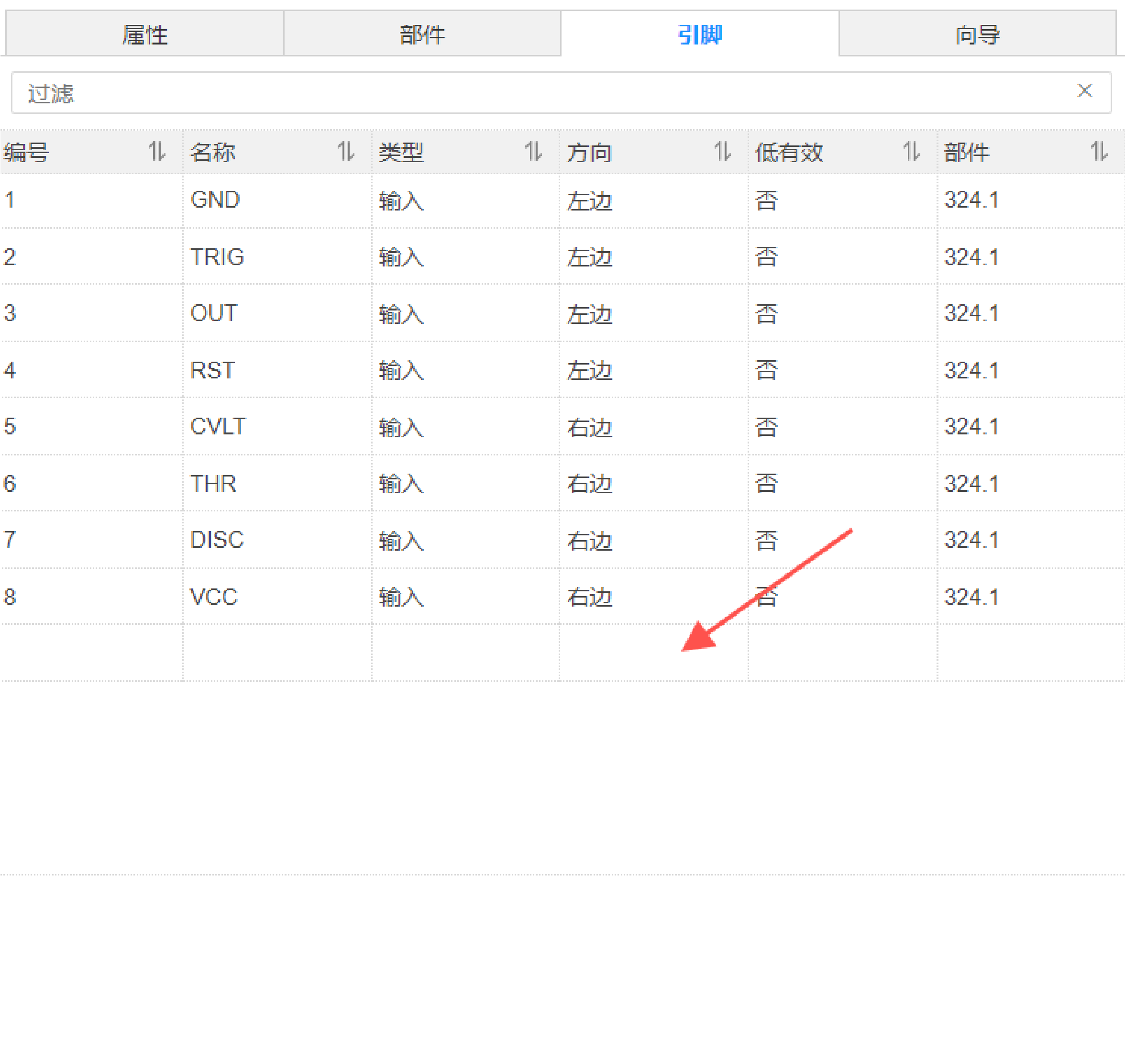
Fill in the pin name

After filling in the pin name, you do not need to click OK, the system will automatically fill in the pin in the figure page according to the name you filled in.



Add Pin

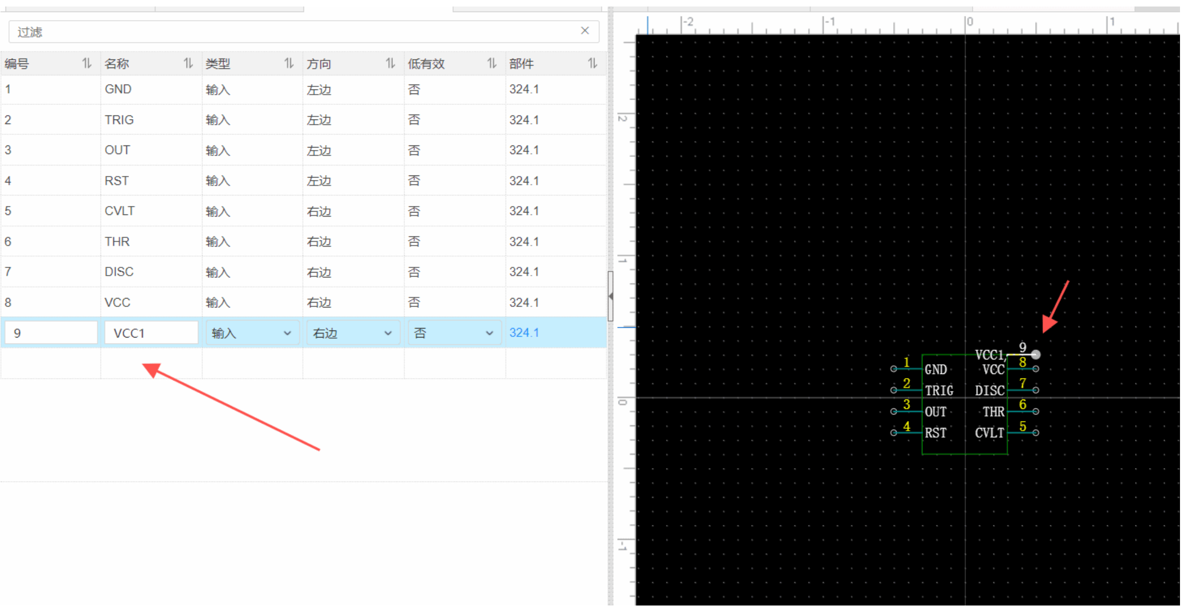
The pin editor can also quickly place pins, just click the blank form below in the pin editor to quickly place pins



属性 | 部件 | 引脚 | 向导

过滤

编号	名称	类型	方向	低有效	部件
1	GND	输入	左边	否	324.1
2	TRIG	输入	左边	否	324.1
3	OUT	输入	左边	否	324.1
4	RST	输入	左边	否	324.1
5	CVLT	输入	右边	否	324.1
6	THR	输入	右边	否	324.1
7	DISC	输入	右边	否	324.1
8	VCC	输入	右边	否	324.1



过滤

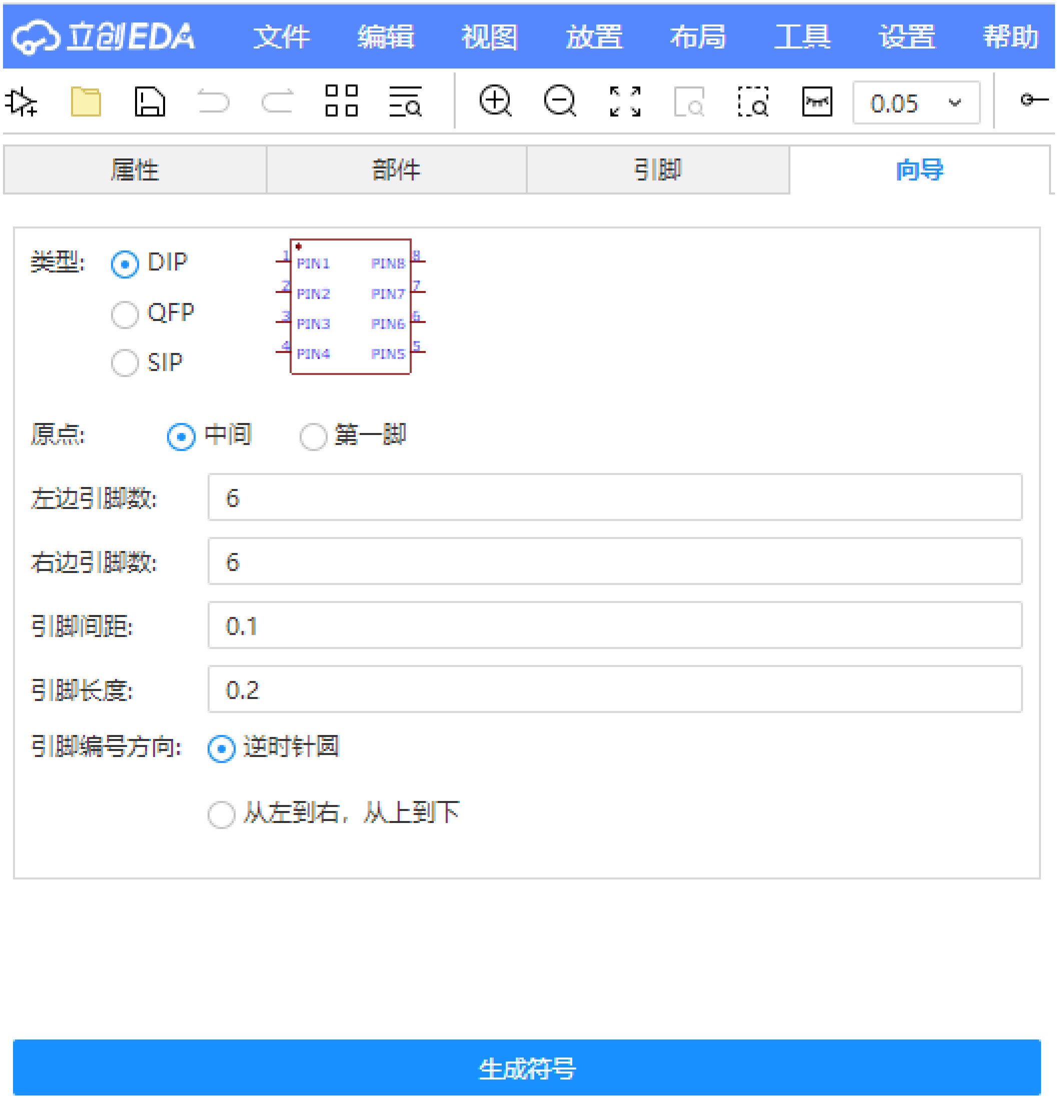
编号	名称	类型	方向	低有效	部件
1	GND	输入	左边	否	324.1
2	TRIG	输入	左边	否	324.1
3	OUT	输入	左边	否	324.1
4	RST	输入	左边	否	324.1
5	CVLT	输入	右边	否	324.1
6	THR	输入	右边	否	324.1
7	DISC	输入	右边	否	324.1
8	VCC	输入	右边	否	324.1
9	VCC1	输入	右边	否	324.1

Symbol Wizard

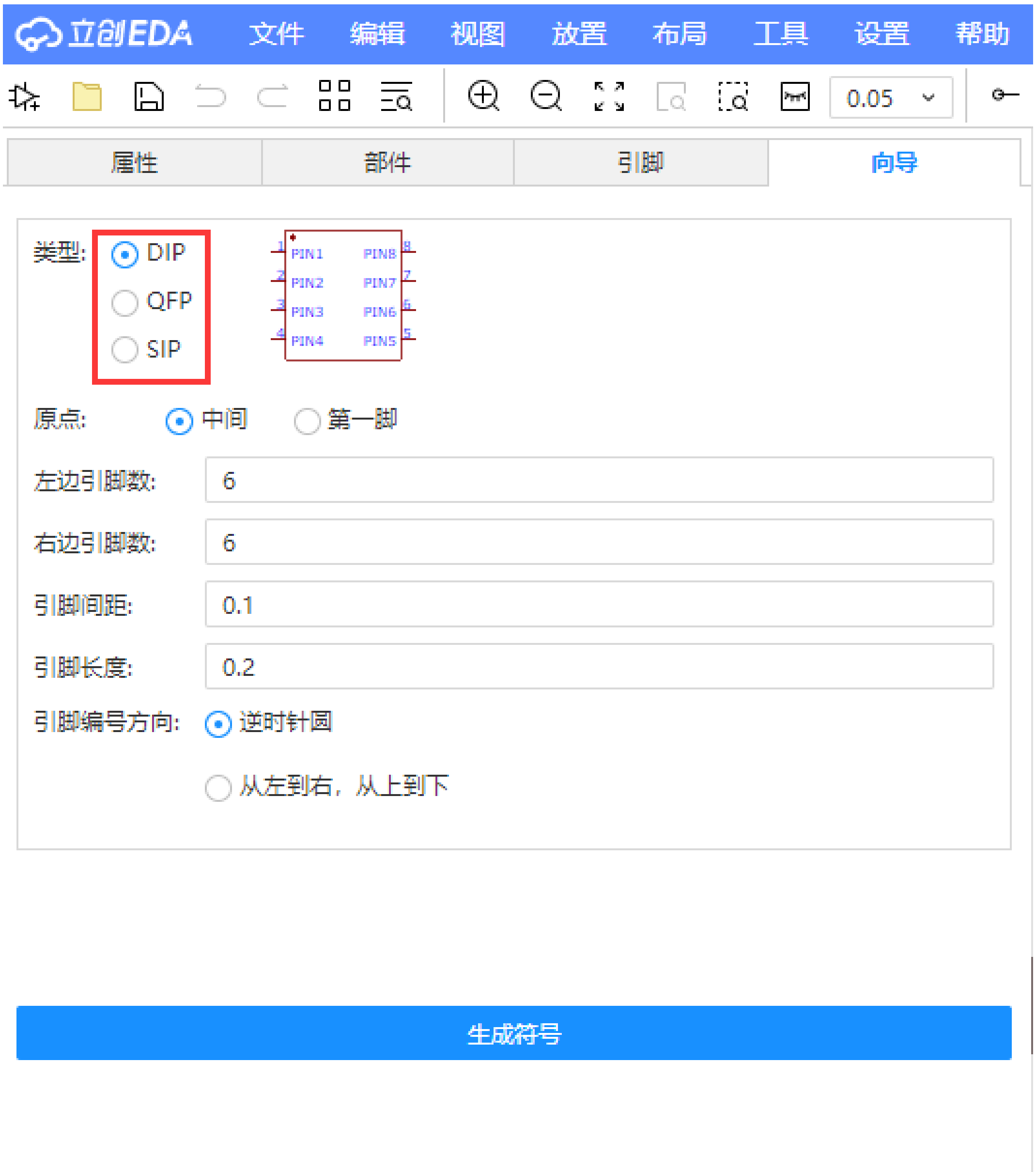
The EasyEDA Pro symbol wizard can quickly create symbols. Currently, it only supports the rapid creation of three types of symbols: DIP, QFP and SIP of IC type.

Steps:

1. In the view on the left side of the Symbol Editor interface, select Wizard

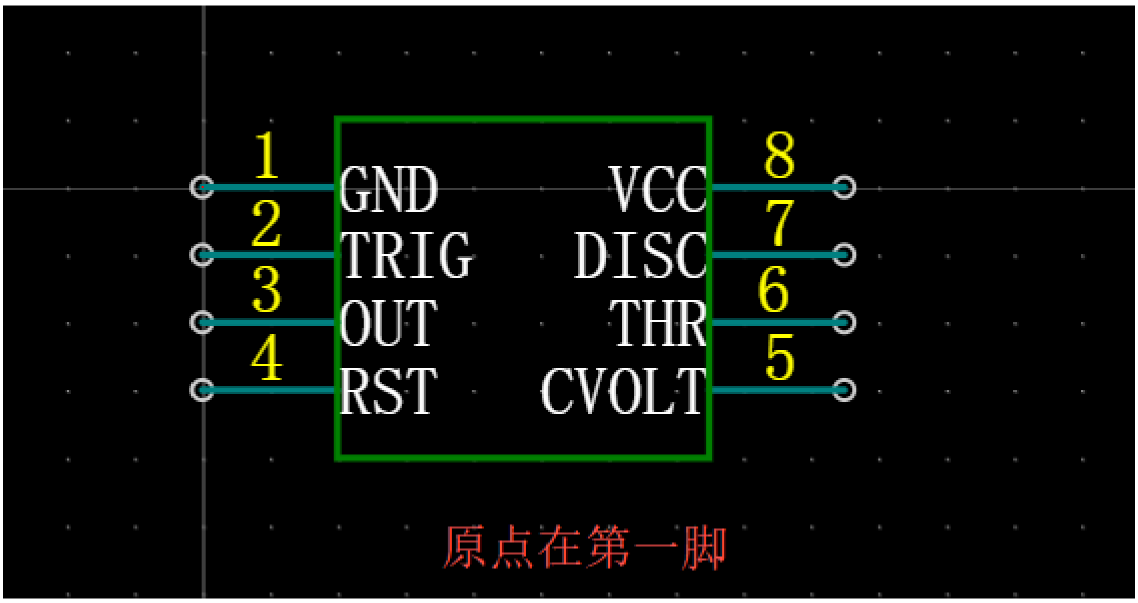


2. Select the type of symbol to be created

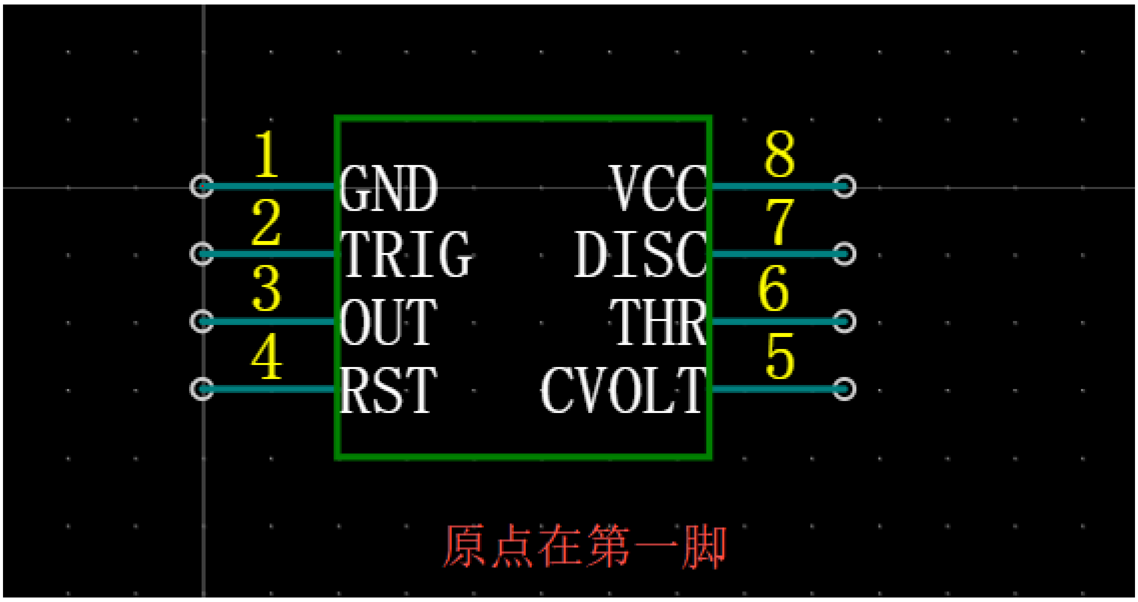


3. Set the origin

The origin is the first pin: set the first pin of the symbol as the origin, and the coordinate values of the first pin are all 0.



Set the origin at the center: Set the center position of the symbol as the origin, and the coordinate value of the center position of the symbol is 0.



Fill in the corresponding number of pins

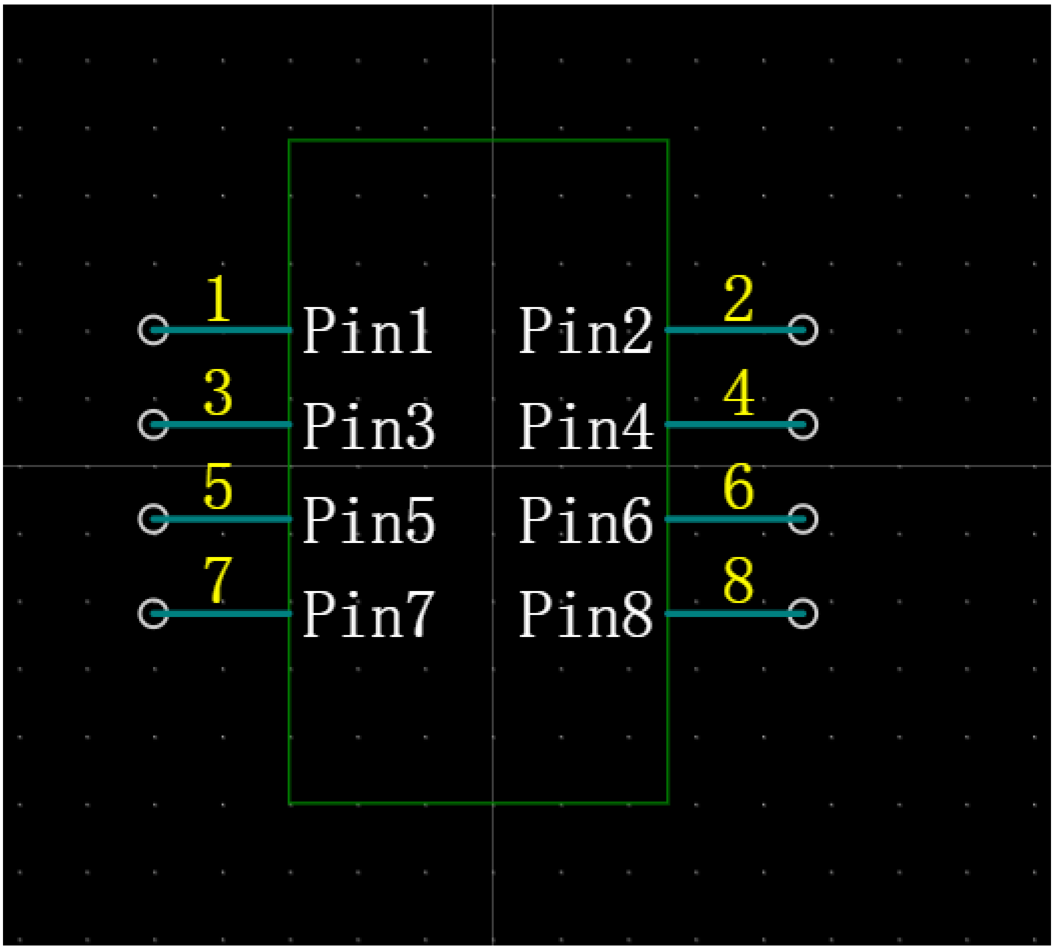
Set the spacing of the pins

Note: The pin spacing cannot be set too small, otherwise, when generating symbols, it will cause the pin names to overlap like the picture below. **The recommended spacing is 0.15 inches.**

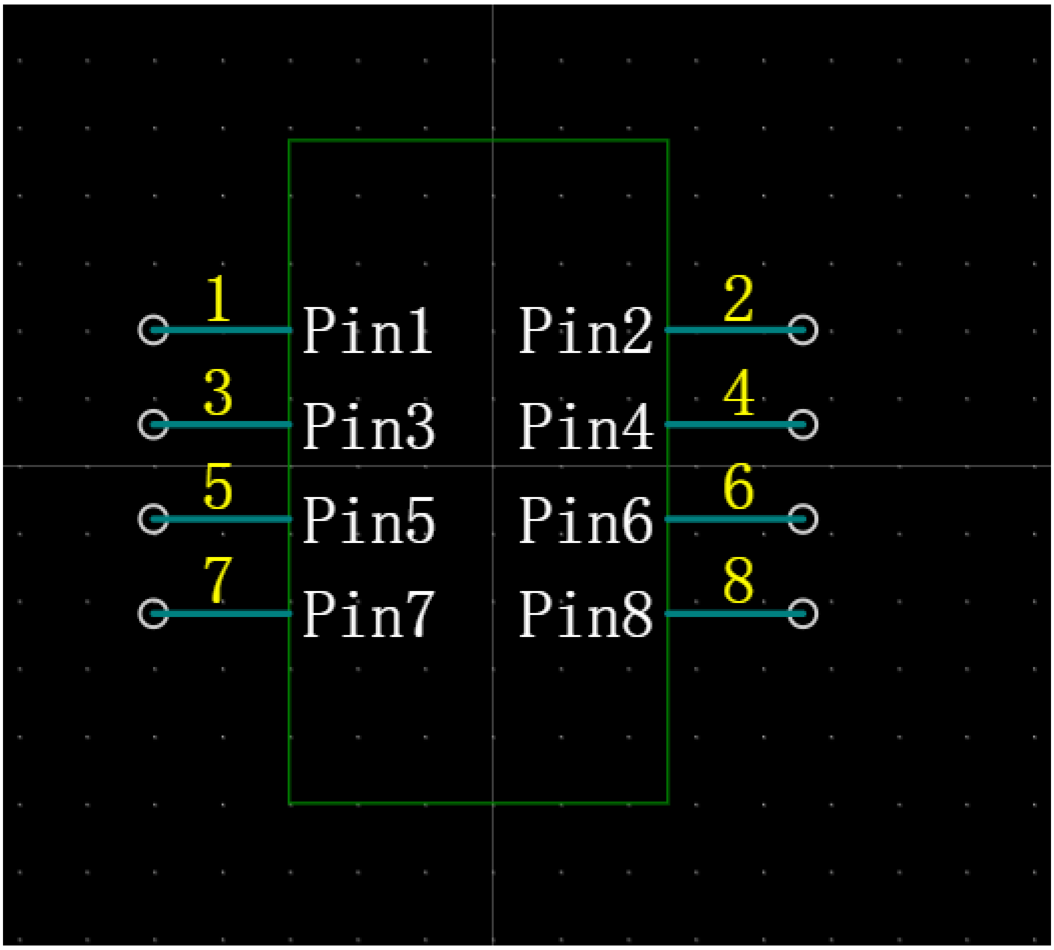
Orientation of pin numbers

The numbering direction is counter-clockwise circle generation

The pin numbering direction is generated from top to bottom. The difference between the two is that the position of the generated serial number of the pin is different, which is more suitable for symbols of other types of pin numbering direction.

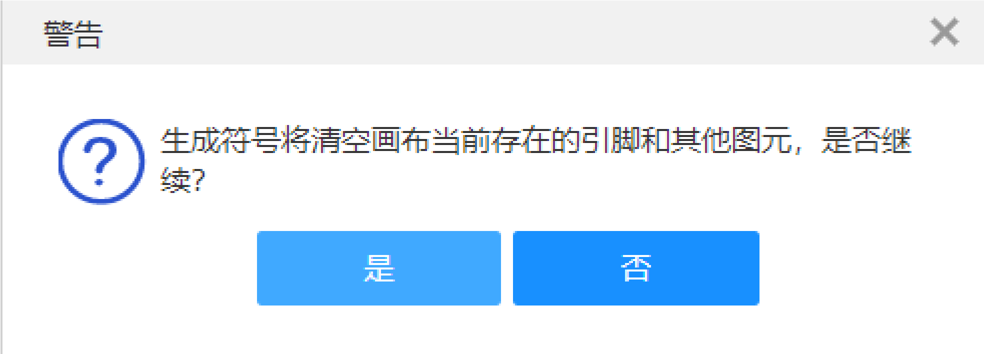


Click to generate

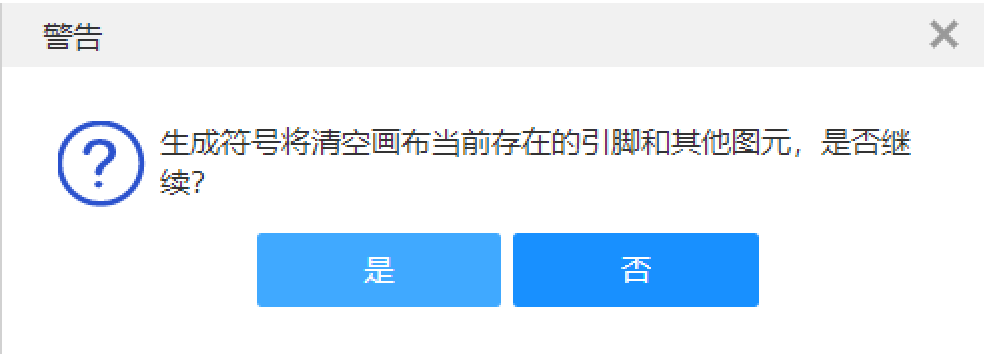


Select Yes to clear the symbols in your current symbol editor and generate them.

Select Else to not generate symbols.



