

EasyEDA Tutorial

2020.08.07

v6.4.3

EasyEDA Editor: <https://lceda.cn/editor>

EasyEDA Desktop Client: <https://lceda.cn/download>

Remark

- This document will be updated as the new features of the editor are updated.
- The latest revision please refer at [EasyEDA Tutorials.pdf](#)

Editor FAQ

Please spend a few minutes reading this FAQ, it will save you lots of time getting started with EasyEDA.

Tutorial

Download for PDF

[EasyEDA-Tutorials.pdf](#)

Video Tutorials

[Youtube - EasyEDA](#)

Ask for Help

[Contact Us](#)

Update Records

[Update Records](#)

Schematic

If I update the schematic, how do I then update the PCB?