Generated DIP type symbols



Pin Properties

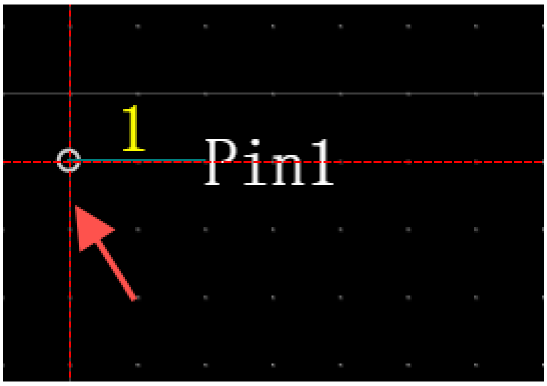
pin properties

The pins of a schematic symbol are one of the most important parts of a component, allowing wire connections to be made to make up an entire circuit.

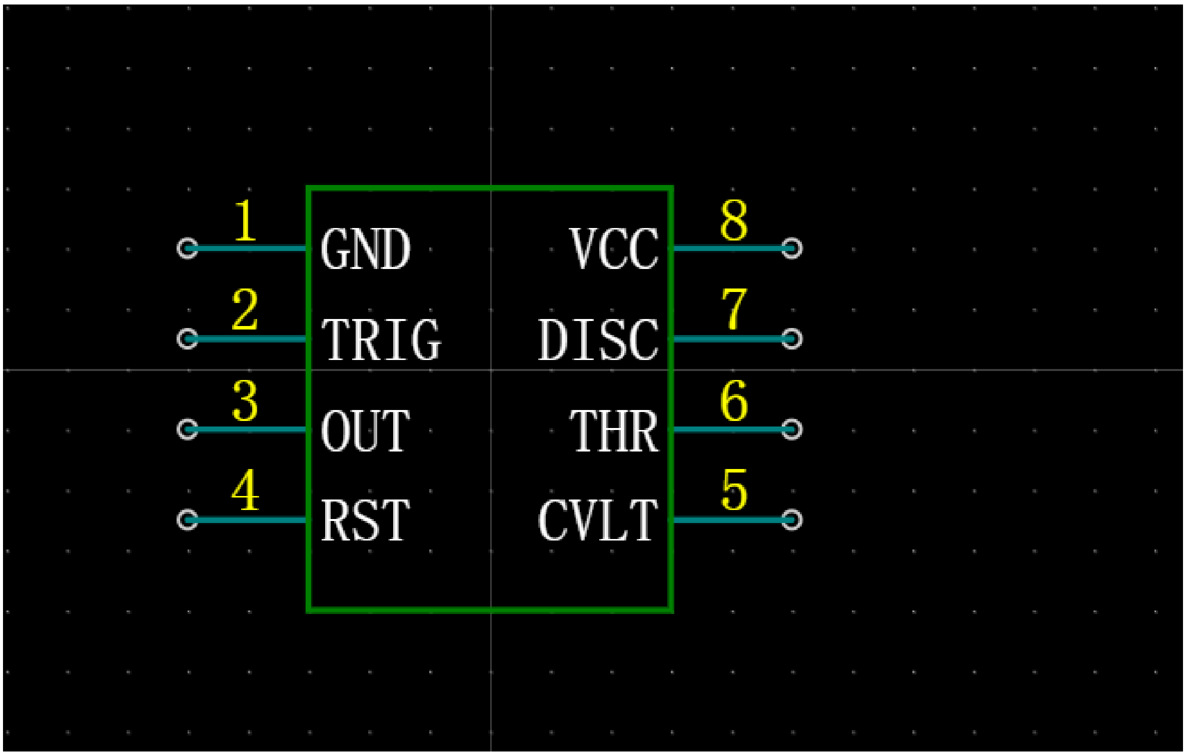
Top Toolbar - Place - Pin or use shortcut "Ait + P

Pin orientation

Before placing on the canvas, you can press the space bar to select to the desired direction. The circled end of the pin is where a wire connection can be made, so this point should normally be placed outwards.

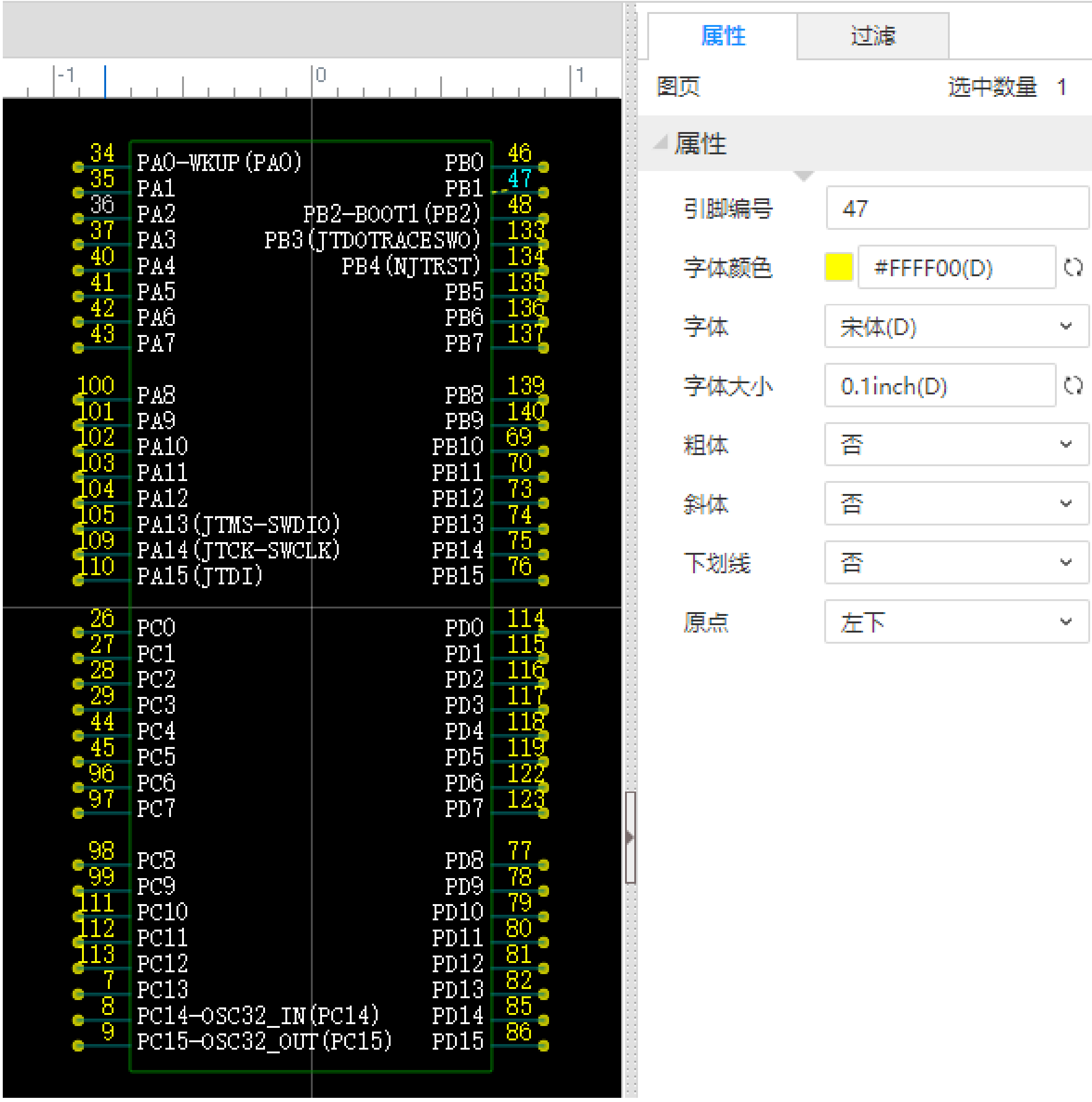


correct format



pin properties

When you select a pin, you can view and modify its properties in the properties panel on the right.



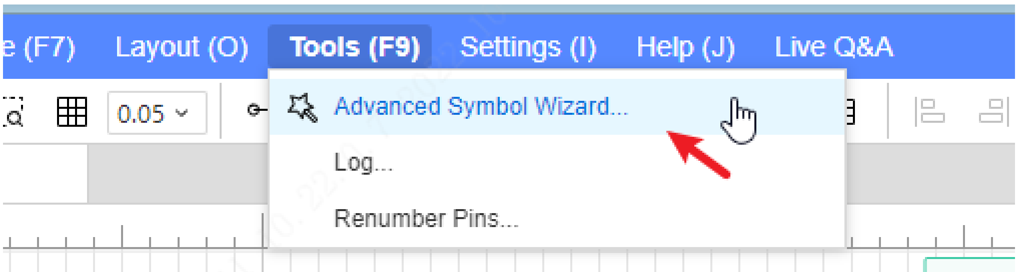
- **Name**: The name of the pin, as shown in the above figure, the name of the selected pin is "VCC".
- **ID**: The ID of the pin is fixed by the system and cannot be changed.
- **Type**: selectable input, output and bidirectional.
- **Pin Number**: The pin number selected as above is "8". **The number corresponds to the pad number of the footprint corresponding to the component, and the numbers of the two need to be consistent.** You can also use numbers with letters, such as A1, B1, C1, A2, B2, etc.
- **Pin Direction**: Only supports up, down, left, and right directions, other directions are not supported for the time being.
- **NON-PIN**: Whether it is a pin selection.
- **Length**: Pin length.
- **Pin Color**: You can modify the color of the pins.
- **Added attributes**: EasyEDA Pro can customize the added attributes.

image-20201217110955842

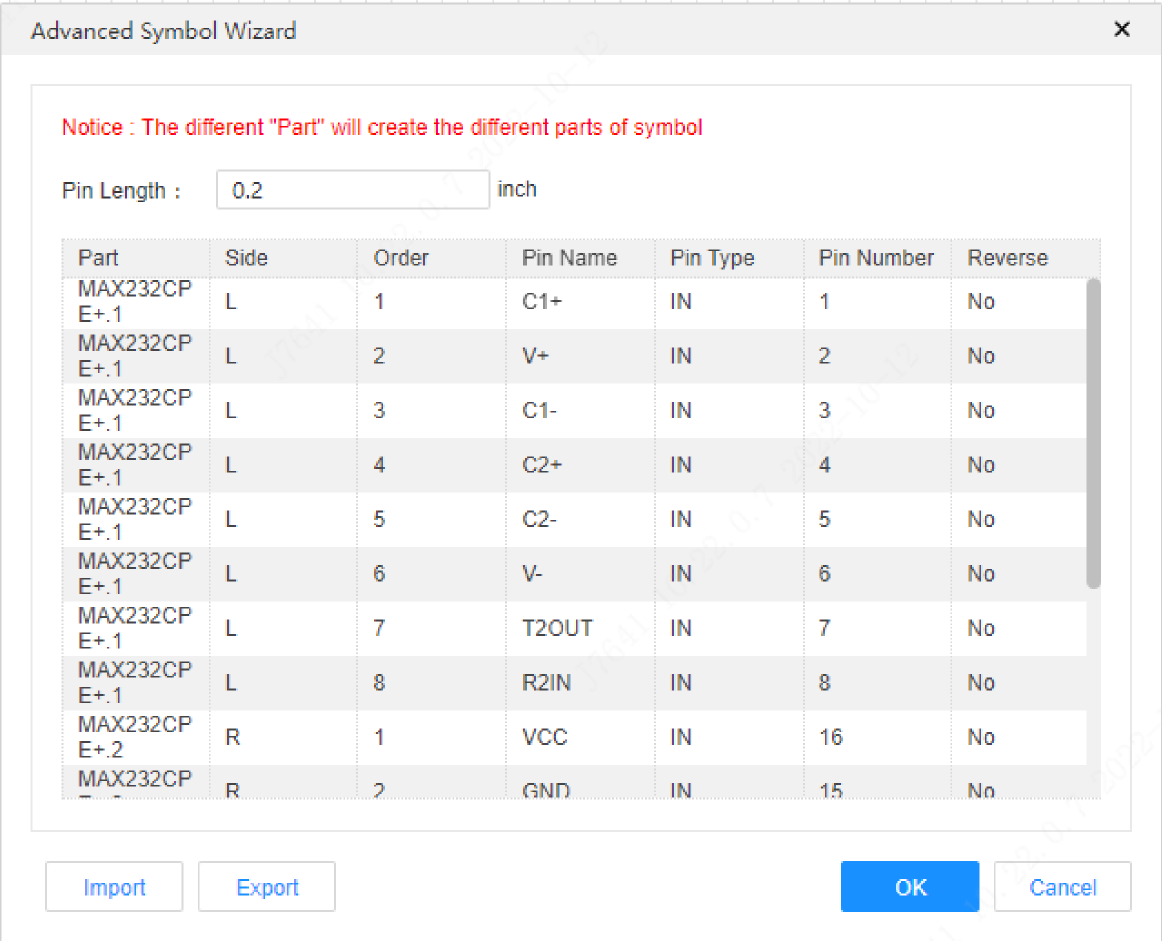
Advanced Symbol Wizard

The advanced symbol wizard is used to quickly create symbols for IC type chips. The advanced symbol wizard does not distinguish the types of symbols. It only requires the user to fill in the corresponding data in the template, and the system can generate symbols according to the filled data.

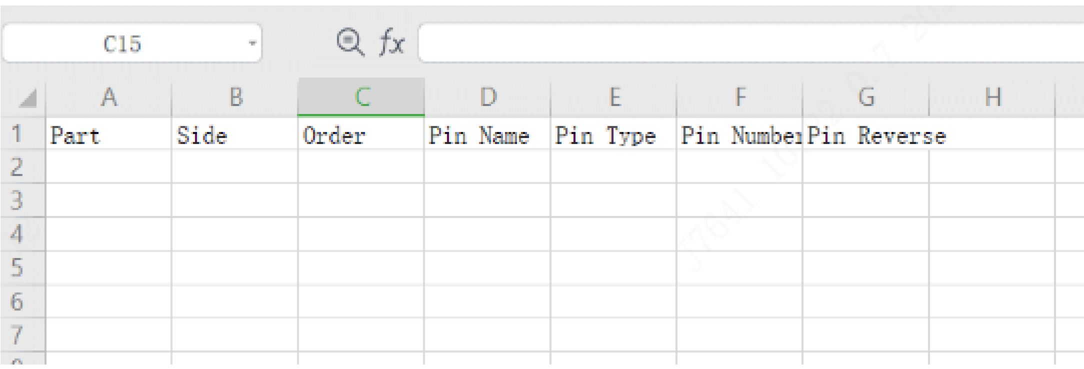
Operation steps: Top Menu - Tools - Advanced Symbol Wizard



Export the template in the template popup



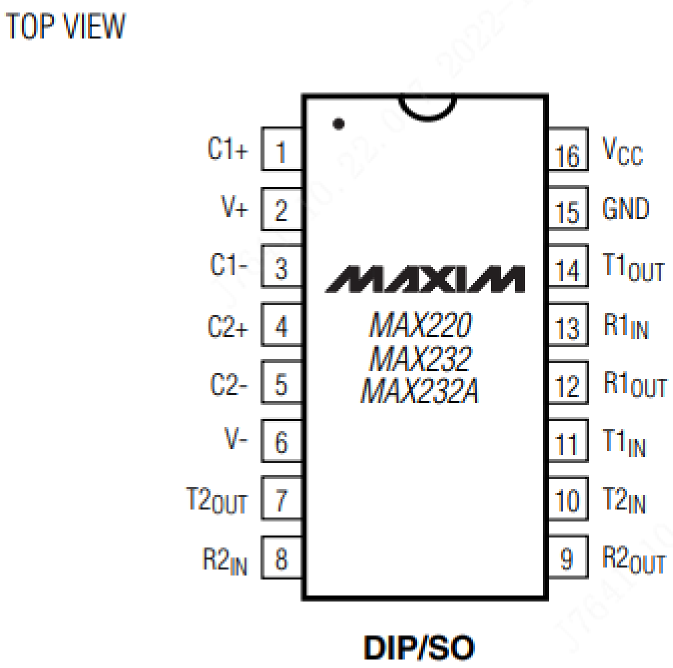
Open the template with "office" "WPS" Excel tool



- **Part**: represents the sub-library page of the symbol, not the pin number;

- **Side (direction)**: the direction of the pin, L represents the left side, T represents the upper side, R represents the right side, and B is the lower side;
- **Order (serial number)**: Determine the positional relationship of the pins in different directions;
- **Pin Name**: the name of the pin;
- **Pin Type**: Set whether the pin is input (IN) or output (OUT) type;
- **Pin Number**: the number of the pin;
- **Pin Reverse (non-pin)**: whether it is a pin type;

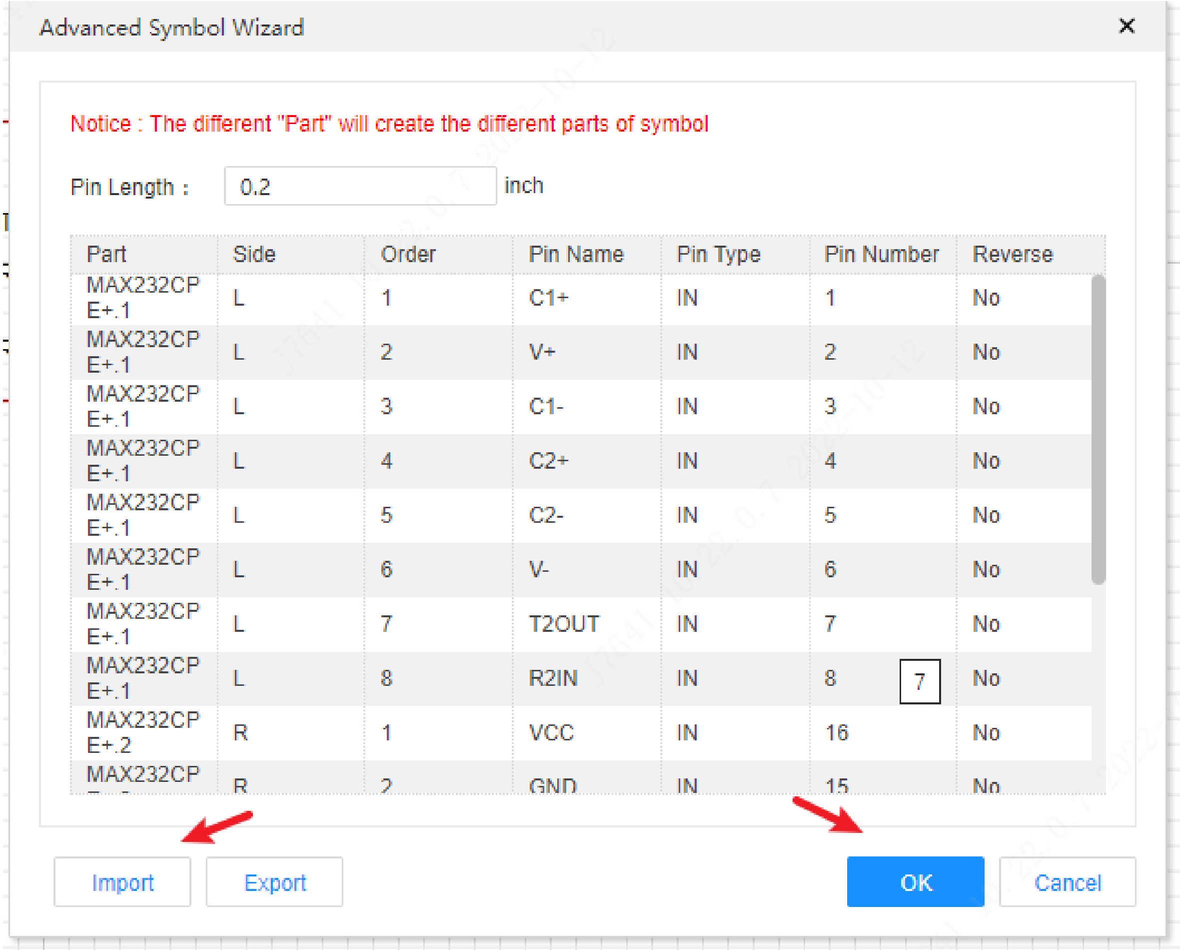
For example：MAX232CPE+



Fill in the pins information at the table:

Part	Side	Order	Pin Name	Pin Type	Pin Number	Reverse
MAX232CPE+.1	L	1	C1+	IN	1	No
MAX232CPE+.1	L	2	V+	IN	2	No
MAX232CPE+.1	L	3	C1−	IN	3	No
MAX232CPE+.1	L	4	C2+	IN	4	No
MAX232CPE+.1	L	5	C2−	IN	5	No
MAX232CPE+.1	L	6	V−	IN	6	No
MAX232CPE+.1	L	7	T2OUT	IN	7	No
MAX232CPE+.1	L	8	R2IN	IN	8	No
MAX232CPE+.1	R	1	VCC	IN	16	No
MAX232CPE+.1	R	2	GND	IN	15	No
MAX232CPE+.1	R	3	T1OUT	IN	14	No
MAX232CPE+.1	R	4	R1IN	IN	13	No
MAX232CPE+.1	R	5	R1OUT	IN	12	No
MAX232CPE+.1	R	6	T1IN	IN	11	No
MAX232CPE+.1	R	7	T2IN	IN	10	No
MAX232CPE+.1	R	8	R2OUT	IN	9	No

Open advanced symbol wizard, import the table

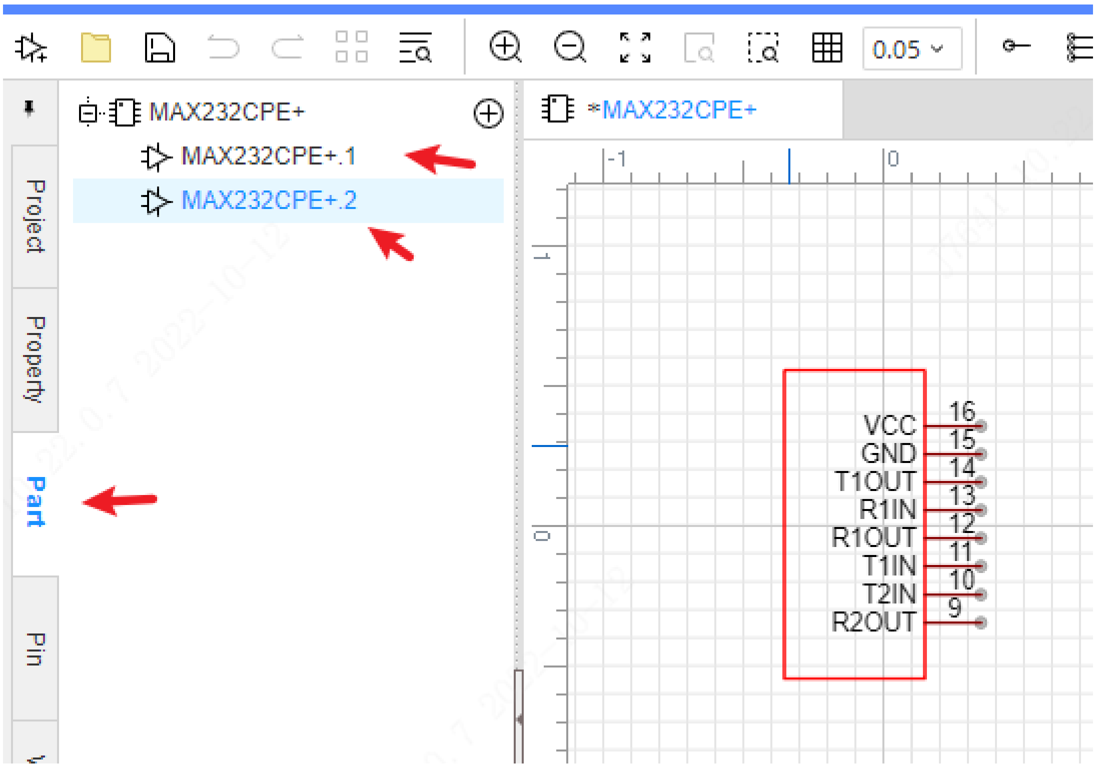


Click OK to finish wizard.

If you set different Part at the table,

	A	B	C	D	E	F	G	H
1	Part	Side	Order	Pin Name	Pin Type	Pin Number	Reverse	
2	MAX232CPE+.1	L	1	C1+	IN	1	No	
3	MAX232CPE+.1	L	2	V+	IN	2	No	
4	MAX232CPE+.1	L	3	C1-	IN	3	No	
5	MAX232CPE+.1	L	4	C2+	IN	4	No	
6	MAX232CPE+.1	L	5	C2-	IN	5	No	
7	MAX232CPE+.1	L	6	V-	IN	6	No	
8	MAX232CPE+.1	L	7	T2OUT	IN	7	No	
9	MAX232CPE+.1	L	8	R2IN	IN	8	No	
0	MAX232CPE+.2	R	1	VCC	IN	16	No	
1	MAX232CPE+.2	R	2	GND	IN	15	No	
2	MAX232CPE+.2	R	3	T1OUT	IN	14	No	
3	MAX232CPE+.2	R	4	R1IN	IN	13	No	
4	MAX232CPE+.2	R	5	R1OUT	IN	12	No	
5	MAX232CPE+.2	R	6	T1IN	IN	11	No	
6	MAX232CPE+.2	R	7	T2IN	IN	10	No	
7	MAX232CPE+.2	R	8	R2OUT	IN	9	No	
8								
9								

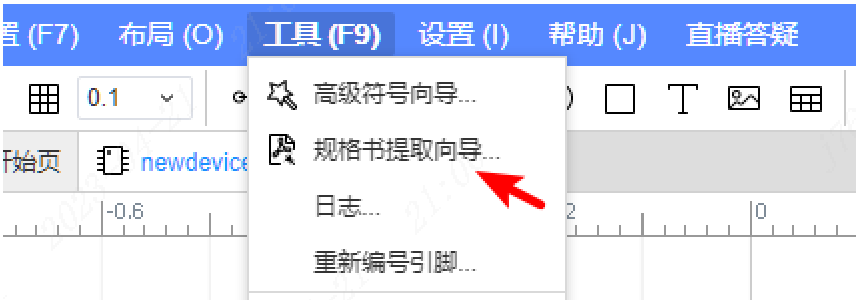
after imported, you can find and edit different parts at left hand Part tab



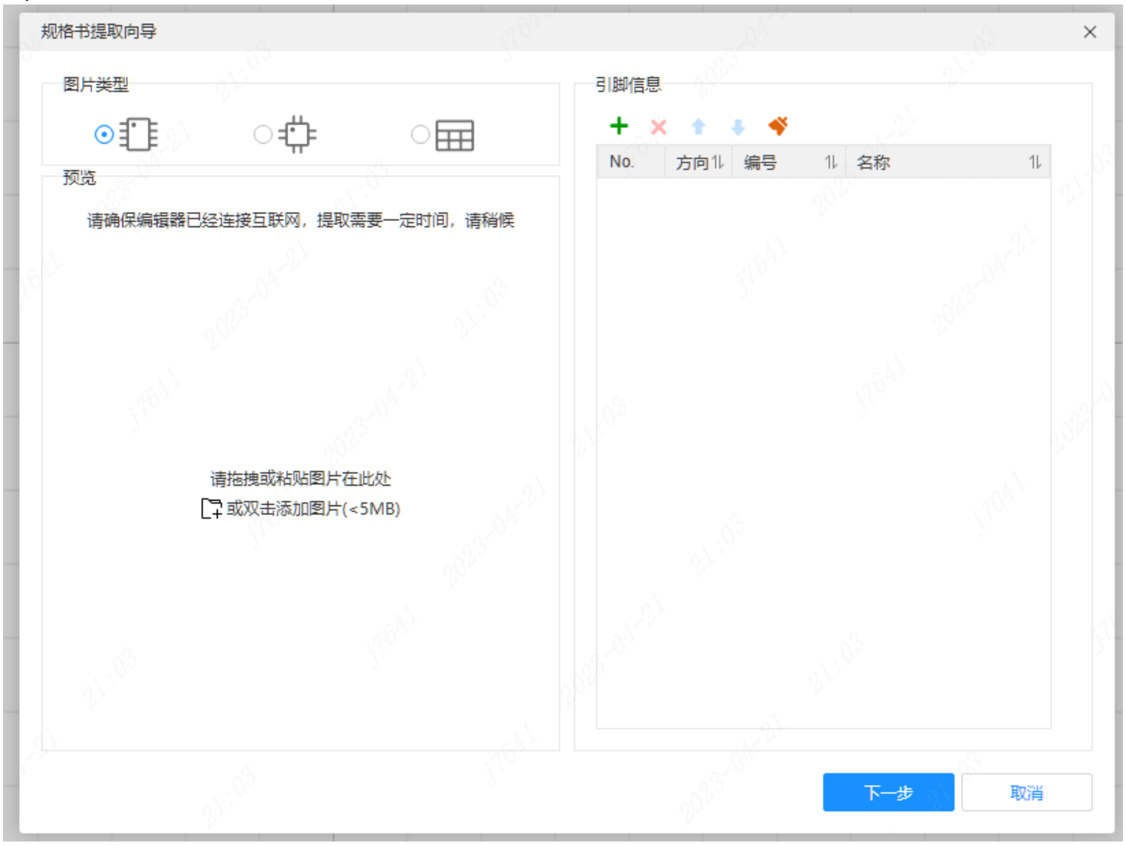
Datasheet Extract Wizard

The Datasheet Extract wizard is used to quickly create symbols for IC type chips. Users only need to paste the screenshot of the Datasheet, and the system can identify the pin information of the symbol according to the picture, which is convenient for quickly creating symbols.

Entrance: Top Menu - Tools - Datasheet Extract Wizard. Note that this function needs to be connected to the Internet, and it may not work properly in the full offline mode of the client.

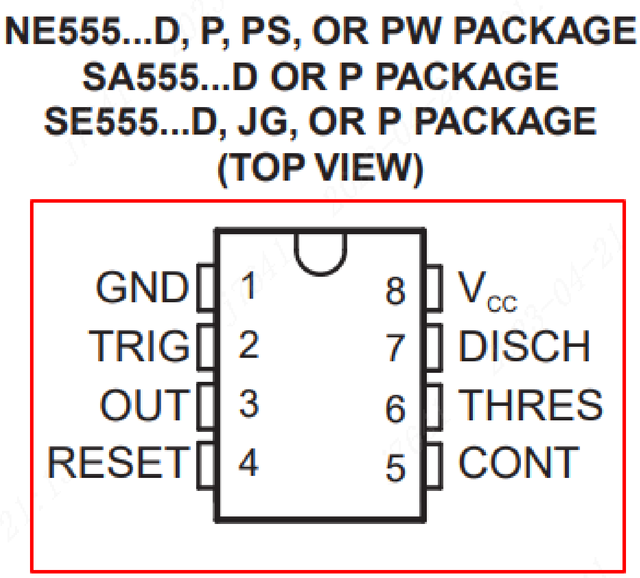


1. Open the Datasheet Extract Wizard.

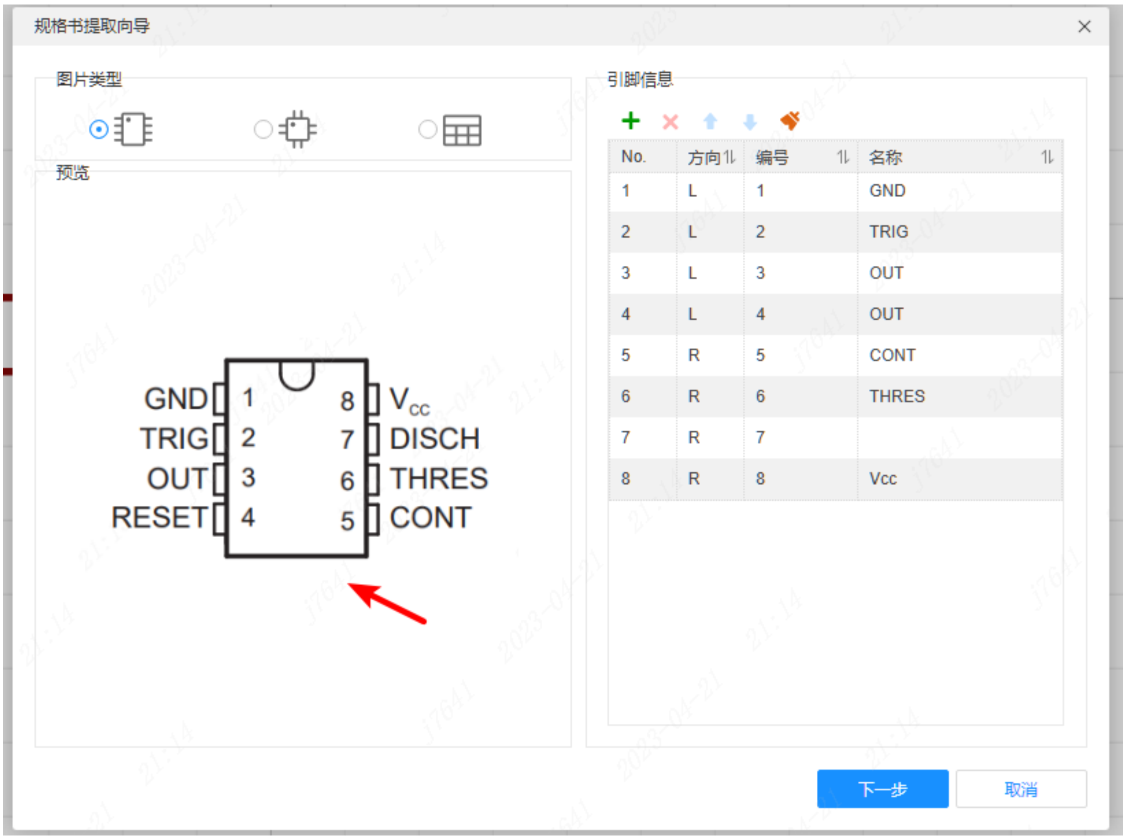


Picture type: This refers to the pin diagram type of the chip in the Datasheet, including SOP, QFN, and table types.
Preview: Paste the screenshot in the preview area, or double-click to open the local image or drag the image to the preview area to load the preview. After loading or pasting the picture, OCR will be automatically performed to identify the pin information in the picture.
Pin information: The server lists the identified pin information in this table, and supports manual modification of the name and order.

2. Open the Datasheet and take a screenshot of the chip pin area. Or save the screenshot locally. Be careful not to intercept non-essential text.

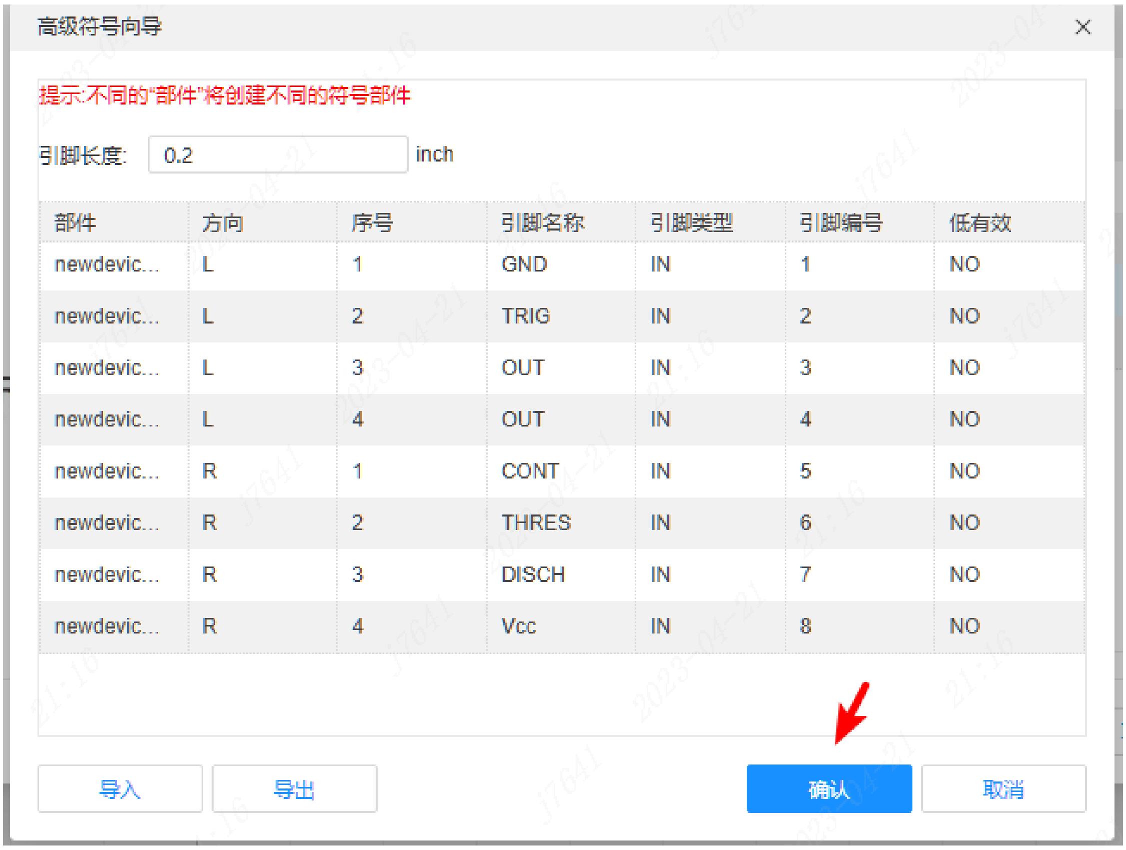


3. Go back to the Datasheet Extract Wizard dialog box, left-click the preview area, press CTRL+SHIFT+V to paste the picture, or double-click the preview area to open the local picture



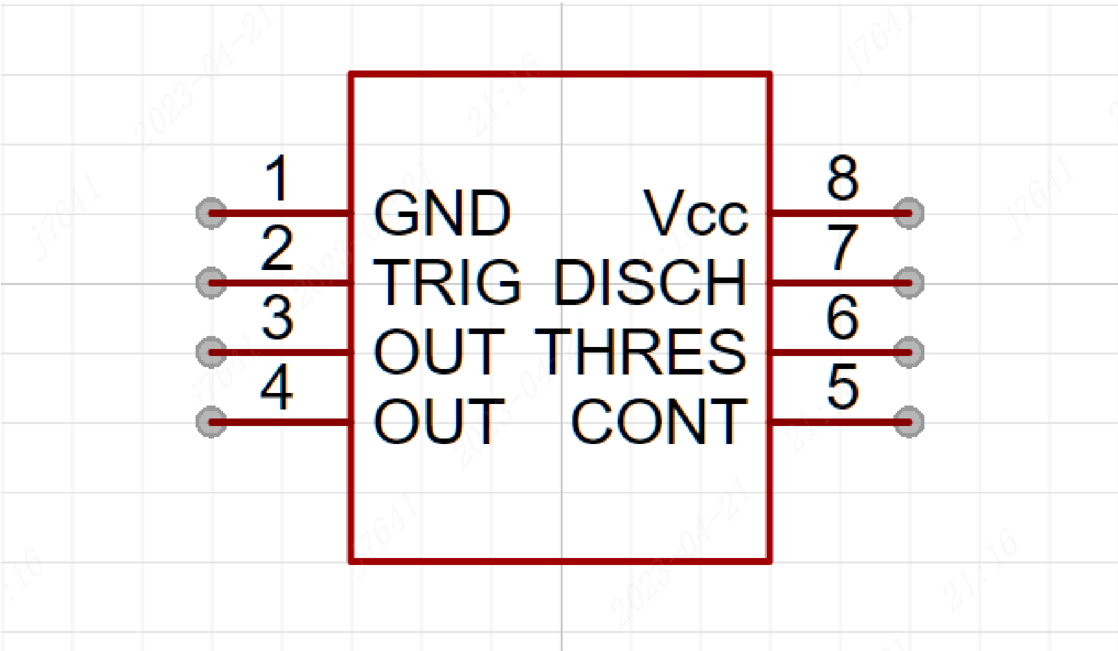
4. Check and modify the identified list in the pin information list, and click Next after checking.

5. The next step will open the advanced symbol wizard, where you can continue to edit more pin information



部件	方向	序号	引脚名称	引脚类型	引脚编号	低有效
newdevic...	L	1	GND	IN	1	NO
newdevic...	L	2	TRIG	IN	2	NO
newdevic...	L	3	OUT	IN	3	NO
newdevic...	L	4	OUT	IN	4	NO
newdevic...	R	1	CONT	IN	5	NO
newdevic...	R	2	THRES	IN	6	NO
newdevic...	R	3	DISCH	IN	7	NO
newdevic...	R	4	Vcc	IN	8	NO

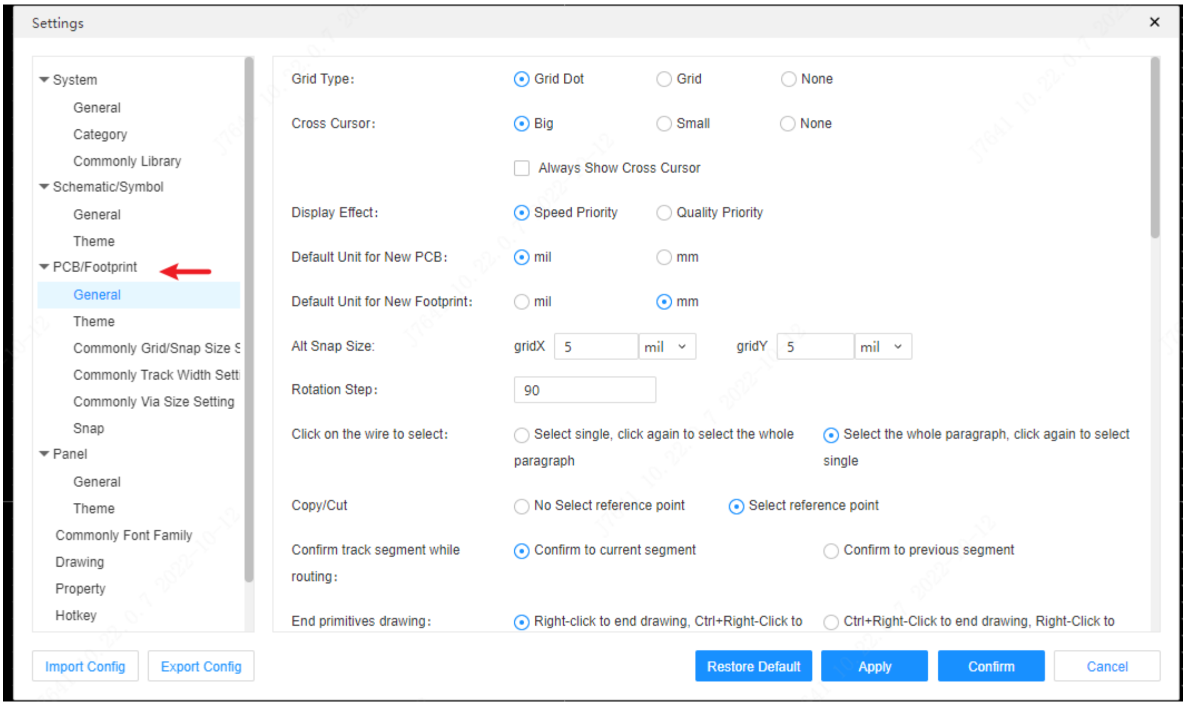
6. After clicking OK, the canvas generates the recognized component symbols.



footprint Settings

Before creating a new footprint, you can set some default values for the footprint.

Entry: Top Menu - Settings - footprint Settings.



Currently PCB settings and footprint settings are merged together because most of their settings are common.

The unit of the new footprint can be preset, etc.

Create a Footprint Library

The footprint library is the various parameters of electronic components, chips, etc. (such as the size of components, length and width, in-line, patch, pad size, length and width of pins, spacing between pins, etc.) in a graphical way. Manifested so that it can be called when drawing the PCB.

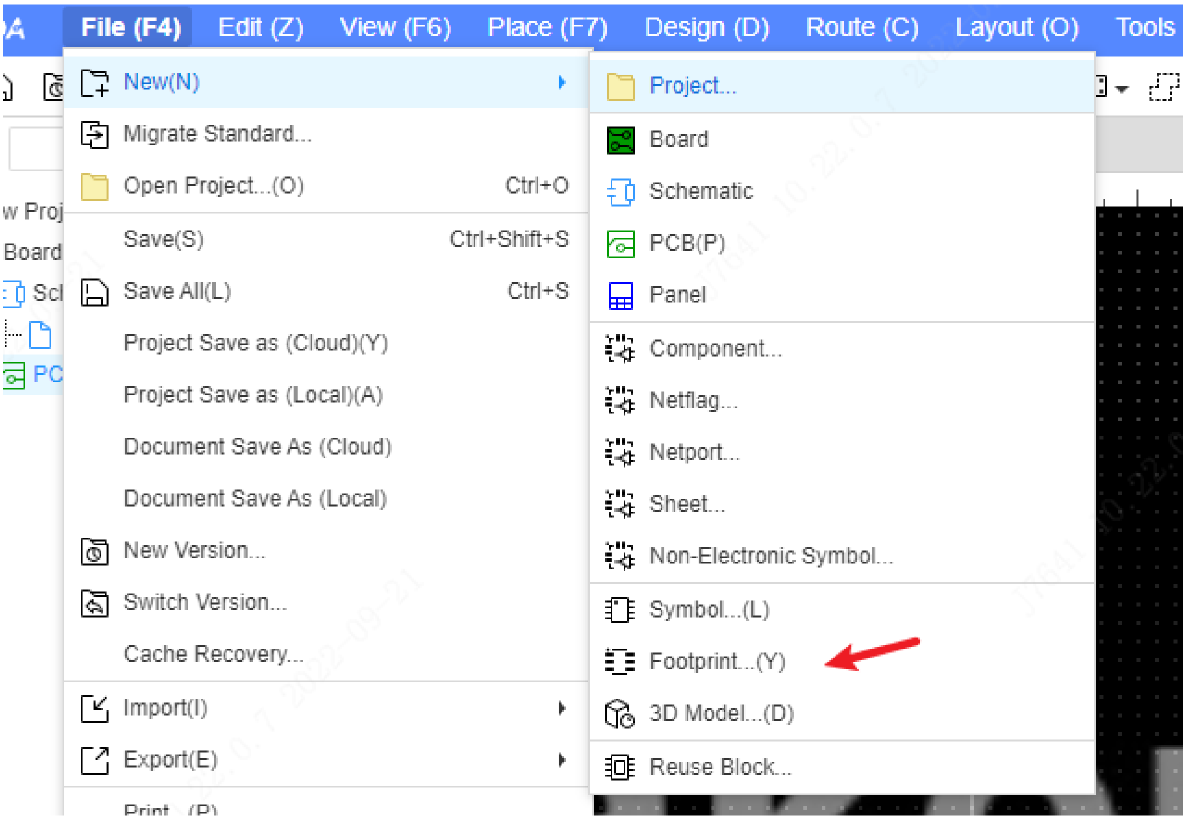
The usage method of the footprint library tool is the same as that of the PCB tool under PCB, but there are some unnecessary functions in the toolbar.

If you need a PCB footprint, but it does not exist in EasyEDA's component library, then you can create it yourself. And creating a device also requires a footprint library to complete.

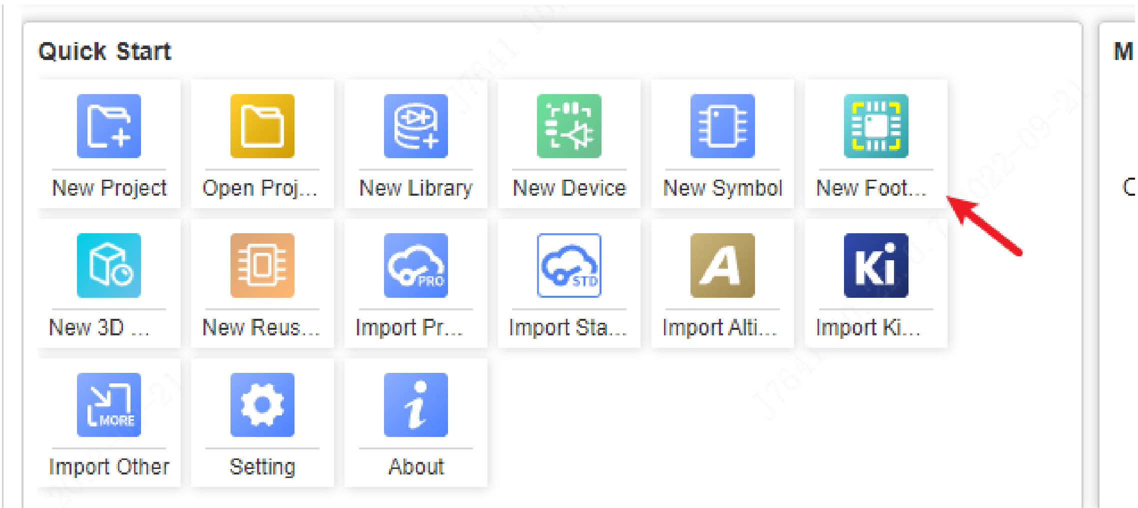
Creating a footprint library is almost identical to creating a symbol library.

New footprint steps:

- Top Menu - File - New - footprint

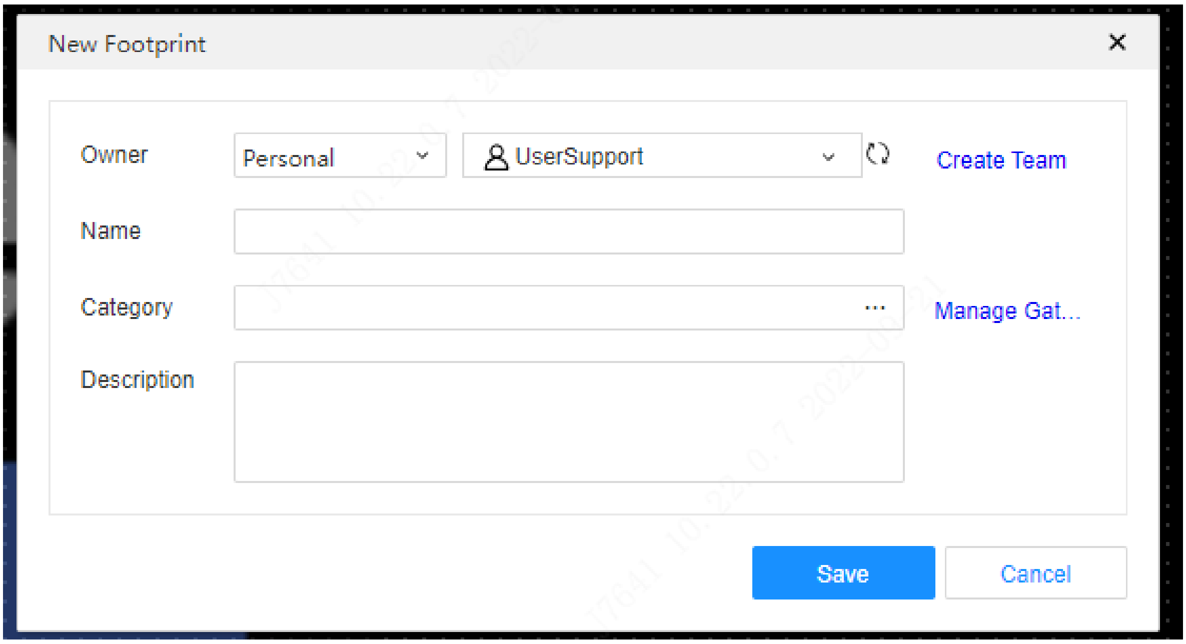


- Start Page - New Footprint



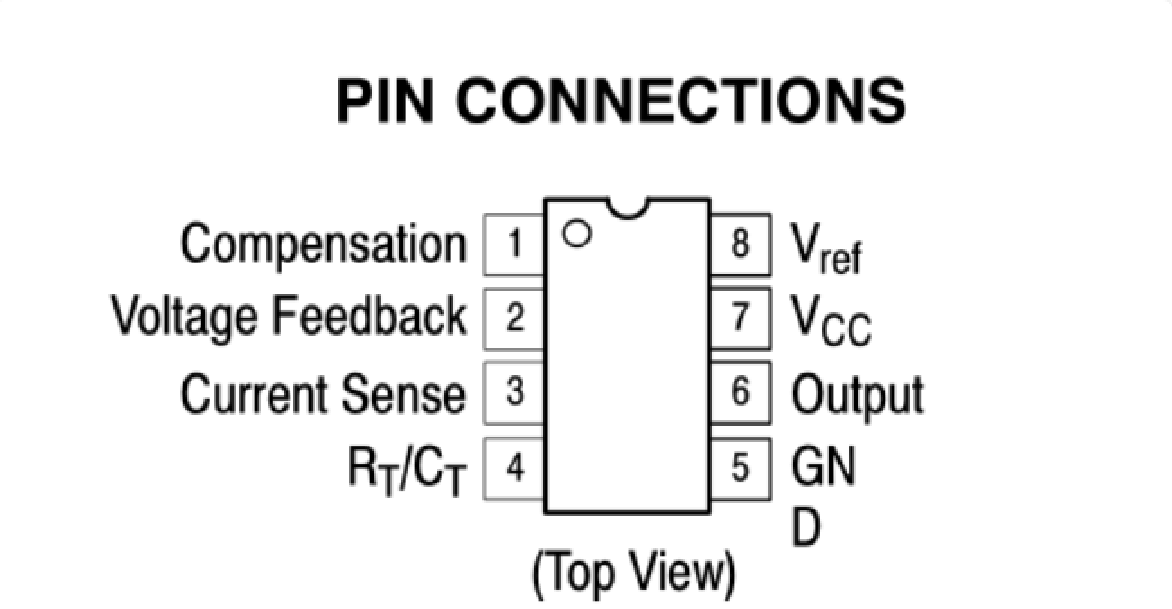
Name the footprint and choose a category.

For the footprint name naming suggestion, please refer to [EasyEDA Footprint Naming Rule Reference.pdf]
to use scientific naming rules to facilitate management and reuse.

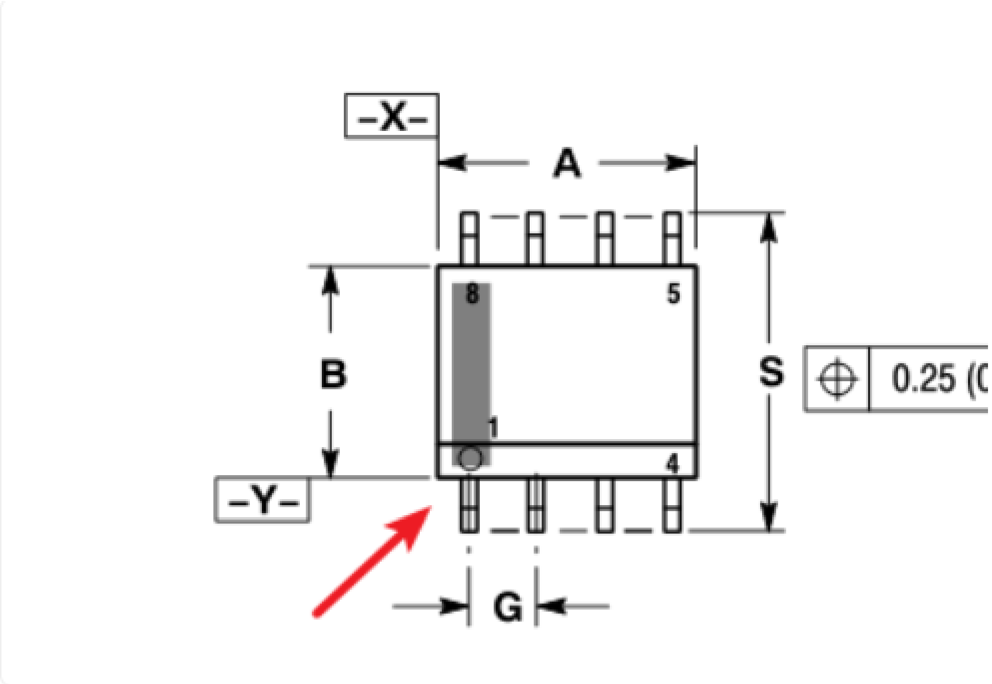


The corresponding footprint is drawn according to the device's data sheet.

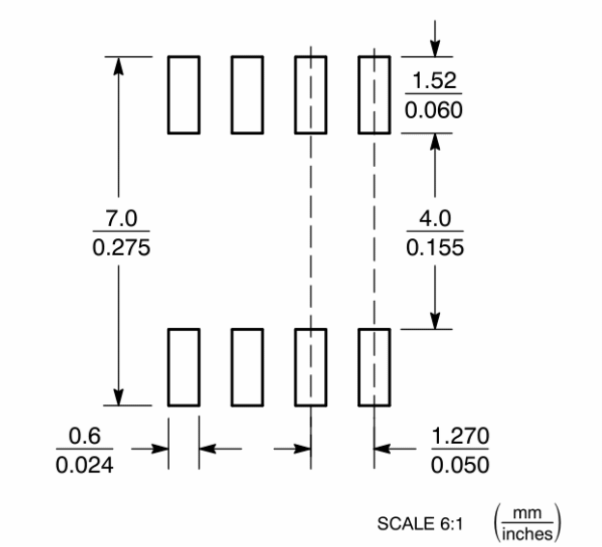
- 1. Download the specifications of the components to be drawn.
  Such as the creation of SOIC-8. [eg PDF: UC2844BD1R2G]
- 2. Read the specification to obtain the footprint size and orientation information.
  When drawing the footprint, you need to pay attention to the 0-degree direction of the footprint (0 degree is the angle you are currently drawing, when the footprint is placed in the PCB, its angle is displayed as 0 degrees in the property panel), which is good for SMT patches. For details, please check: [PCB footprint library 0 degree graphics production standard] (http://club.szlcsc.com/article/details_12157_1.html)
- 3. Check the footprint size, pin direction and polarity, and then place the pads on the canvas. You need to adjust the shape and size of the pad according to your actual situation.
  - Component pin orientation, page 1, counts 1 to 8 counterclockwise.

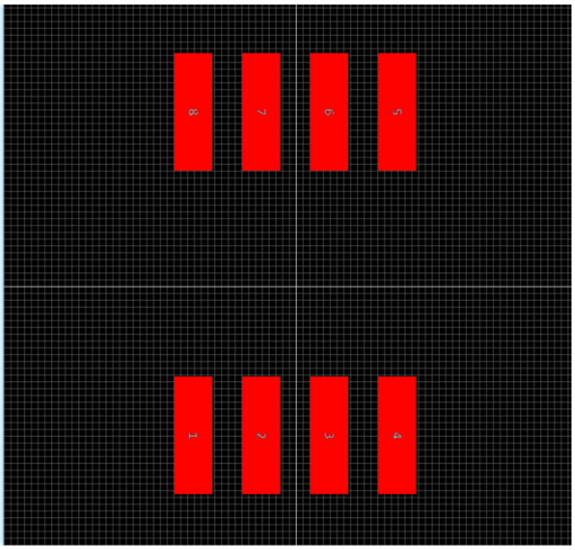


Component polarity, page 1, page 18, the orientation is horizontal.



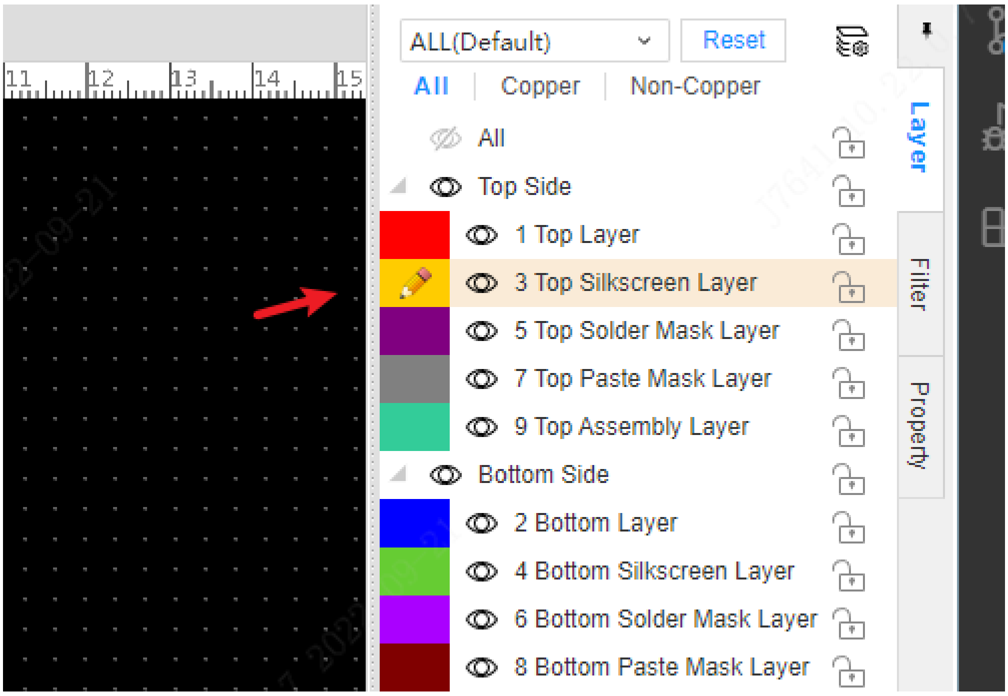
Place the pads on the top layer according to the dimension drawing on page 18 (some gold finger footprints require the top and bottom layers to be placed at the same time). Need to modify the properties of the pad, including pad number, type, size. First determine the coordinates of the first pad, and then place multiple, evenly distributed using the alignment tool on the top menu. If the moving pad step distance is not suitable, you can modify the grid size on the right.

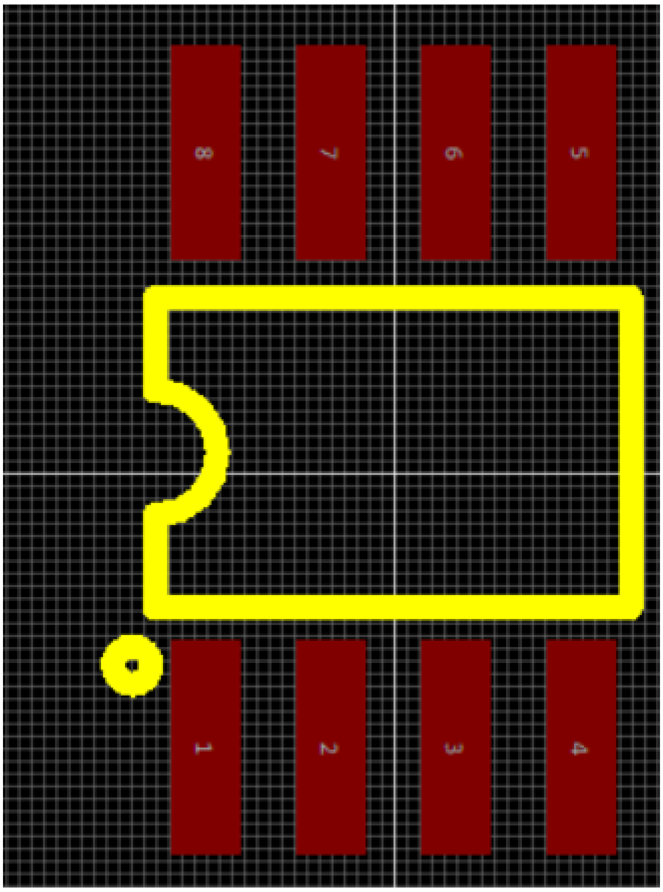




Draw component border silkscreen. Sometimes the corresponding logo graphics can be placed on the mechanical layer and the document layer.

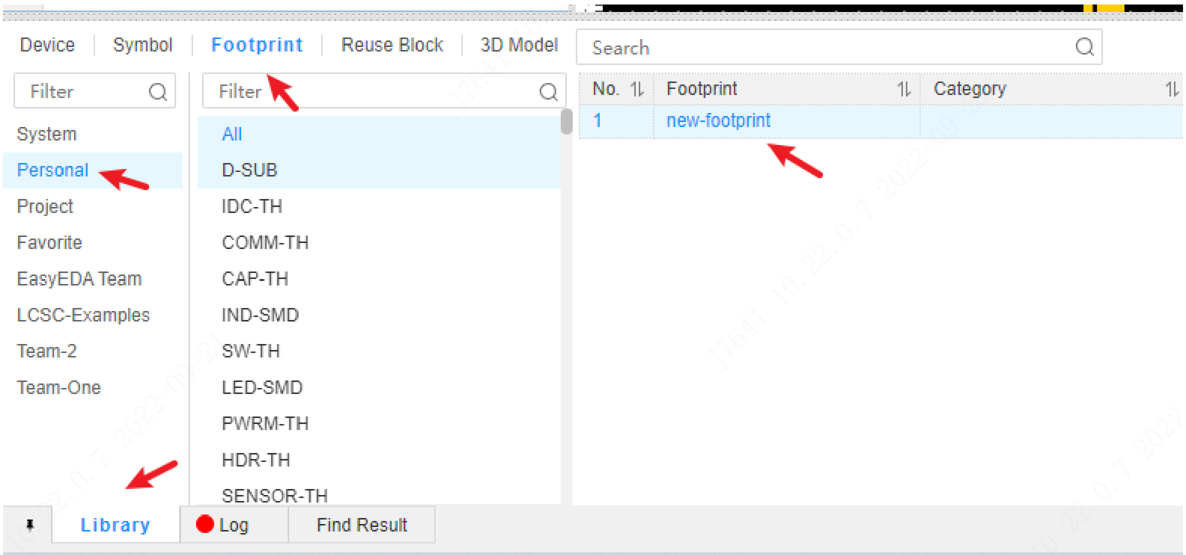
- Switch layer to top silkscreen.
- Use wire, arc and other tools to draw component silkscreen.





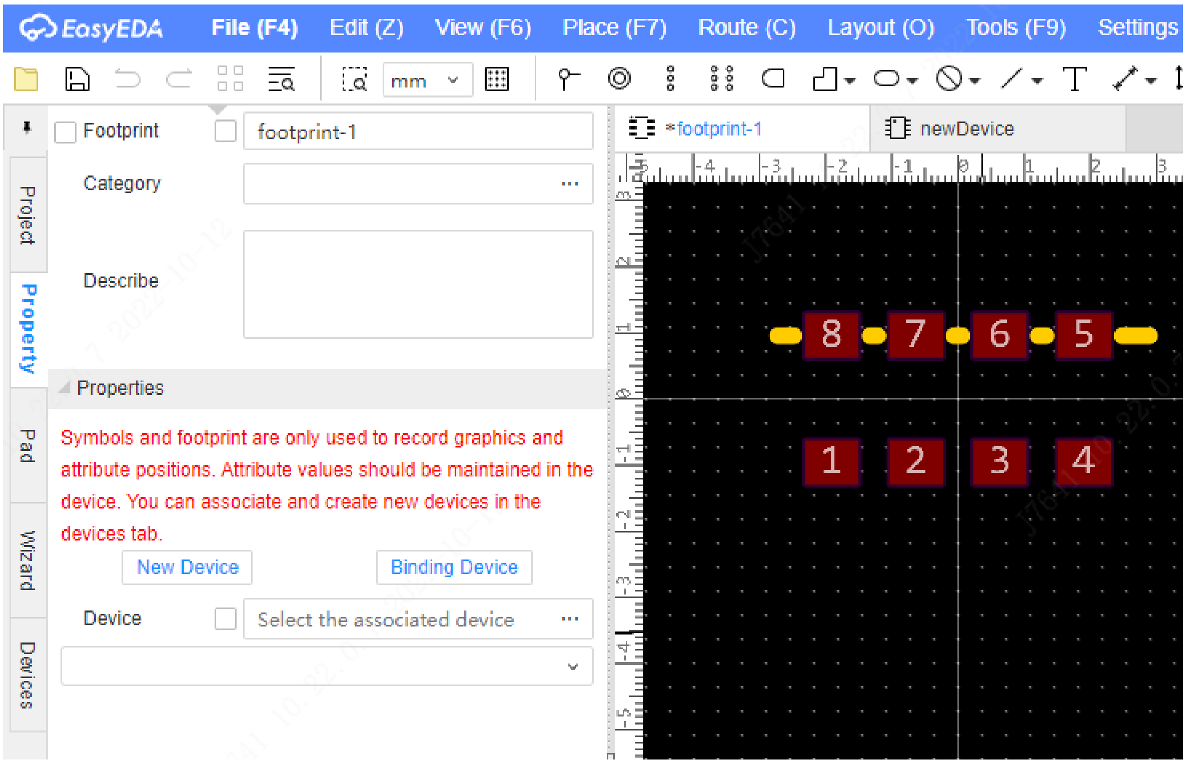
- At least one center point of the placed pad should be on the grid line as much as possible, so as to avoid the difficulty of connecting the wire to the pad when using the footprint.
- Numbers and letters can be set for the pad number, which must correspond to the pin number of the component one-to-one, otherwise the component will report an error in the footprint manager in the schematic diagram after this footprint is specified, and cannot be converted to PCB.
- Pad numbers can be incremented by mouse Place, and will remain the same if copied by pasting.
- After drawing the footprint, it is recommended to modify the coordinate origin of the footprint to the center of the footprint: it is convenient to keep the mouse in the center of the graphic when rotating; the SMT machine reduces the component adjustment work when recognizing the center of the component.

after finish, you can find it at: Bottom Panel - Library - Footprint - Personal



Footprint Properties

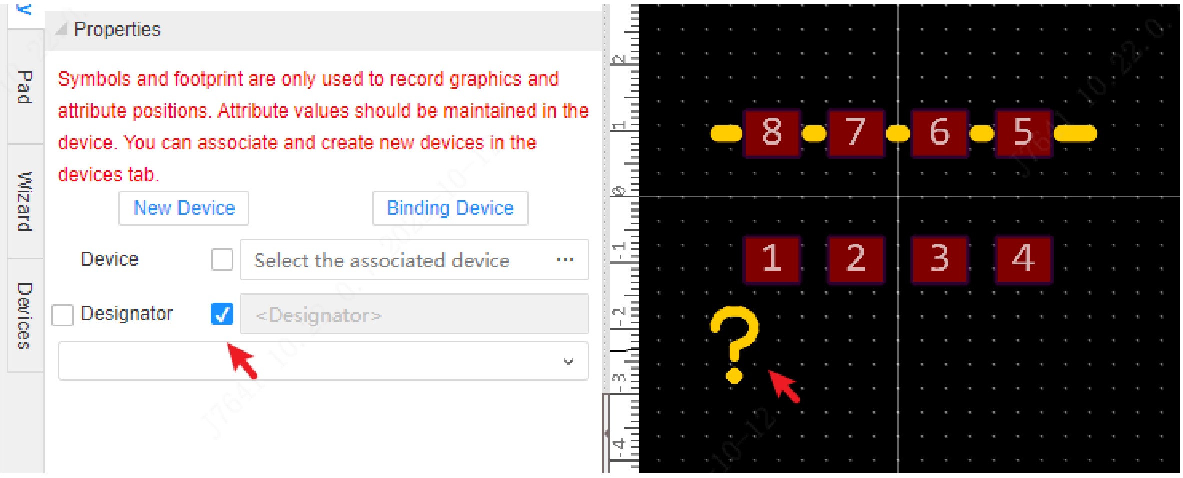
The right panel of the footprint interface is to display the attributes, tag numbers, category and other information of the entire footprint.



Footprint: Used to name the footprint.

Designator

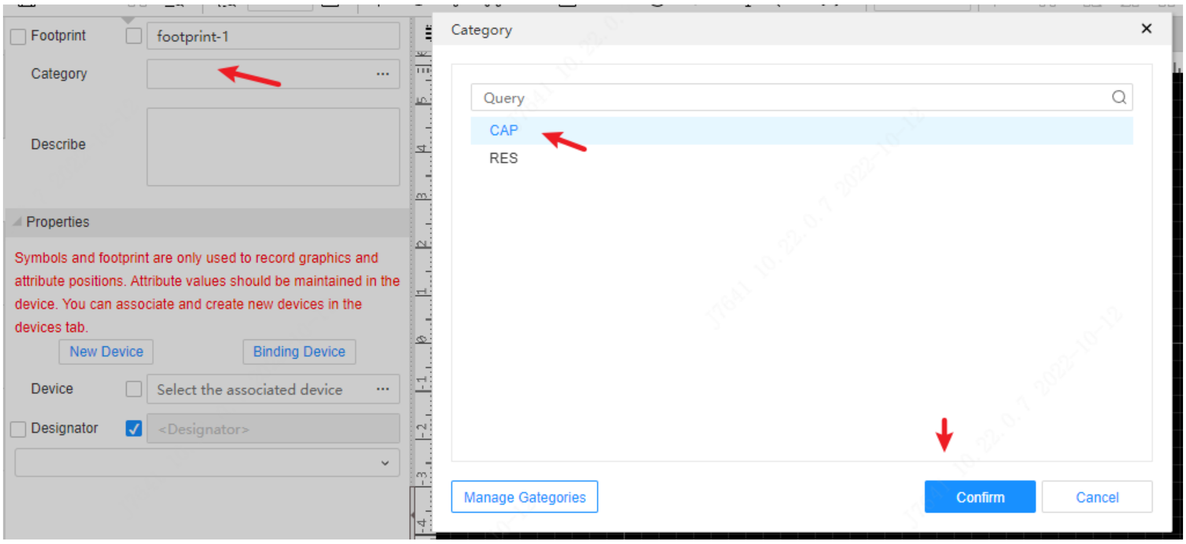
Set the designator for the footprint. After ticking the checkbox next to it, the designator can be displayed in the footprint. when convert schematic to PCB, the designator location will use footprint's designator location first



category

The category of footprints can be modified here to facilitate later management and maintenance.

Click the input box of the category with the mouse, and a pop-up window will pop up, where you can modify the category.

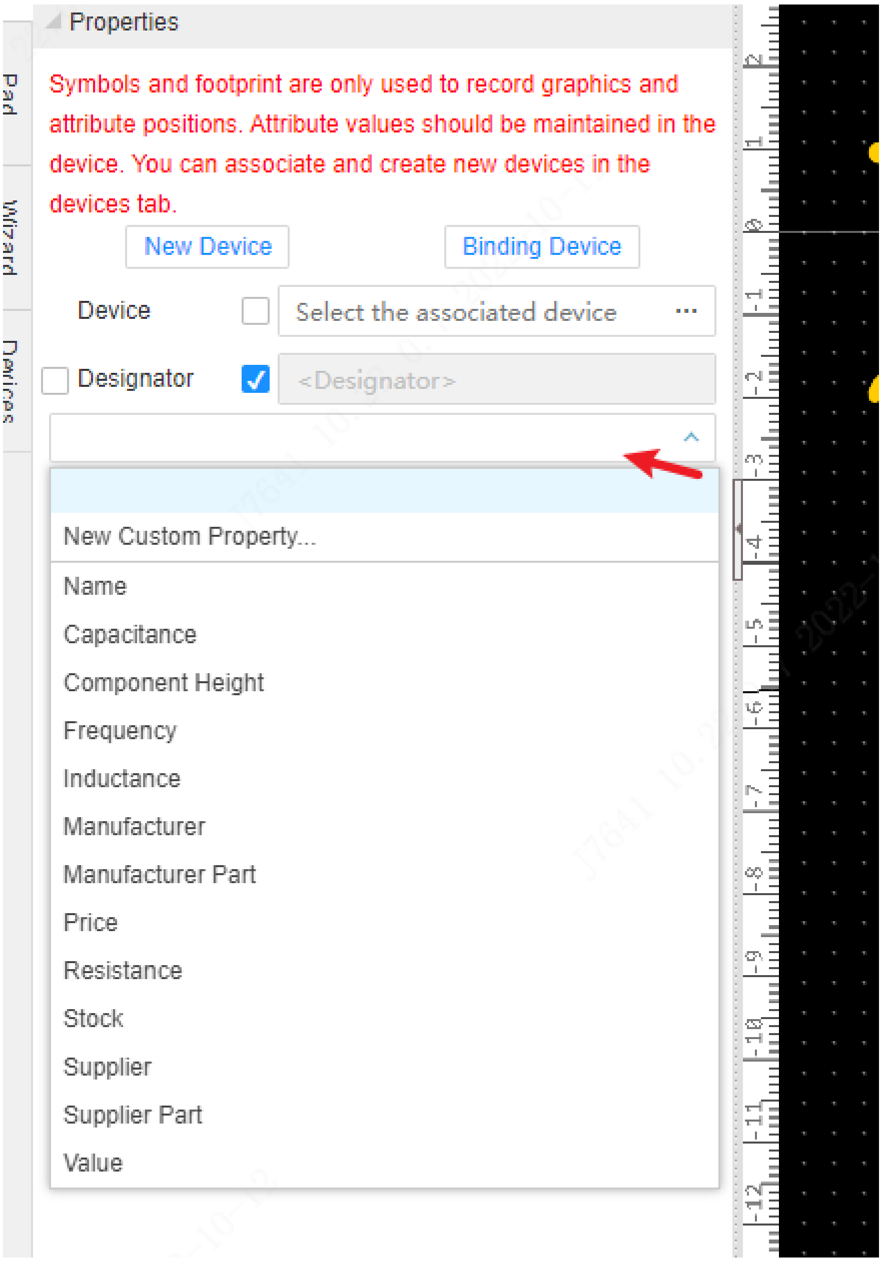


describe

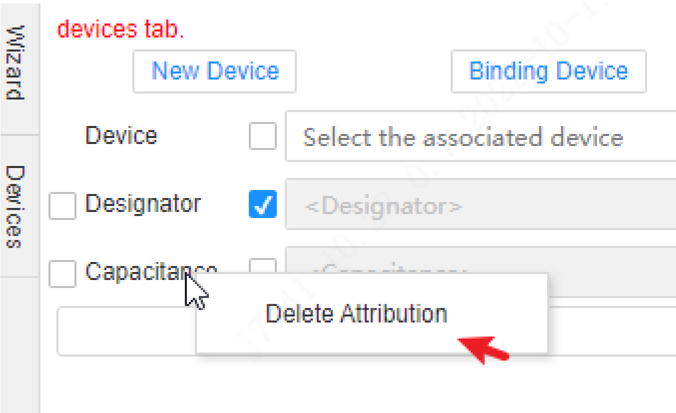
Used to add some textual description to the footprint.

Add Properties

You can add properties into footprint to set the default property location, but these properties only save the location, the content will use device's properties.

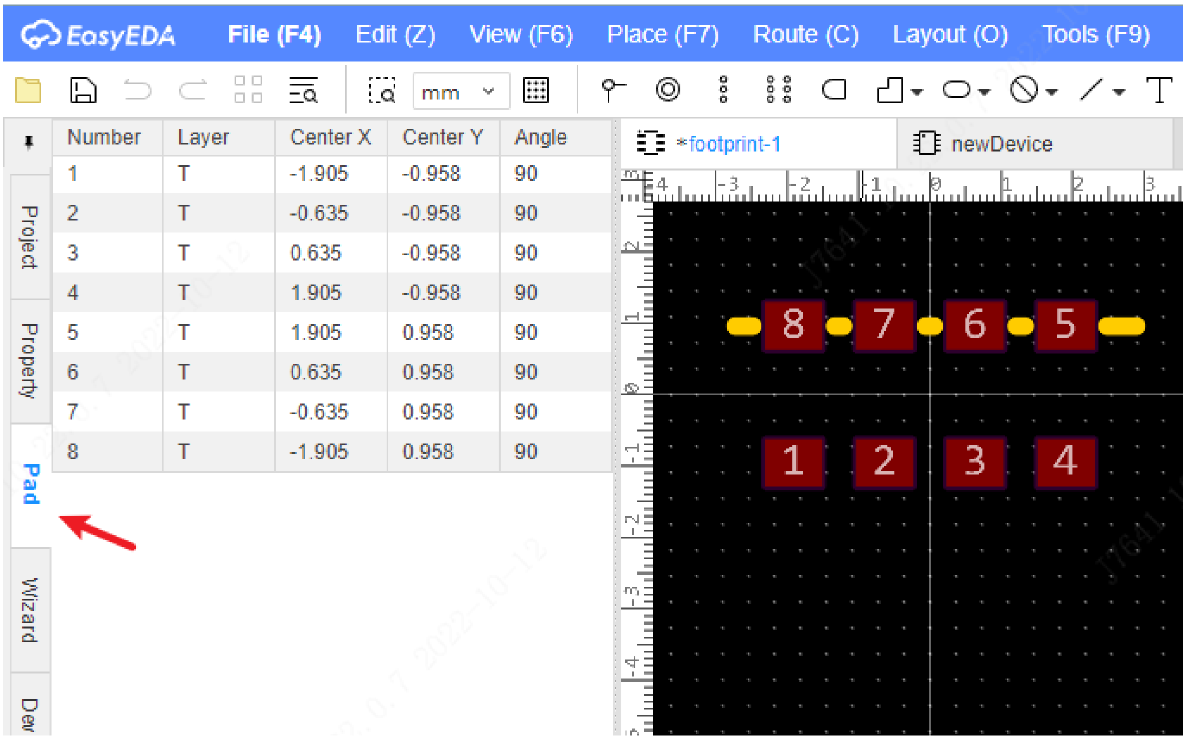


Hover the mouse in the middle of the blank space for adding properties, and click `right mouse button` to delete the added properties. The system default properties cannot be deleted.



Pad Editor

The footprint pad editor is to edit and view all the pads in the footprint drawing page. Double-click the list to modify the properties of the pad.

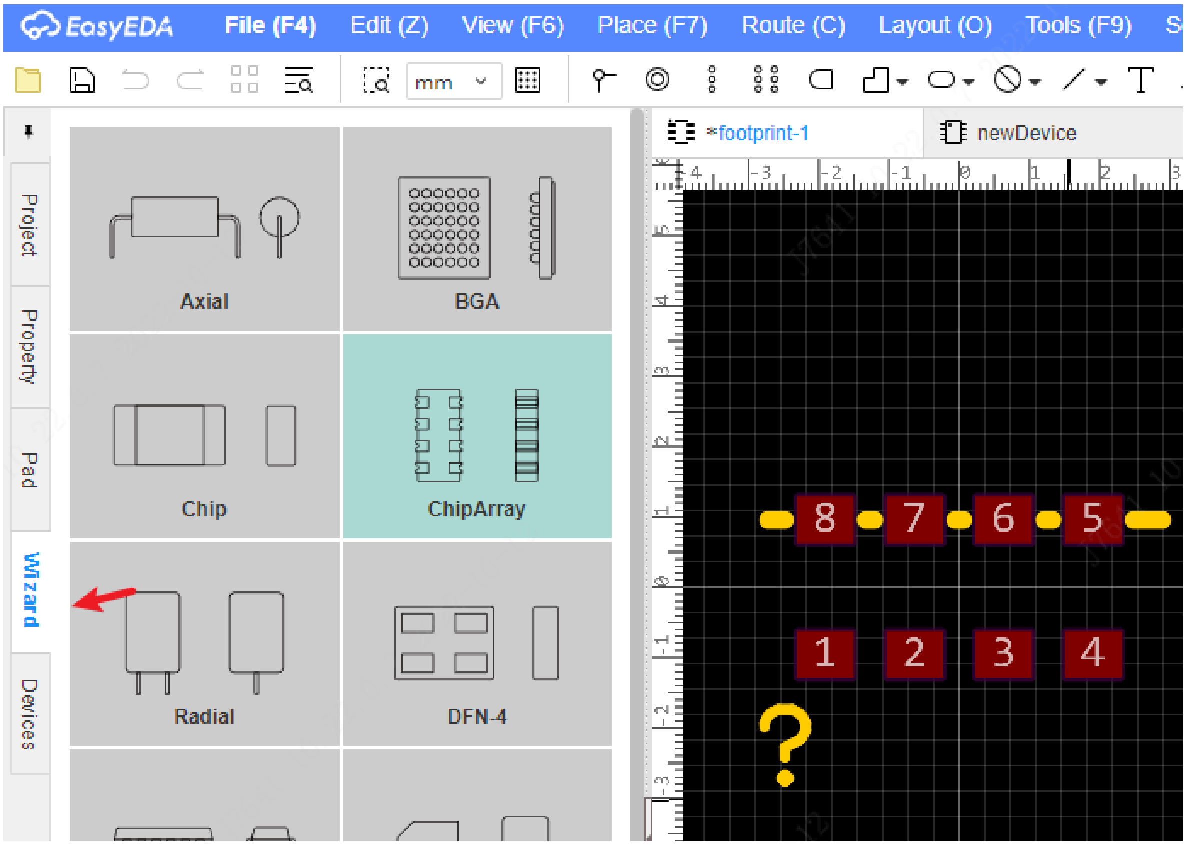


- **Attribute**: Indicates the number of the pad
- **Layer**: T is the pad for the top layer, B is the pad for the bottom layer, M is the pad for the through-hole type.
- **Center X**: Indicates the X coordinate value of the pad on the footprint page.
- **Center Y**: Indicates the Y coordinate value of the pad on the footprint page.
- **Angle**: You can set the angle of the pad.

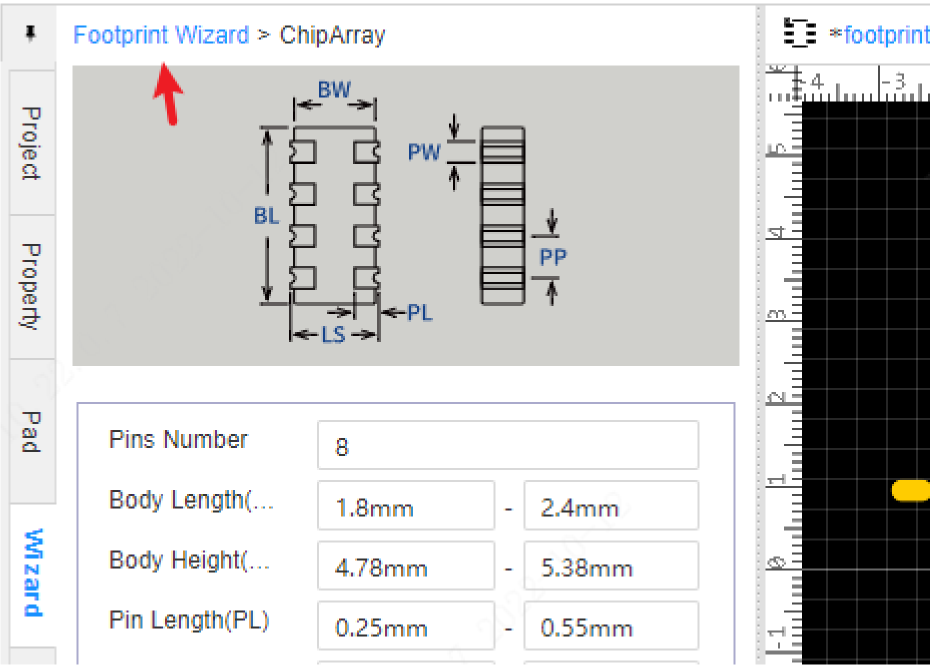
Footprint Wizard

EasyEDA provides a common footprint and powerful creation wizard. Using the footprint wizard, you can quickly create a footprint according to the specification.

Entrance: Left Panel - Wizard

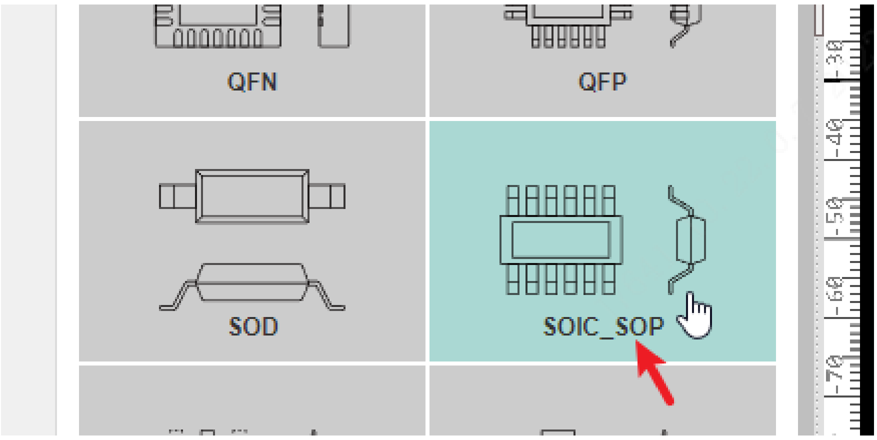


Click the footprint type thumbnail to enter the parameter filling interface, and click the preview image and the top navigation on the parameter filling interface to return to the previous level.



Steps for usage:

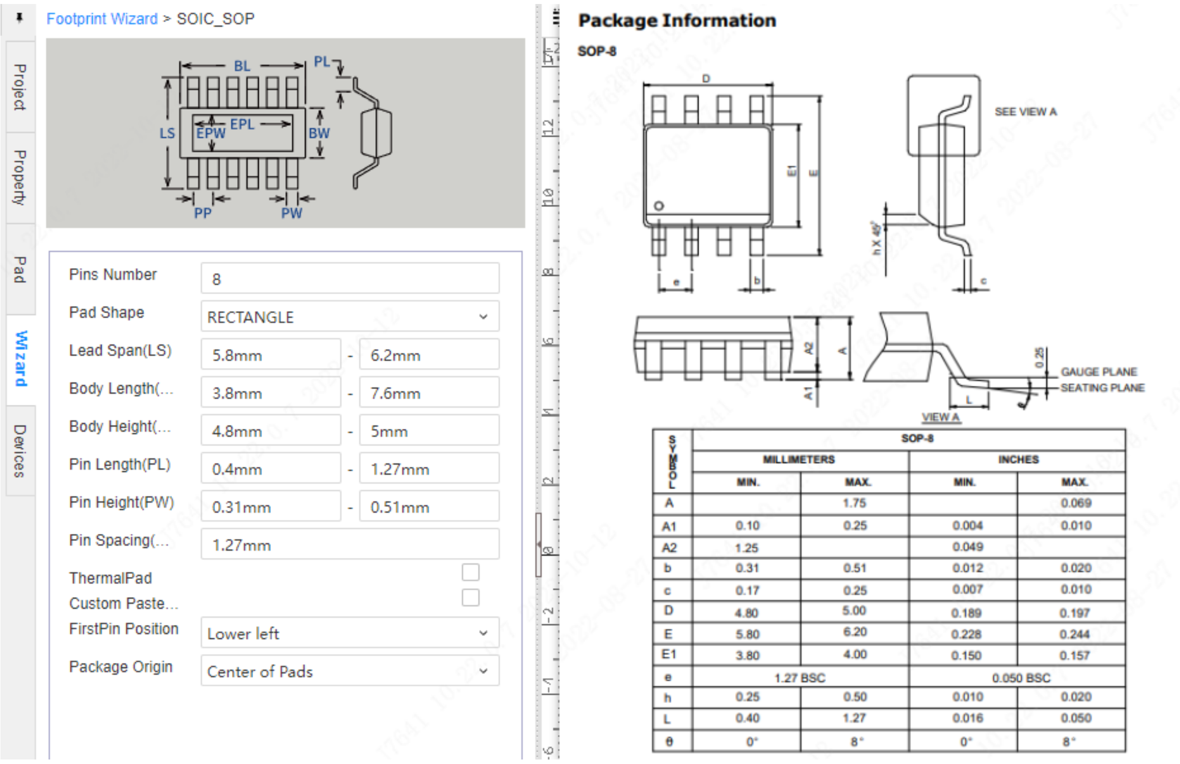
1. Select the footprint type to be created. Click on the diagram to enter. For example, select SOIC_SOP type footprint:



2. Set the physical size of the footprint according to the specification. It involves the shape length BL, shape width BW, pin span LS, pin spacing PP, pin size PW, if there is a heat dissipation pad, there is a heat dissipation pad length EPL, and the pad width EPW.

For example, you need to create a SOP-8 footprint, the reference address of the specification: [C368696 APW7172]()

According to the specification above, the physical dimensions of the footprint are as follows, in mm:

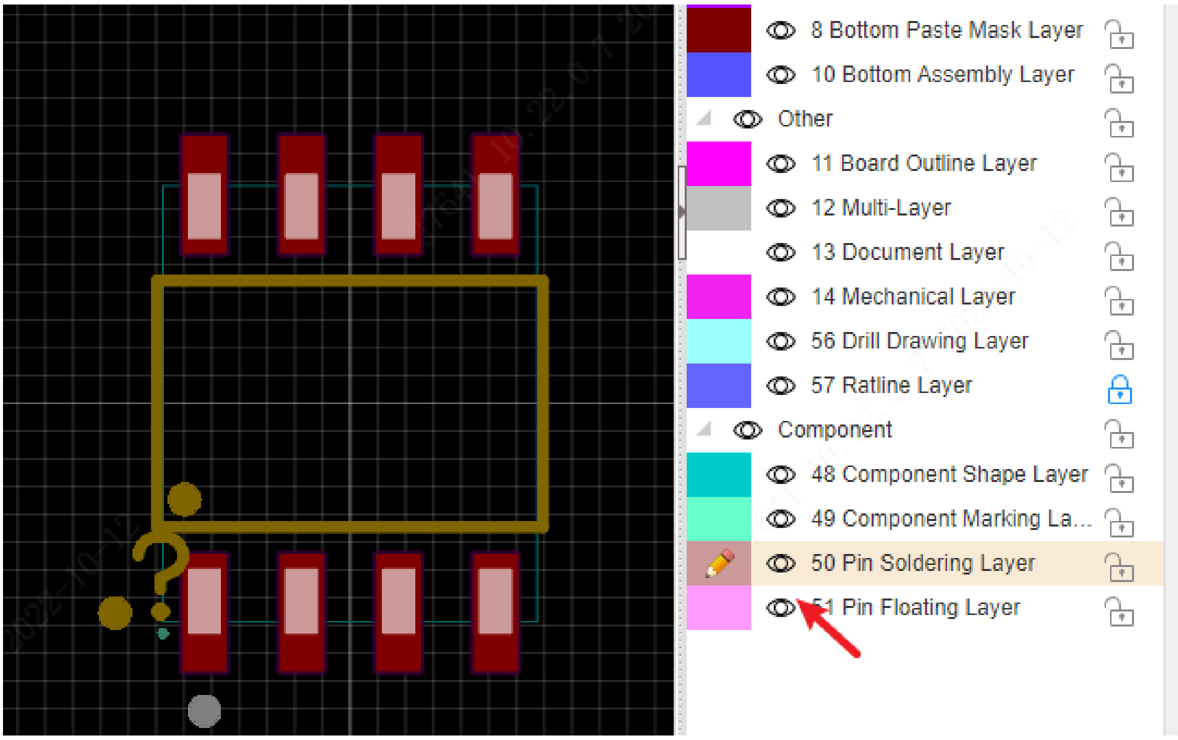


```
Number of pins: 8
Pad Shape: Rectangle
Pin Span (LS): 5.80 ~ 6.20. Corresponding to the minimum and maximum value of
dimension E
Body length (BW): 3.80 ~ 4.00. Minimum and maximum values ••corresponding to
dimension E1
Body width (BL): 4.80 ~ 5.00. Minimum and maximum values ••corresponding to
dimension D
Pin length (PL): 0.40 ~ 1.27. Corresponding to the minimum and maximum value of
dimension L
Pin width (PW): 0.31 ~ 0.51. The minimum and maximum values ••corresponding to
dimension b
Pin pitch (PP): 1.27. The minimum and maximum values ••corresponding to
dimension b
```

If you need to customize the thermal pad and the custom solder paste layer, you can check and set the corresponding parameters. The first pin location is usually seated, and the footprint origin is usually in the center of the pad.

3. Click the Generate footprint button

After clicking the Generate footprint button below, the canvas will generate the corresponding footprint content. Open the component-related layer to see the generated pin welding size and component size



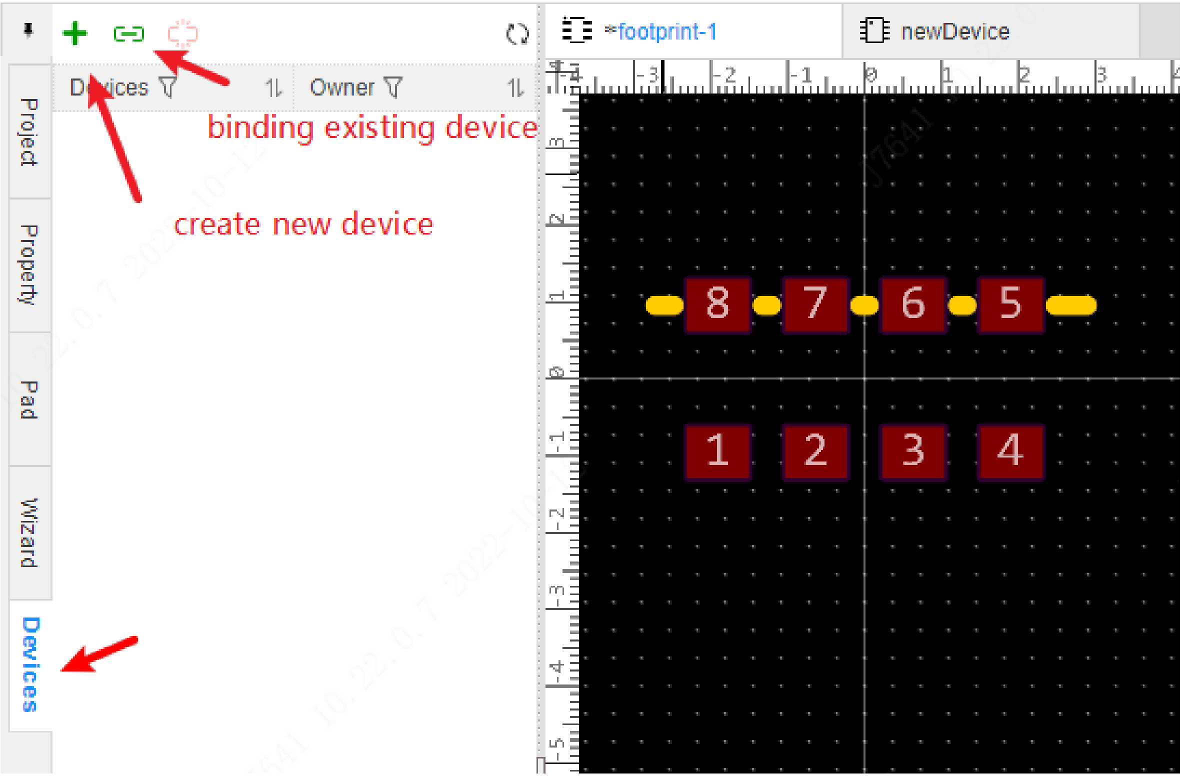
Notice:

- The wizard fills in the physical size of the footprint, not the size of the footprint pad. The wizard will automatically reserve the margin according to the filled parameters to generate the pad.
- The parameters and generated dimensions provided by the footprint wizard are for reference only. Please adjust the generated footprint dimensions according to the recommended values in the specification and relevant information of actual production.

Device Association

Devices require associated footprints and symbols before they can be placed on a schematic or PCB.

The device tab page on the right panel of the footprint interface shows which devices this footprint is associated with, which is convenient for new devices to associate with the current footprint, and it is also convenient to directly remove the binding relationship of this footprint in the existing bound devices.



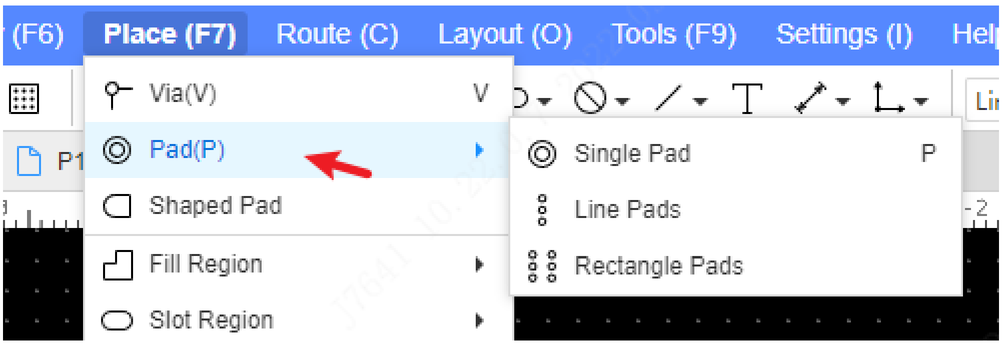
Pad

The pad is the basic unit of surface mount assembly, which is used to form the land pattern of the circuit board, that is, various combinations of pads designed for special component types.

Place pad steps:

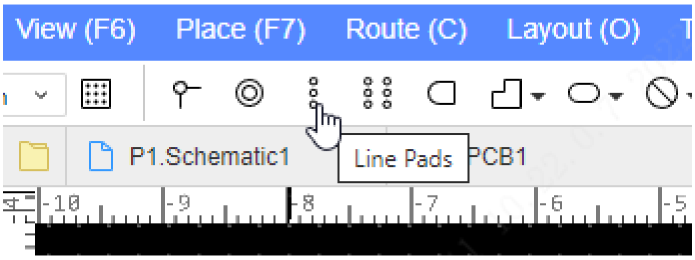
- Place using the shortcut `P` or the shortcut toolbar at the top.

When a pad is selected, its corresponding properties can be modified in the property panel on the right, or double-click with the left mouse button to pop up the properties dialog box to modify its properties.



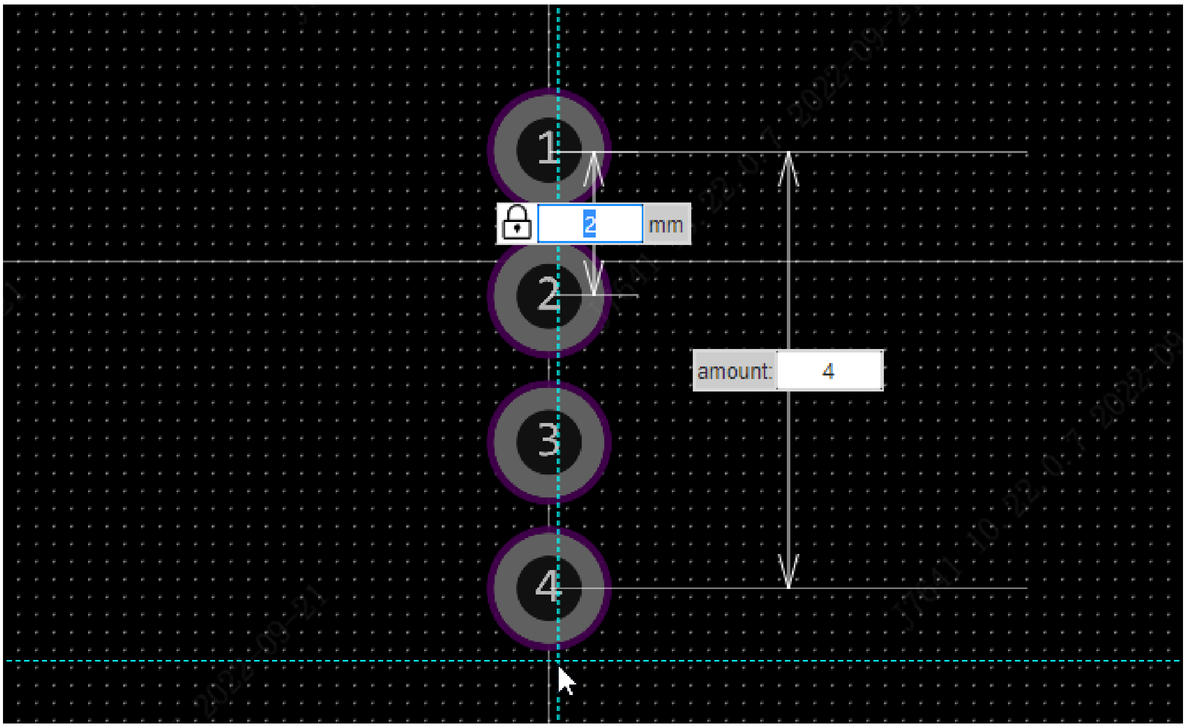
Line pads

Pad arrays are generated from the drawn bars, and pad arrays can also be generated from the values entered in the dynamic input box.



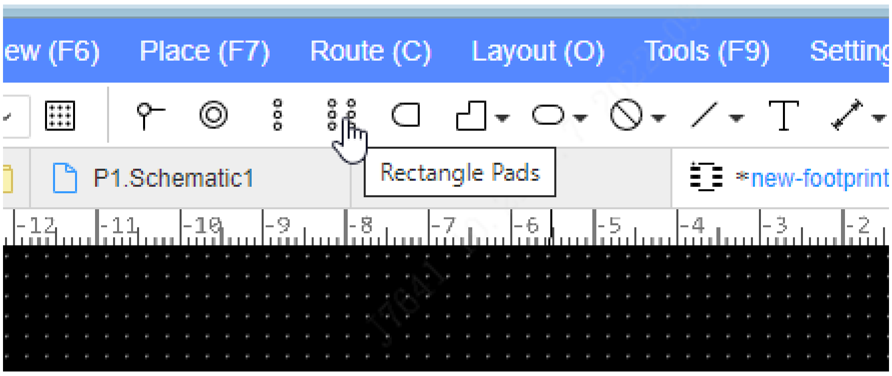
illustrate:

- Pad arrays can only be generated by drawing straight lines, not polygons.
- Does not support generating pads in multiple directions at the same time, the market is generated in one direction at a time.

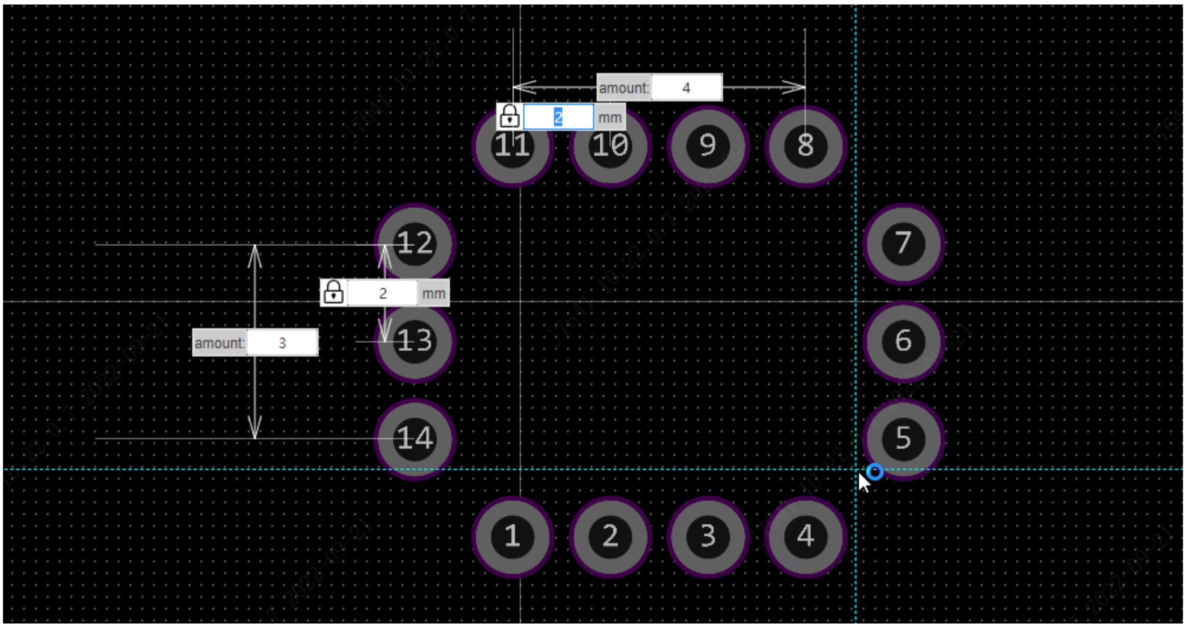


Press Tab key to change the input filed to change the parameters.

Rectangular Multipad

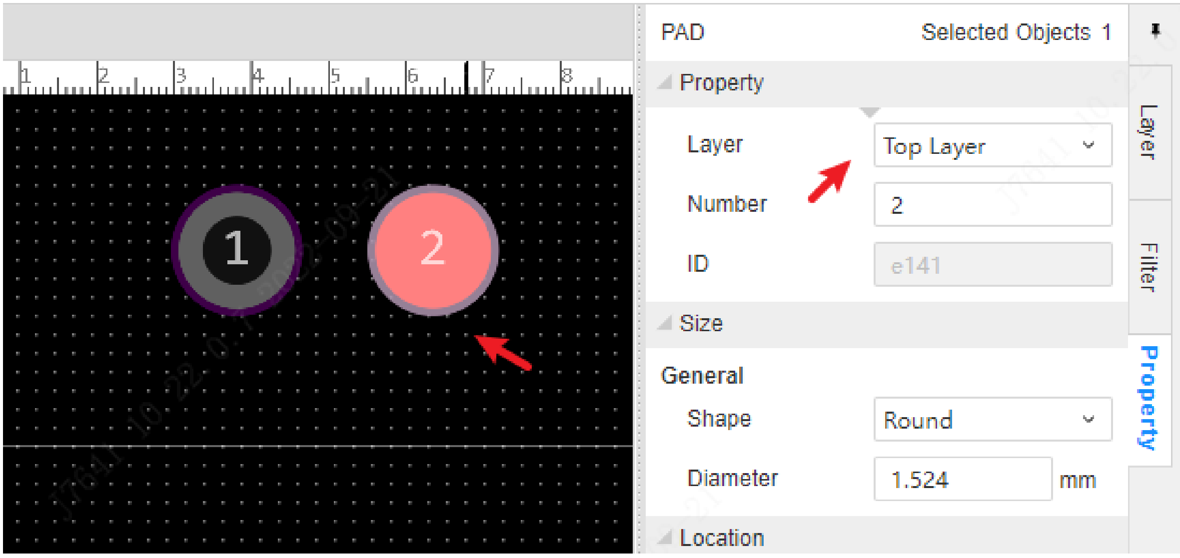


Generate a pad array based on the drawn rectangle, or you can generate a pad array by entering the value in the dynamic input box.



Modify the SMD pads

Select the pad and set the layer to the top or bottom layer in the property panel on the right to modify the pad into a SMD pad.

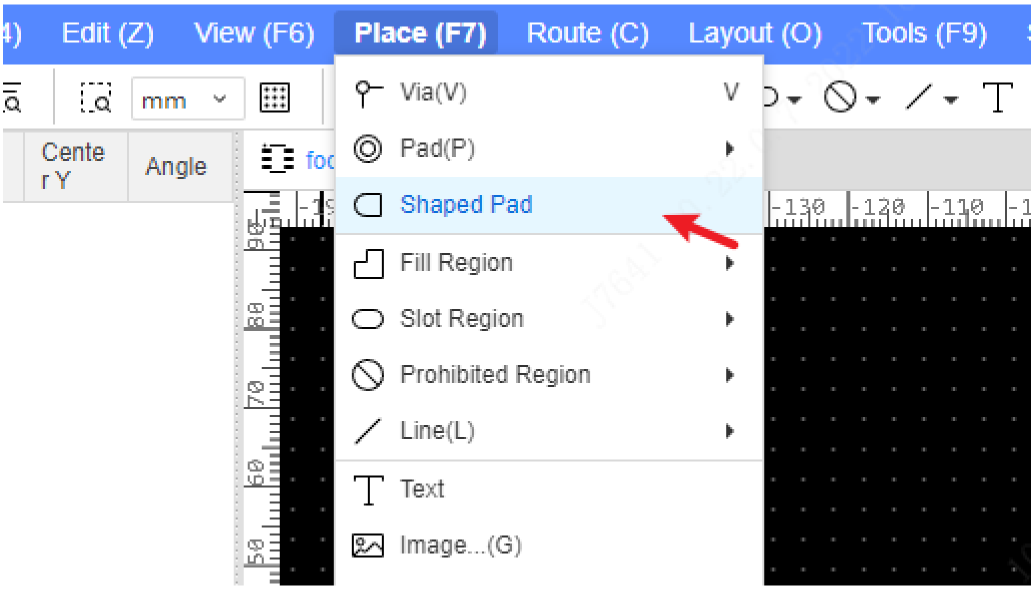


Shaped Pad

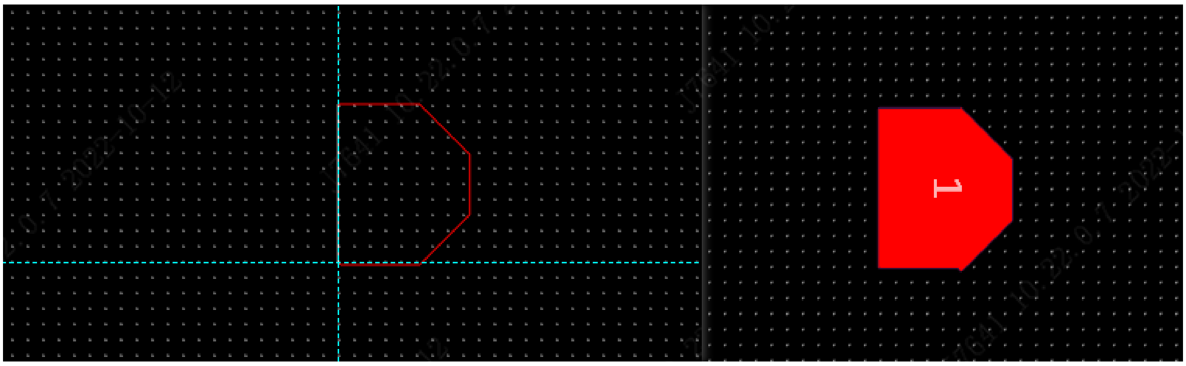
Generally irregular pads are called special-shaped pads, typically gold fingers, large device pads or copper foils of special shapes that need to be added to the board.

Place Shaped Pad Steps:

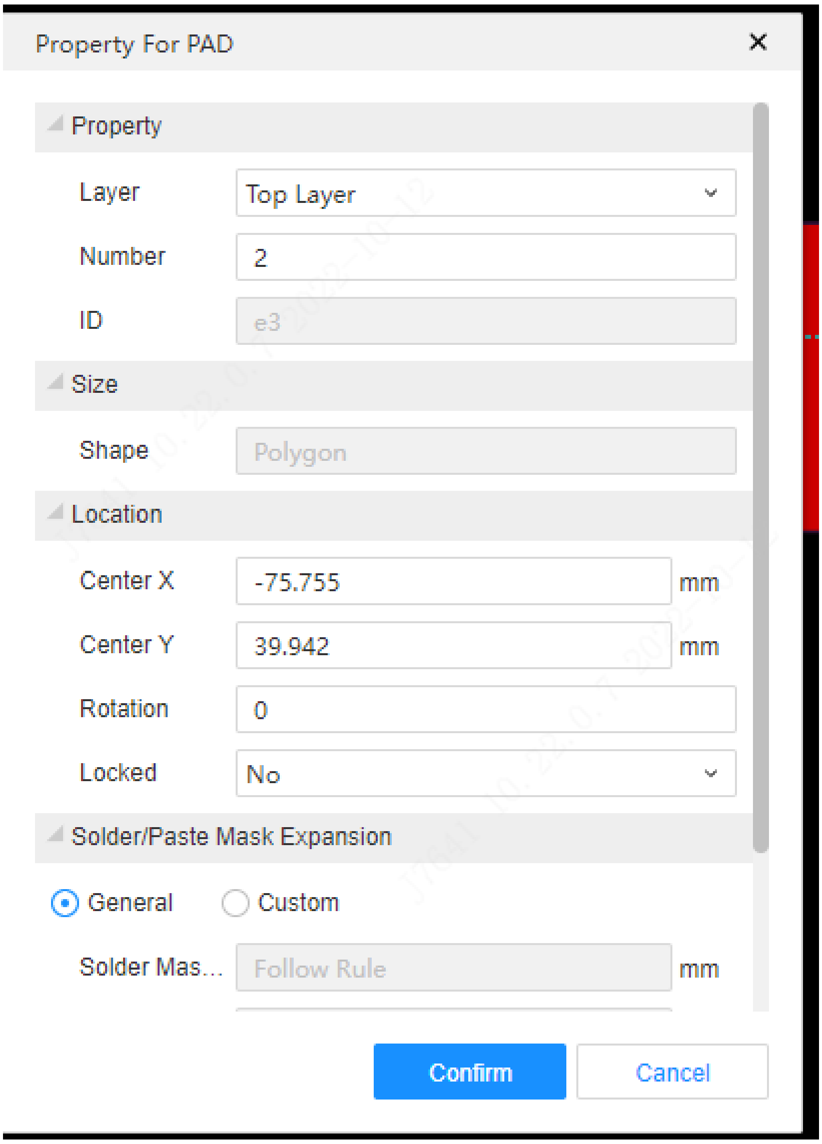
- Top Menu - Place - Shaped Pad or select Shaped Pad in the top shortcut bar



When the cursor turns into a cross state, you can start to draw the special-shaped pad likes draw a polygon fill region. You can change routing corner at Route menu or hotkey L

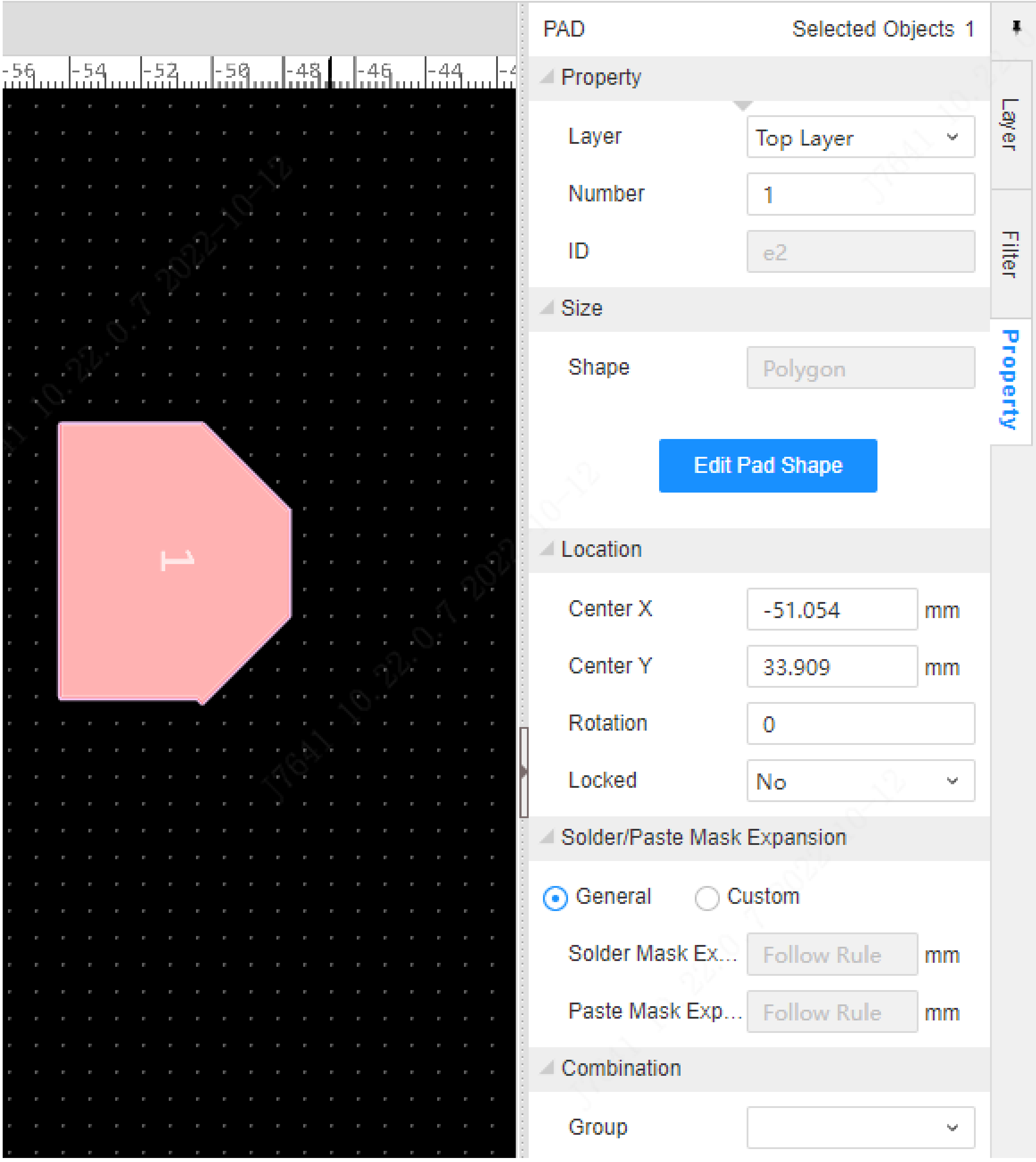


After drawing the special-shaped pad, the attribute shape pop-up window will pop up, and set the attribute shape of the special-shaped pad.



- **Layer**: Shaped pads can be set as top and bottom layers or multiple layers.
- **Number**: The number of the special-shaped pad, which cannot be repeated.
- **ID**: System default and cannot be modified.
- **Shape**: The default special-shaped pad, which cannot be modified.
- **CenterX**: The X-axis position where the pad is drawn.
- **Center Y**: The Y-axis position where the pad is drawn.
- **Angle**: The angle setting of the special-shaped pad, the default is 0.
- **Lock**: Lock the special-shaped pad and cannot move it.
- **Solder Mask Extension**: Follow the system rules by default.
- **Paste Mask Extension**: Follow system rules by default.

Click the created special-shaped pad to modify the attribute shape in the left panel.

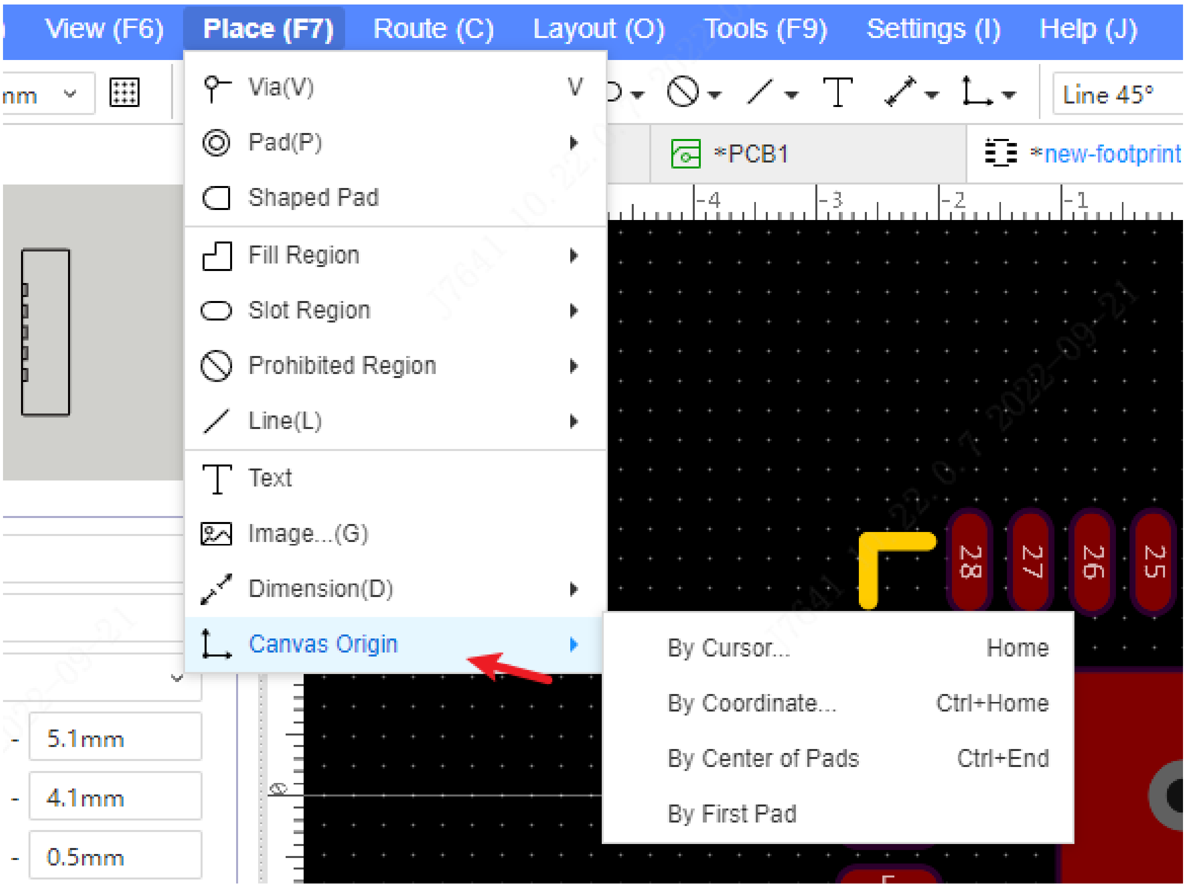


Canvas Origin

Set the origin position of the drawn footprint

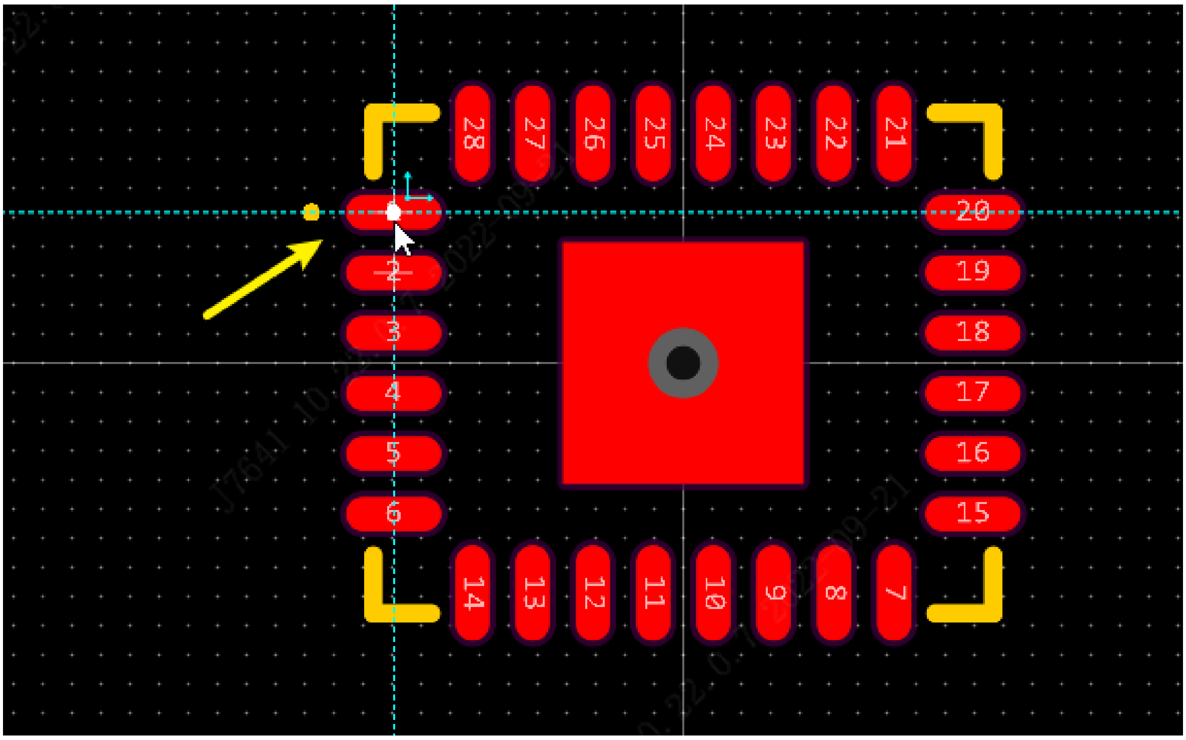
Place the cursor origin step:

- Top Menu - Place - Canvas Origin - From Cursor



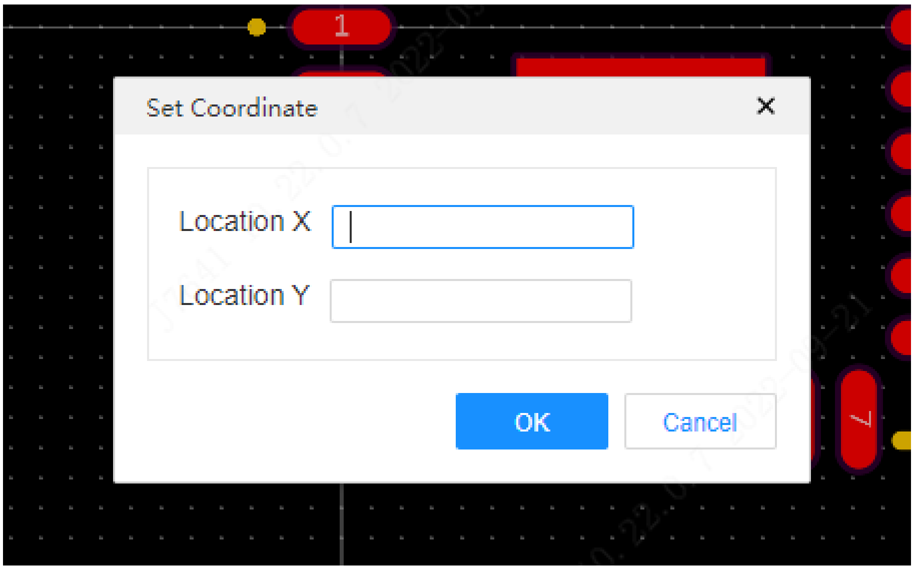
By Cursor

After selecting, you can use the mouse to select the origin position. After selecting the post-position, the coordinate value there will become zero.



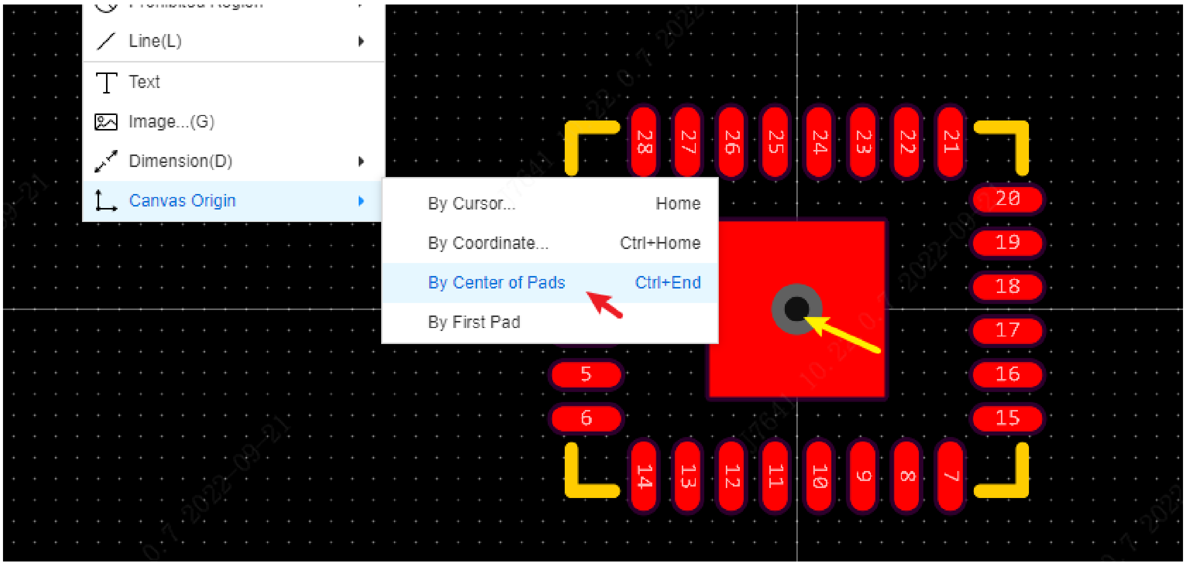
By Coordinate Point

After clicking, a dialog box will pop up, enter the coordinate value you want to set as the origin in it, and you can adjust the coordinate value you entered as the origin.



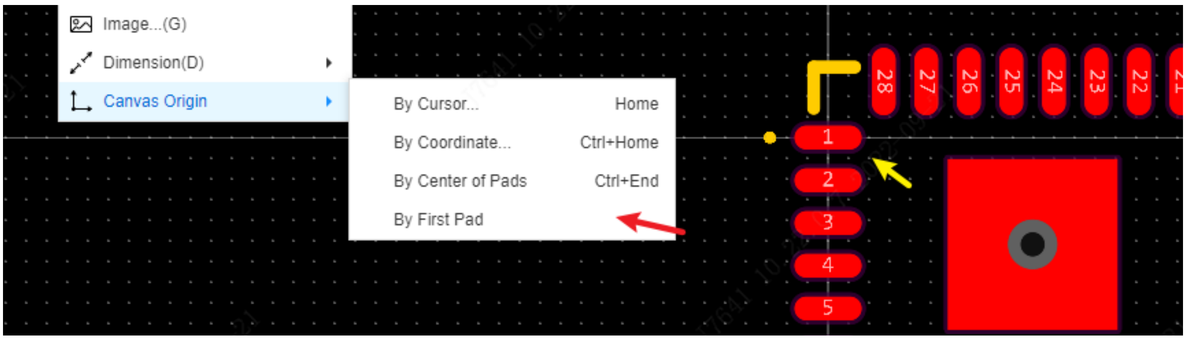
By the Center of the Pads

After clicking, the system will select the middle of the pads on both sides as the origin.



By the First Pad

Set the origin to the first pin(pad number is 1 or A1 etc.) of the pad as the origin

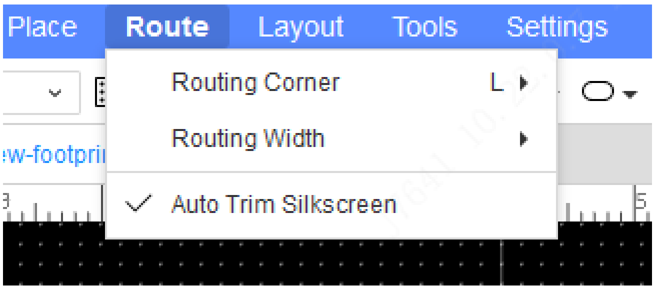


Auto Trim Silkscreen

The automatic cutting silk screen is to cut off the silk screen drawn on the pad. Used when quickly drawing footprint outline silkscreen.

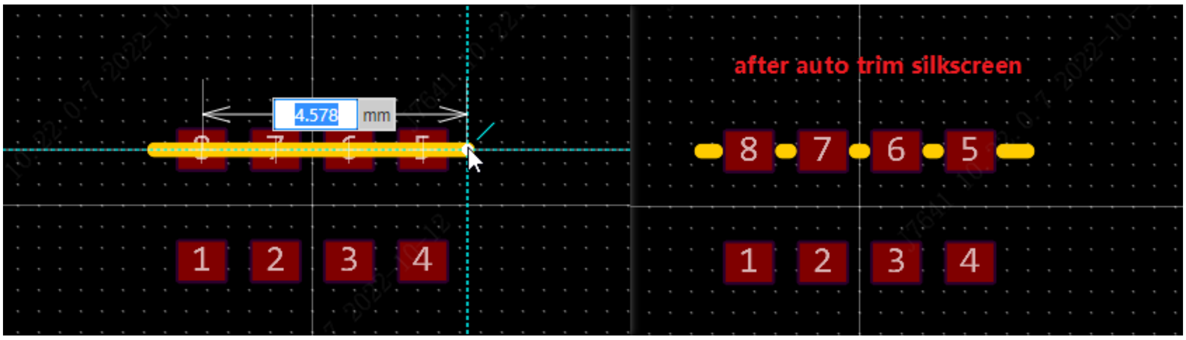
How to operate:

- Top Menu - Routing - Auto Trim Silkscreen



illustrate:

- Automatic cropping silk screen can only be used for polylines; primitives such as rectangles and circles cannot be used effectively
- Cut silk screen function is enabled by default
- It only works for

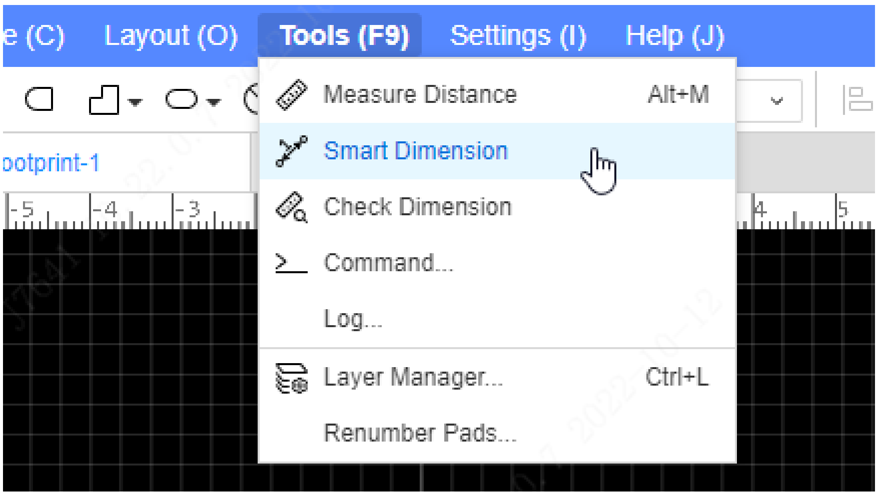


Smart Dimensions

The Smart Dimension of EasyEDA Pro has the same functions as the Smart Dimension of the Standard Edition, both for measuring and drawing footprints quickly.

Instructions:

- Top Menu - Tools - Smart Dimensions



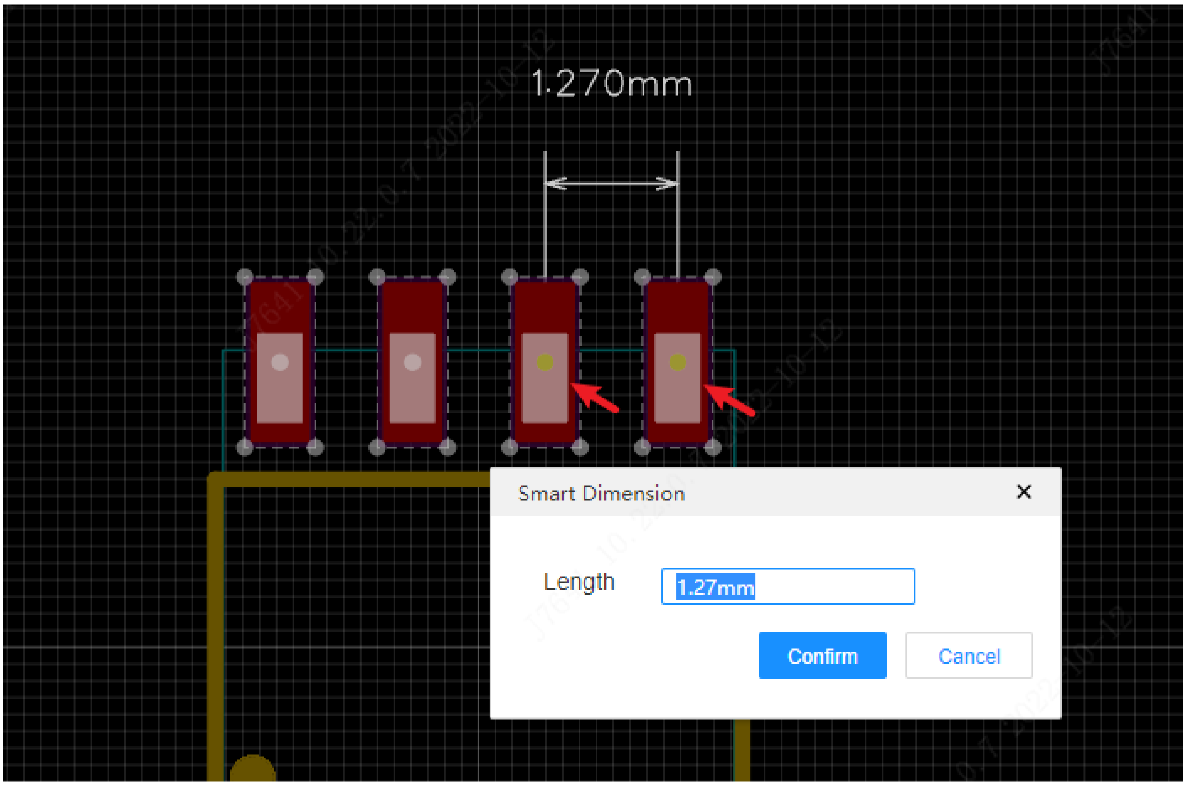
When a small white circle appears around the pad, Smart Dimension has been successfully turned on.

illustrate:

- Measurement points of the same pad cannot be selected

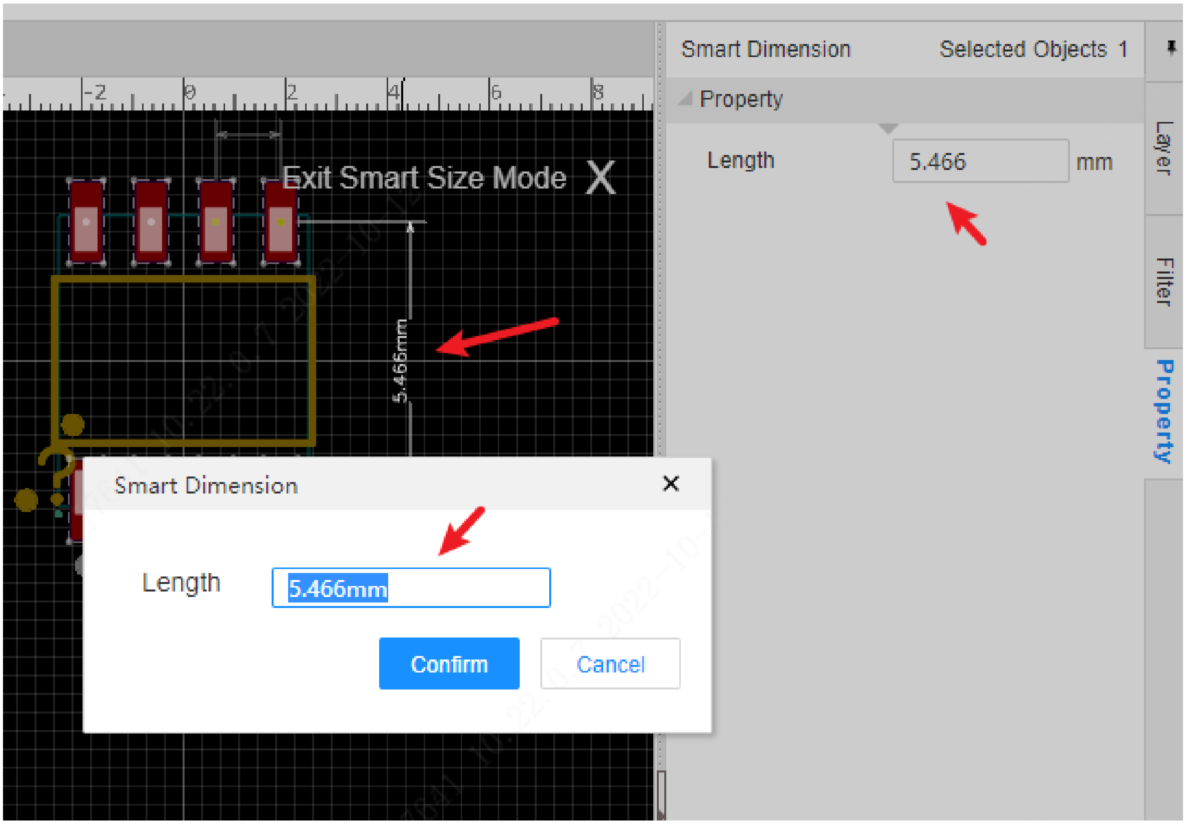
Measurement

Enter the smart dimension mode, you can click the pad's five control dots in two clicks, and then move the mouse cursor, you will get the dimension, left click will popup the length dialog.



edit

when you double click the dimension value, you can modify it

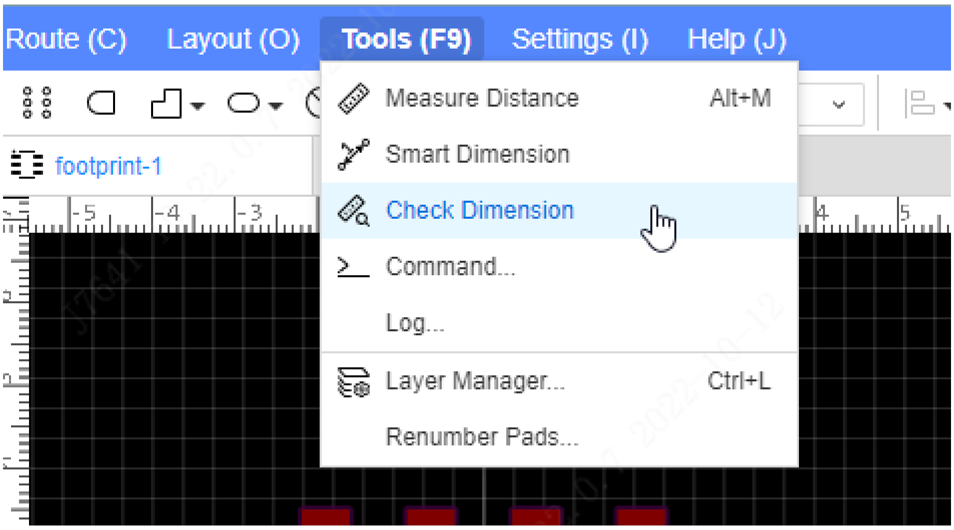


Check Dimensions

The check size function is used to quickly check the sealed size. Different from the smart size, the check size cannot be edited, but the content displayed will be more than that of the smart size.

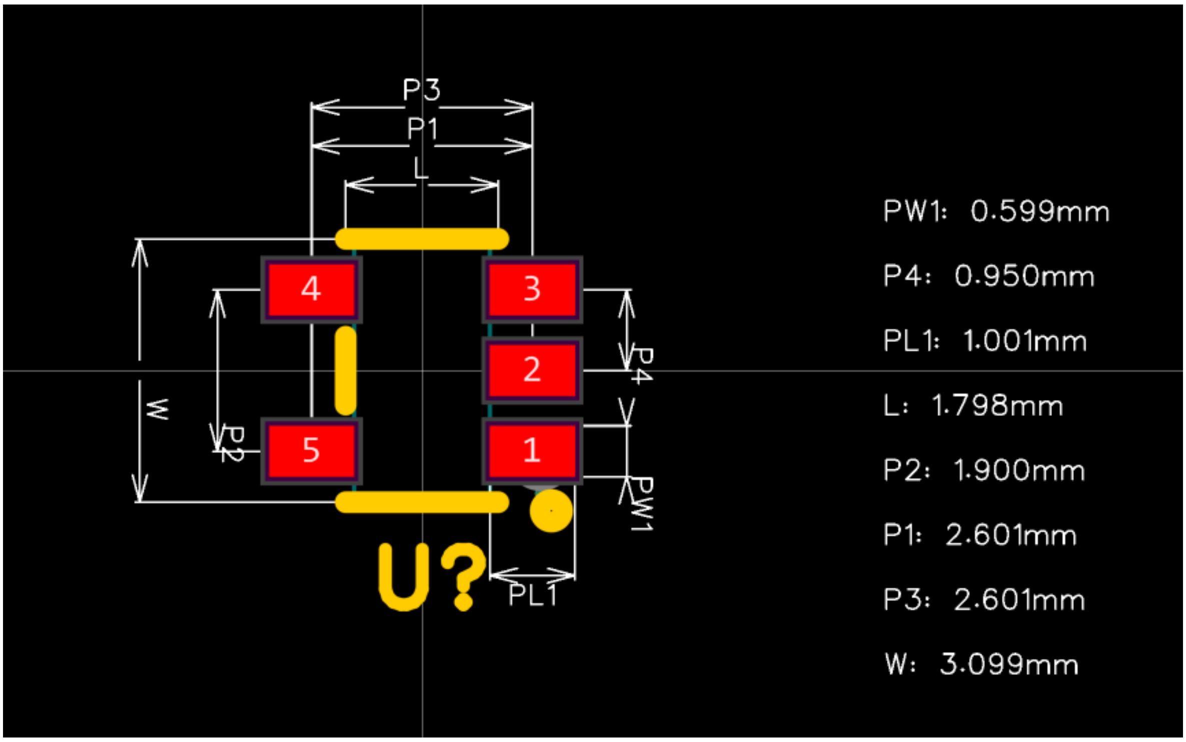
Instructions:

- Top Menu - Tools - Check Dimensions



illustrate:

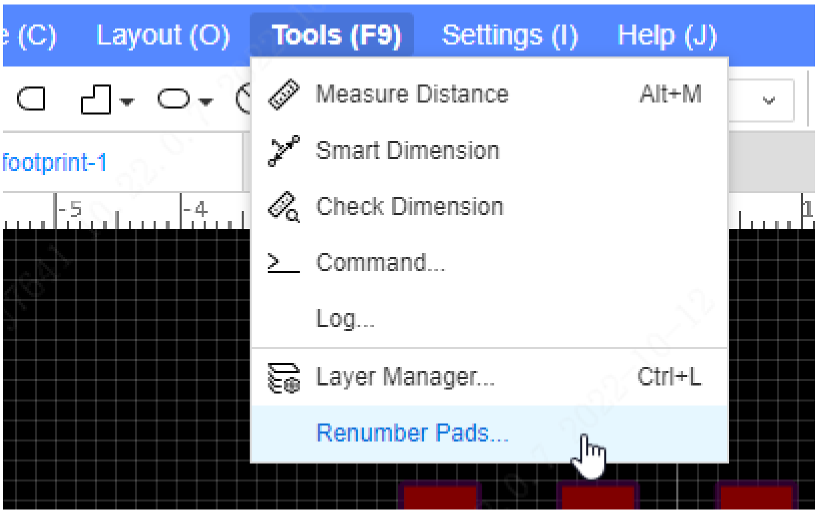
- The content displayed in the check size is generated according to the current device. After the device is modified or edited, the check function will be automatically exited.



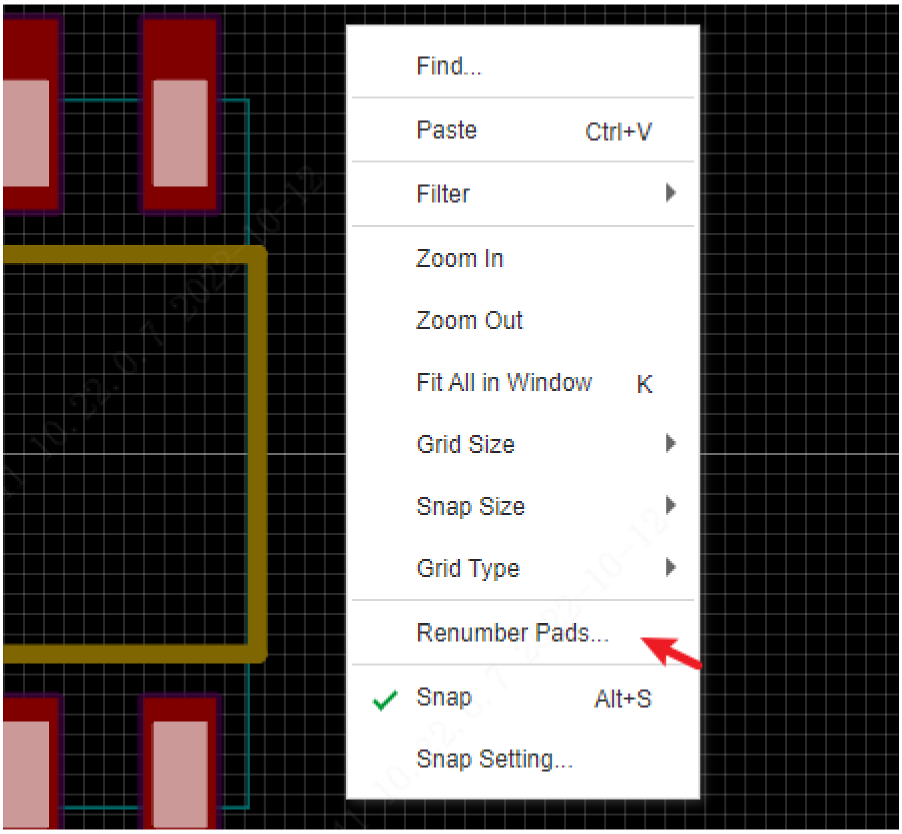
Renumber Pads

If you have placed too many pads and need to renumber the pads, you can use the "Renumber Pads" function.

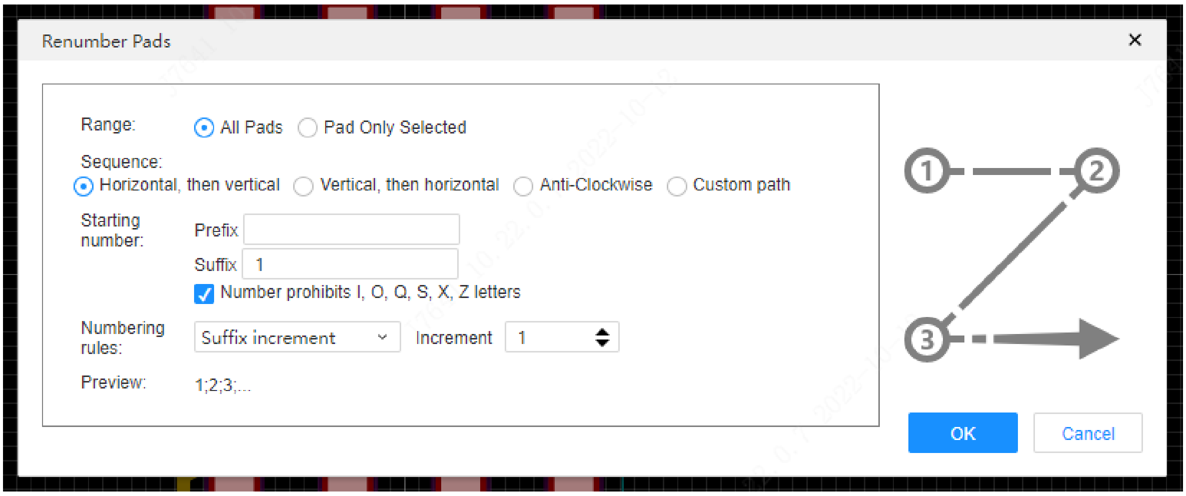
Entry: Tools - Renumber Pads.



Or the canvas right-click menu "Renumber Pads".



You can select the pads you want to change the pad numbers



this tools is easily to renumber the BGA pads or other pads.

Footprint Naming Rule

EasyEDA Footprint Naming Rule

EasyEDA Footprint Naming Rule Reference

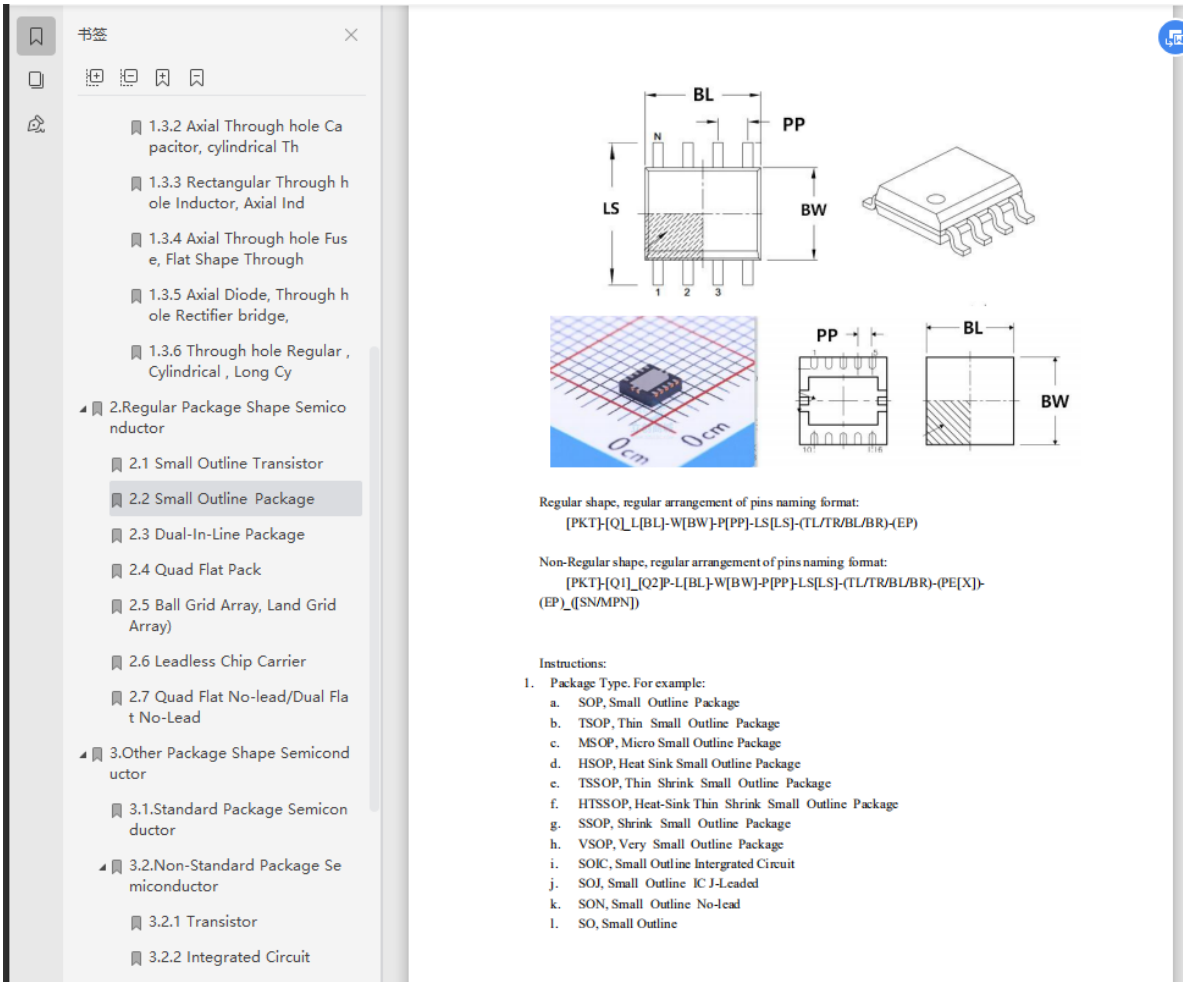
Introduction:

Believe that the vast number of electronic engineers will encounter the problem of footprint name naming, and now EasyEDA to provide everyone with a reference scheme - "EasyEDA Footprint Naming Rule Reference".

Each company should have its own footprint naming specification,EasyEDA is no exception, EasyEDA has more than 180,000 of official library (LCSC library), multiple engineers in the construction of footprint, more need unified library rules and footprint naming rules to ensure library consistency and footprint reuse.

Written by LCSC engineering department and EasyEDA team, after close one year of running in, now we are very happy to release the "EasyEDA Footprint Naming Rule Reference".

EasyEDA has been established according to the new footprint naming specification Footprints for more than half a year, and EasyEDA will continue to draw new library according to this rule.



The majority of EasyEDA users can also according to this rule:

1. Find the components of the specified footprint type;
2. Create your own or team's or company's footprint according to this rule;
3. Quickly reuse the official footprint.

Characteristics:

1. The rules of "footprint type _ feet number - body width - foot distance - body length - foot azimuth - polarity direction _ series name" are adopted in naming, so that users can quickly and clearly footprint most of the information
2. It covers most common component classification and encapsulation types and can quickly locate and query
3. Continuously expand new naming rules according to new components or packaging types, and continuously update and maintain
4. Public distribution, free of charge for both individuals and enterprises

Disadvantages:

Titles of some footprint types are too long

Update record:

2019.12.27 First release

Download:

Download: [EasyEDA Footprint Naming Rule Reference.pdf]

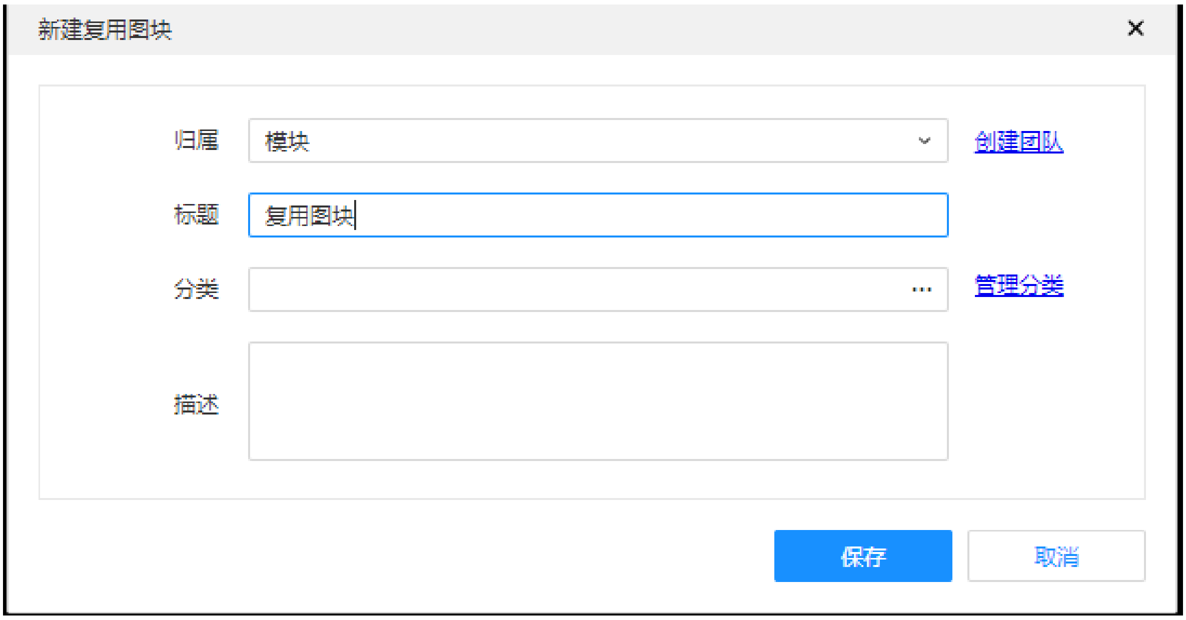
Reuse Block Library

Create Reuse Block

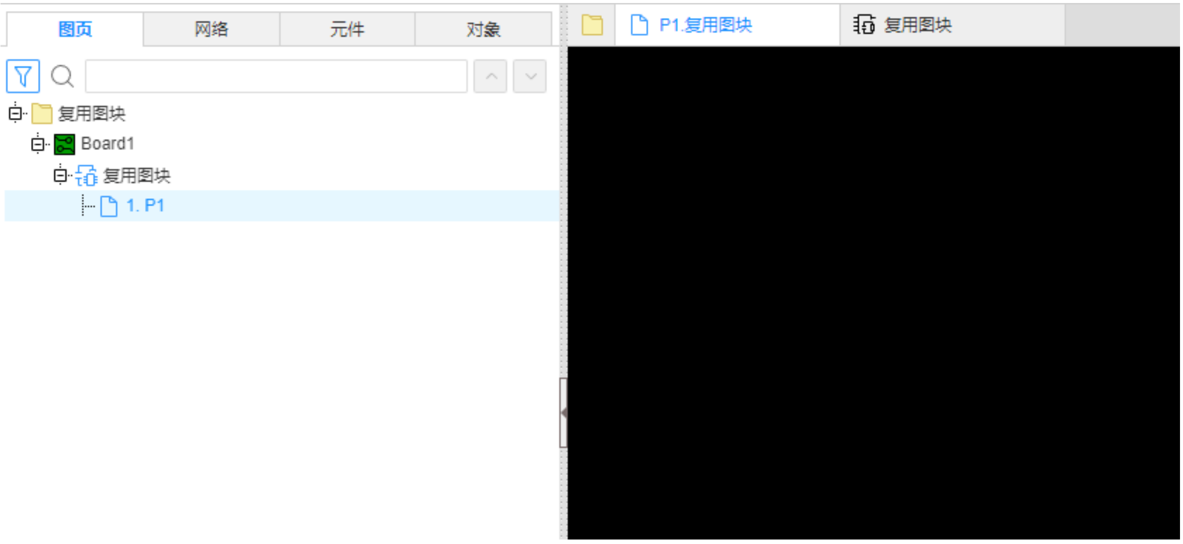
When drawing complex schematic diagrams, you need to use reuse blocks. In other words, reuse blocks are hierarchical schematic design.

- Top Menu - File - New - Reuse Block

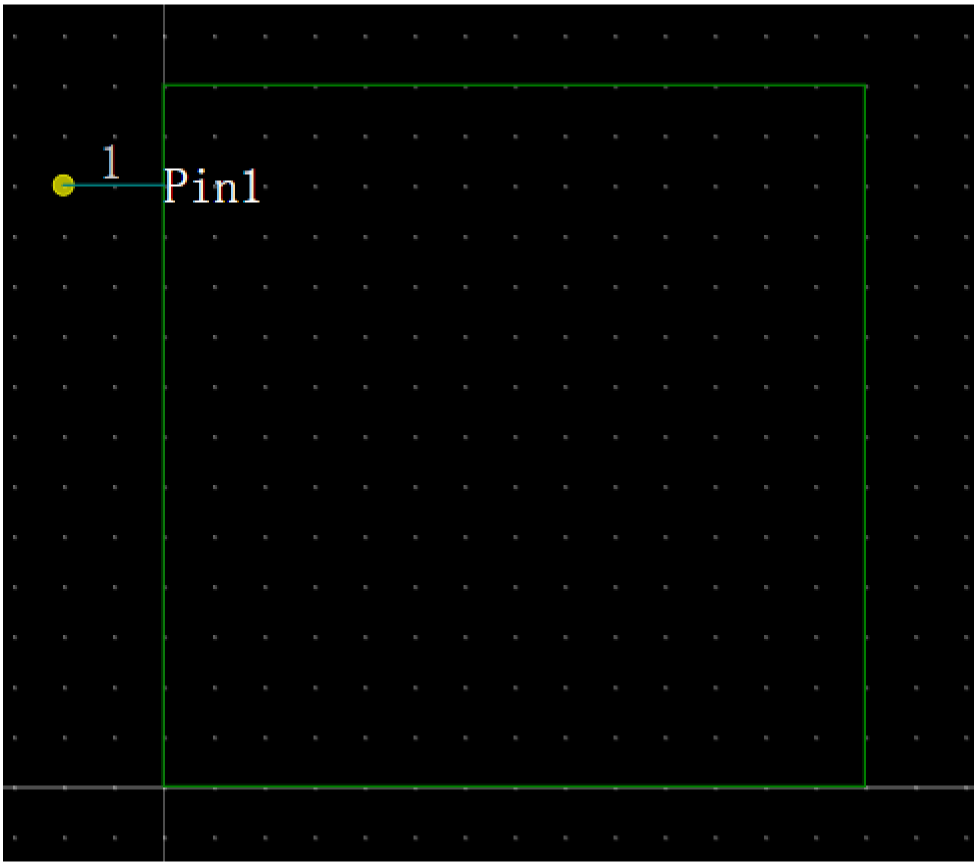
Fill in the information of the reuse blocks, click OK to generate reuse blocks



After the creation is completed, the system will automatically create a schematic diagram and a multiplexing block. These two files are related to each other and mainly include two parts: the main circuit diagram and the sub-circuit diagram. The relationship between the main circuit diagram and the sub-circuit diagram is the relationship between the parent circuit and the sub-circuit, and the sub-circuit diagram can still include the next-level sub-circuit.



The main circuit diagram is equivalent to the block diagram in the circuit diagram of the whole machine. A block diagram is equivalent to a module. Devices cannot be placed in the main circuit diagram, but only pins can be placed to connect to the sub-circuits.



Generally, the sub-circuit diagrams are some specific circuit schematic diagrams. The connection between the sub-circuit diagram and the main circuit diagram is realized through the ports in the block diagram.

Generate reuse blocks from schematic

The design of generating reuse blocks from schematic diagrams is a bottom-up design that generates parent diagrams from sub-diagrams

Steps:

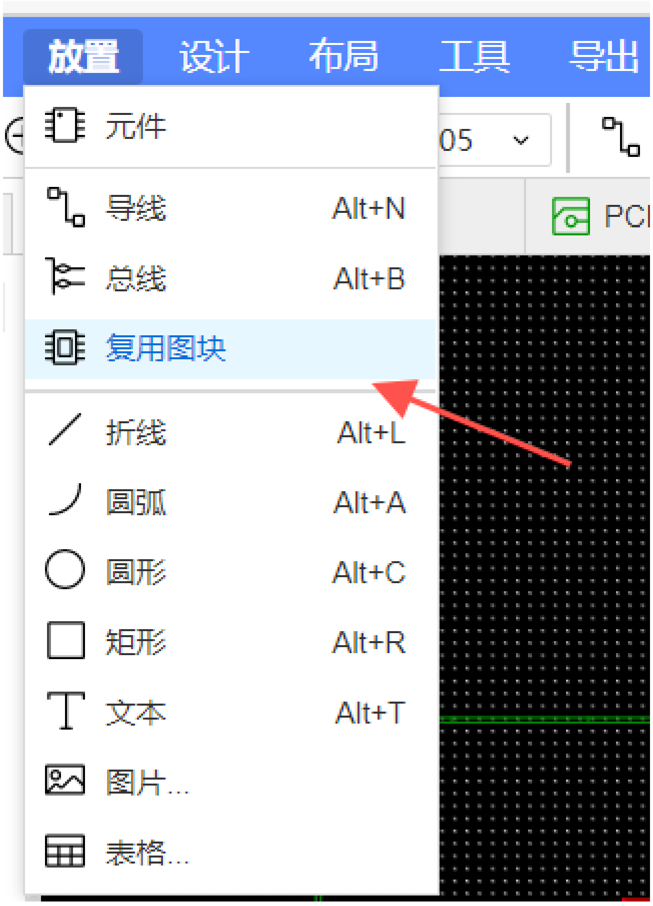
- Top Menu - Design - Generate/Update block Symbols
- Top Toolbar - Select Generate/Update block Symbols

Place to create reuse blocks

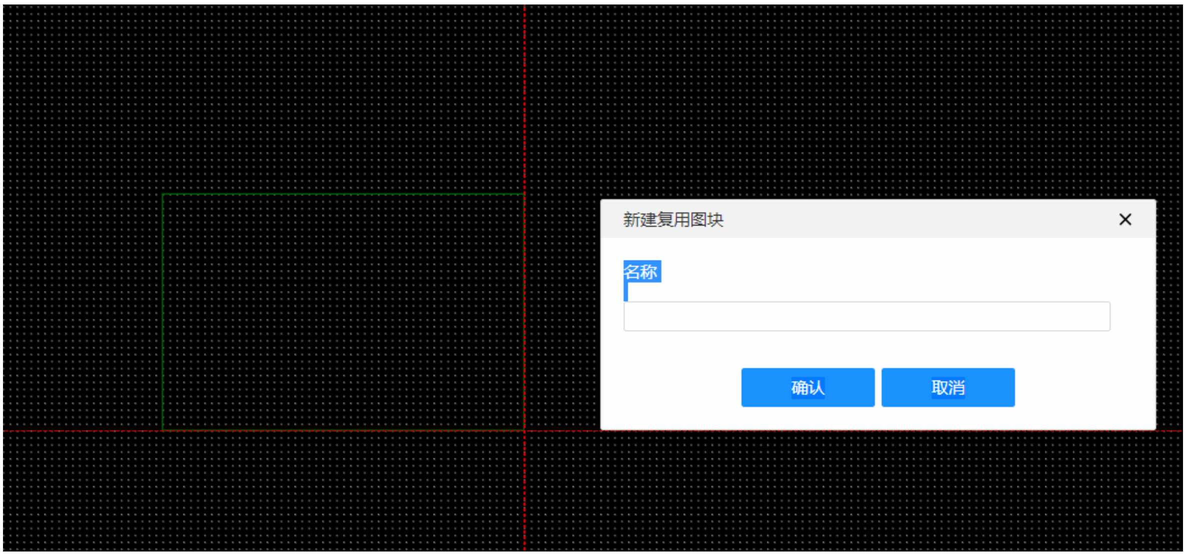
Placing reuse blocks in the schematic to create

Put operation:

- Top Menu - Place - Multiplexing block
- Bottom Component Library - Reuse Blocks



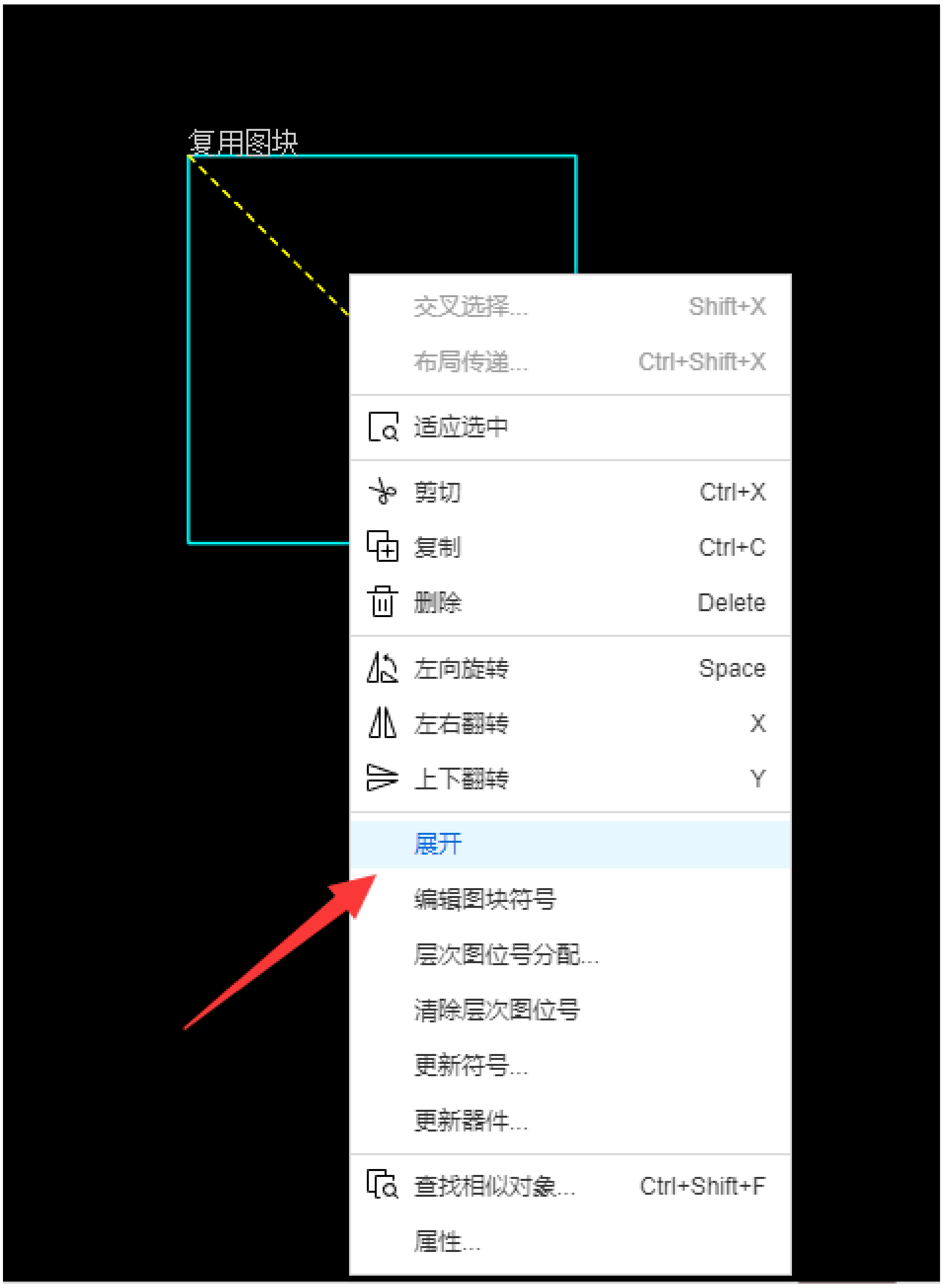
After selection, draw a block box in the schematic, and then name the block box to complete the creation



Reuse blocks created from schematics will be displayed under Project in the left panel

image-20211227141356057

Click the left panel to open the sub-schematic diagram or click the multiplex block box - click the right mouse button - select Expand, you can expand the multiplex block sub-schematic diagram



Place Reuse Block

Place the already created reuse block.

Steps:

- Bottom panel - Component library - Reuse block - click to place

image-20210521104815341

Edit/Update Reuse Block

Edit and modify the created reuse block

Steps:

- Bottom Component Library - Select Reuse Block - Right Mouse Button - Edit Schematic/Symbol

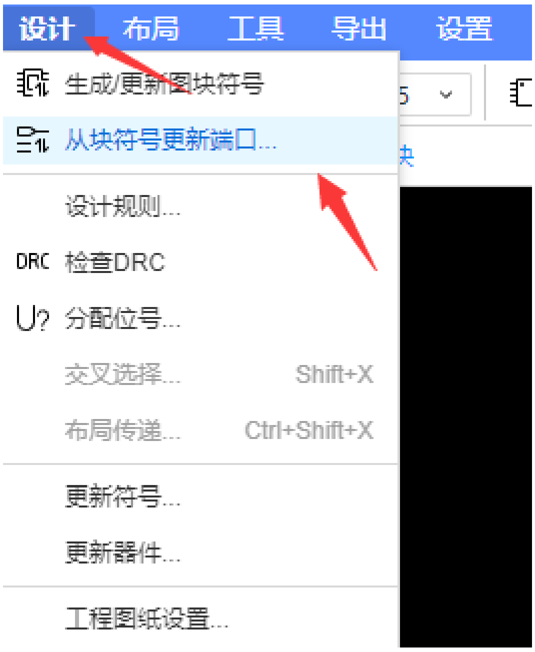
!

Update the reuse block from the schematic

Ports modified in schematic block, updated from schematic to reuse block

Steps:

- Top Menu - Design - Generate/Update Block Symbols
- Top Toolbar - Shortcut Icons

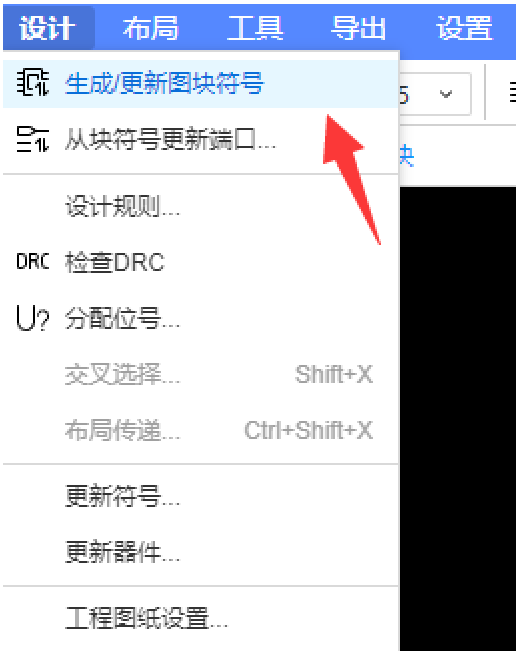


Update from reuse block symbols

Updates to Sub-Schematics Modified in Reuse Blocks

Steps:

- Top Menu - Design - Generate/Update Block Schematic
- Top Toolbar - - Generate/Update Block Schematic

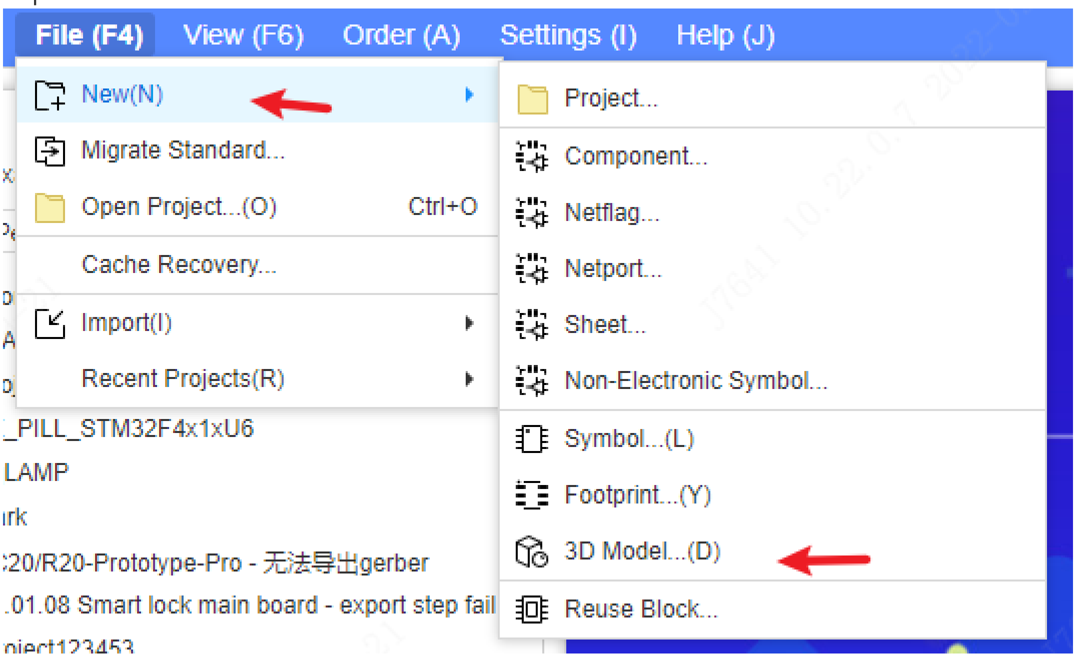


3D Model Library

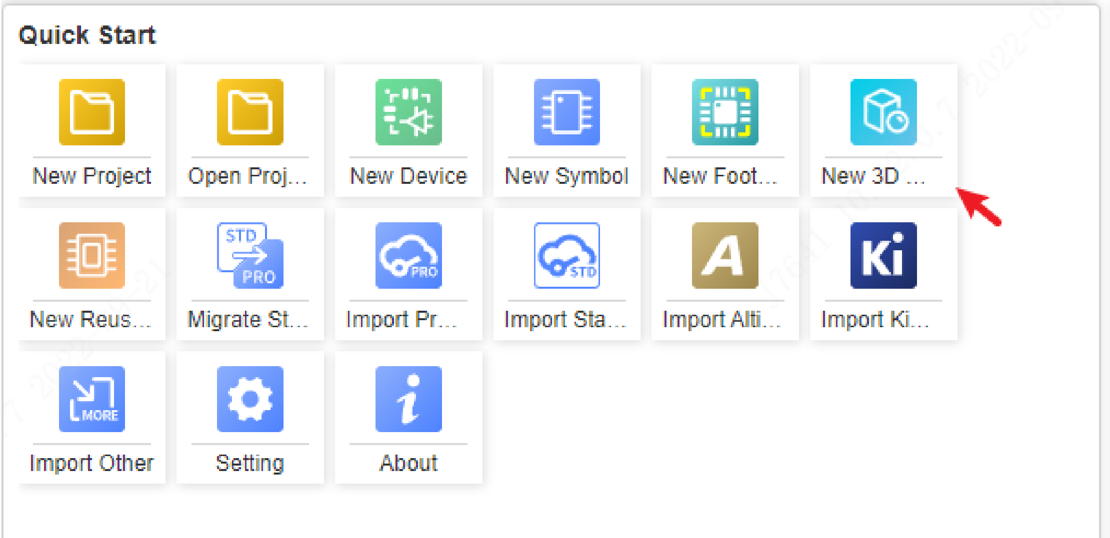
Create 3D Library

Before binding the device library, you need to create a 3D library file before binding. The operation steps are as follows:

- Top Menu - File - New - 3D Model

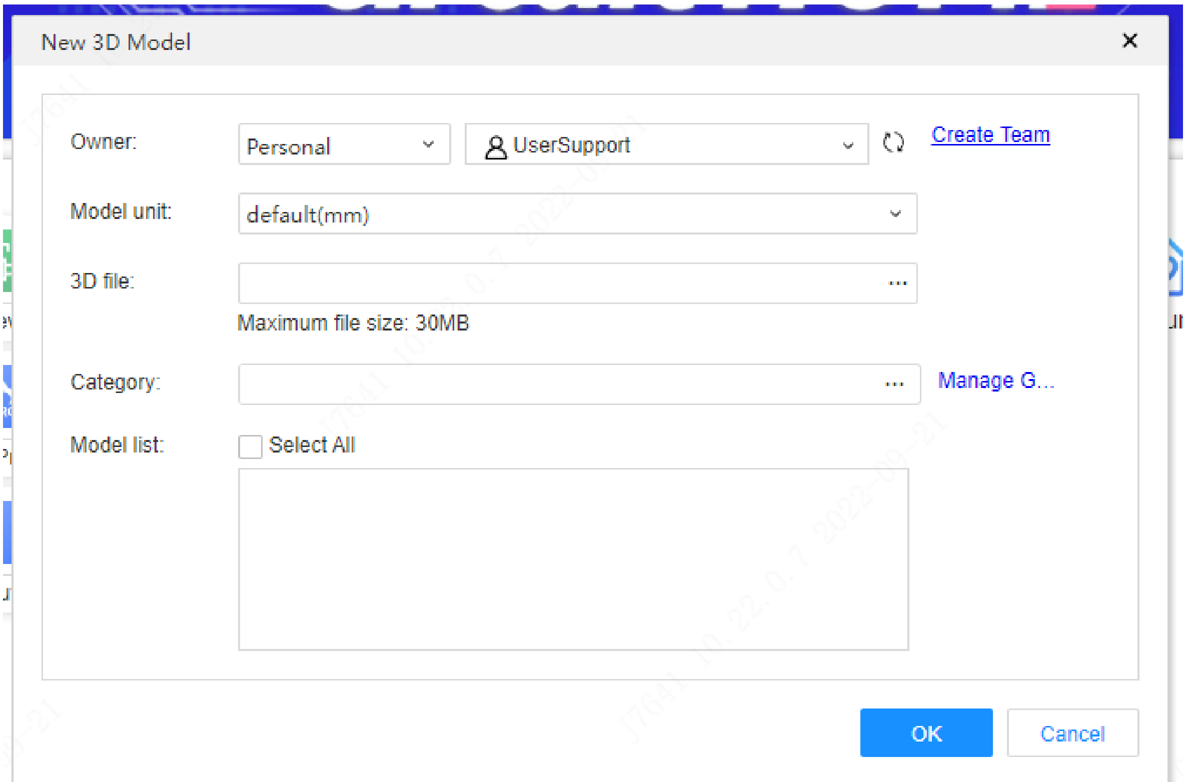


- Or at start page - quick start



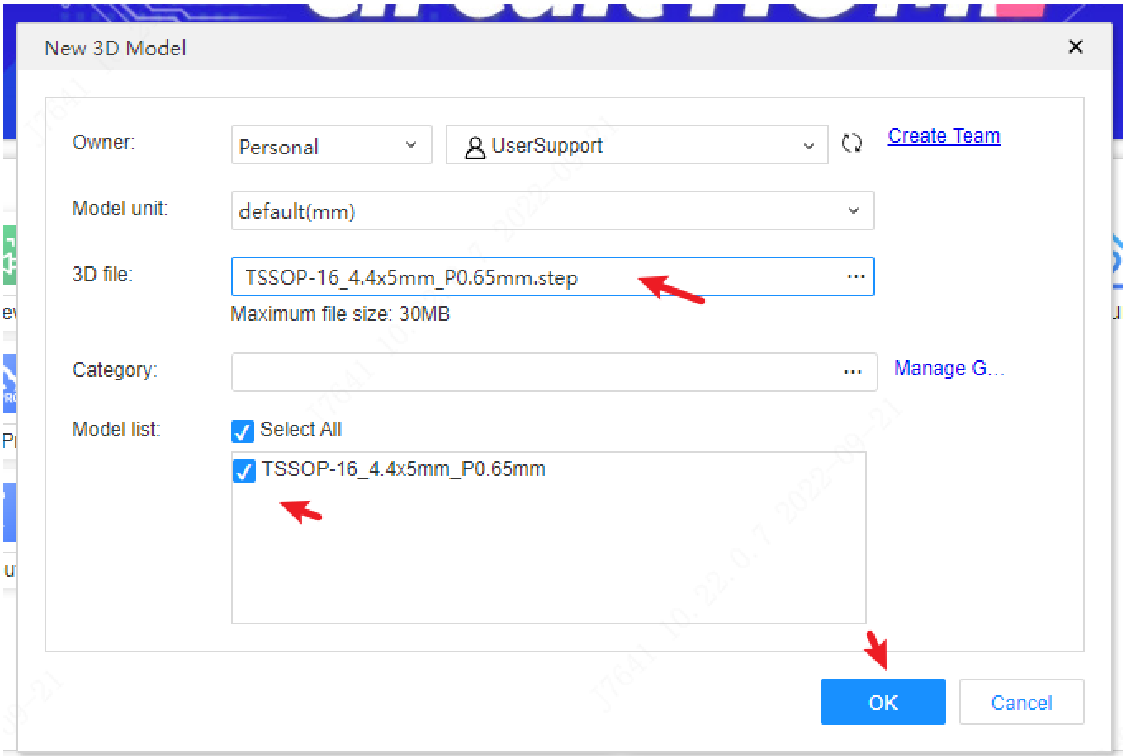
Enter relevant information in the pop-up window

- Owner: the user or team who created the 3D Warehouse;
- Model unit: The unit can choose cm, m, mil, inch. According to the size of the drawn 3D model, the default size is mm;
- 3D file: select the imported 3D file, you can choose zip, step, stp, wrl, obj format file. 3D models can be imported in batches, just compress the 3D files into a zip file footprint and import;
- category: classify the created 3D models;
- Model list: After selecting the imported 3D file, the list will display the files that successfully imported the 3D model;

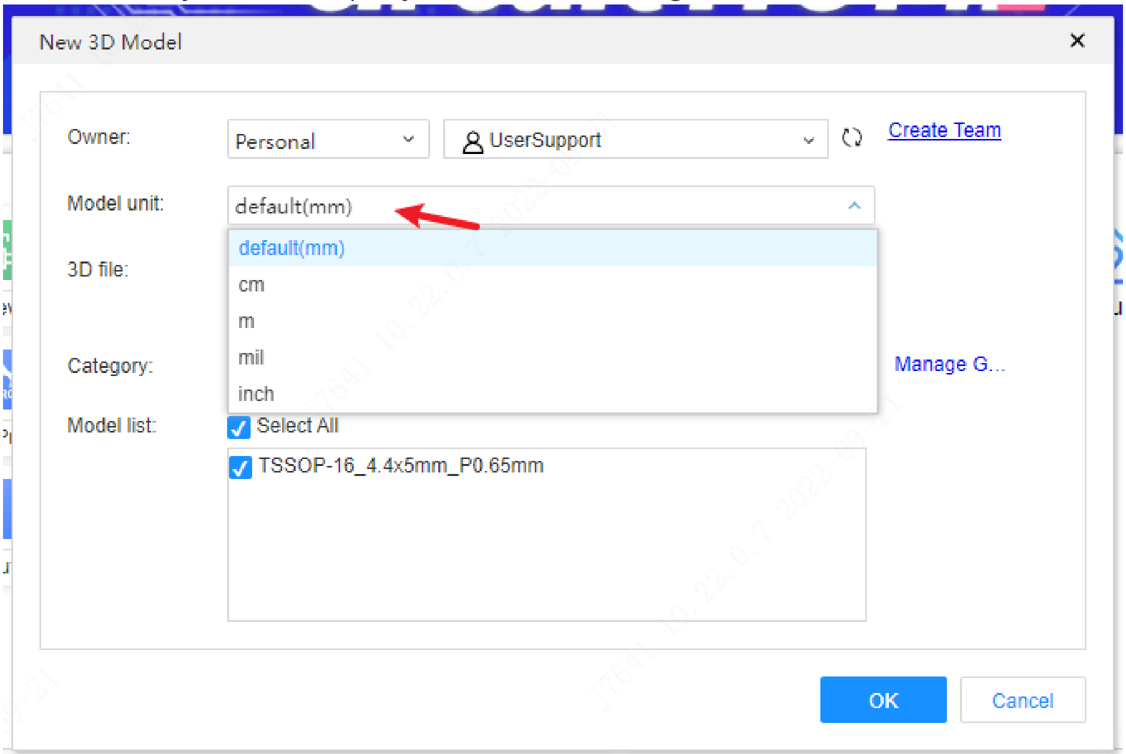


Steps:

1. Select the attribution user of the model



2. When the model is created, the default is mm. There is no need to modify it again here. Just select the default mm unit. You need to select the correct unit for your model, if you don't know the unit, you need to open your model at 3D design tool to check.

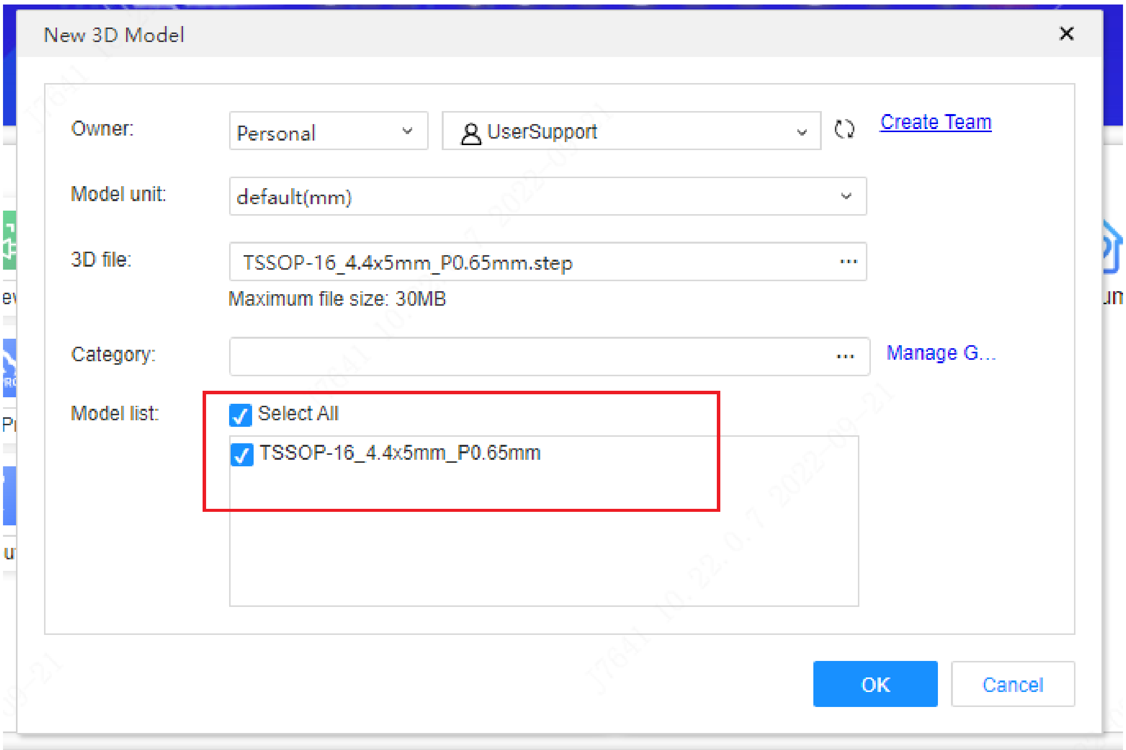


3. Click Select File to select a 3D model file in STEP/OBJ/WRL/STP/ZIP format.

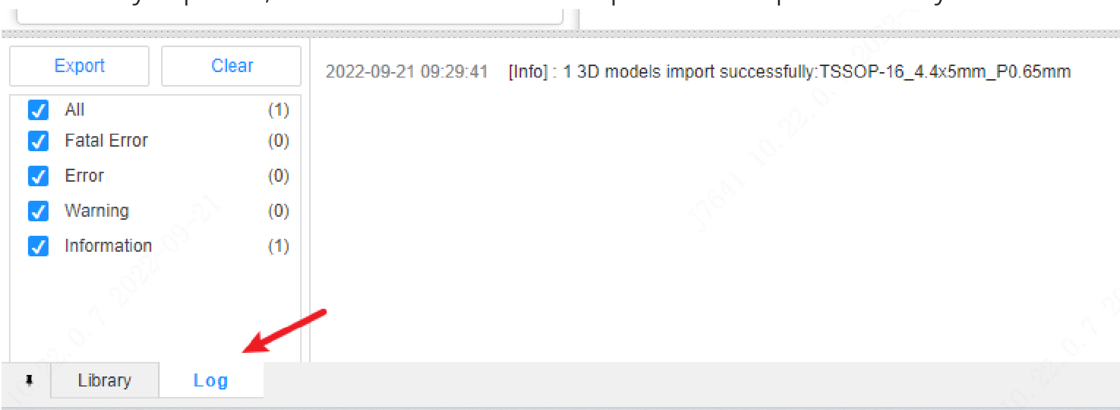
4. Click category to classify the model, which is convenient for later search. If there is no category, you can choose to manage the category to create.

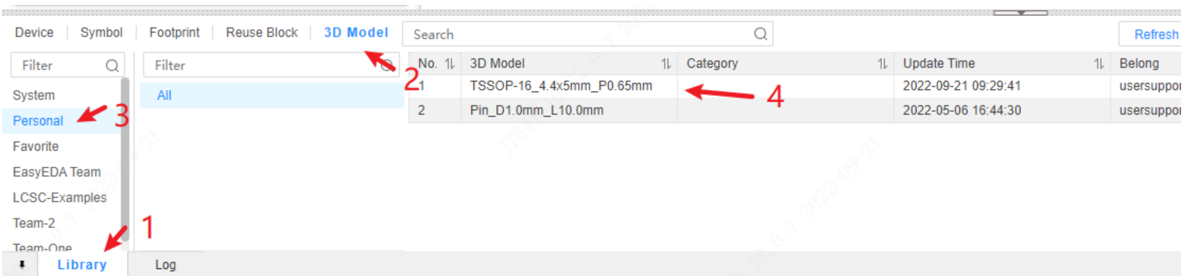
5. Click the model you just selected in the model list, and click to check it. If you don't check it, you cannot create a 3D model file. If you import WRL or OBJ zip files, this list will list all

models



6. Finally, click OK, and the "Log" in the bottom panel shows that the 3D model was successfully imported, which can be viewed in the personal component library.

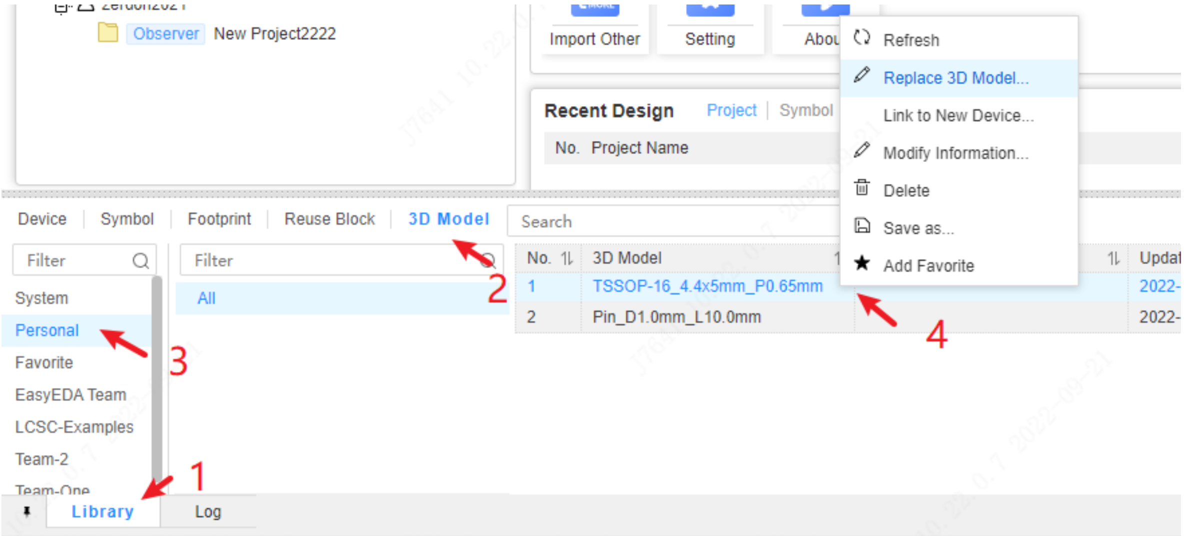




Edit 3D Library

EasyEDA Pro supports 3D models that replace in 3D libraries.

In the component library at the bottom, after finding the 3D model, click on the personal 3D model list, right-click to replace the 3D model



You can also right-click to modify the 3D model title and other information.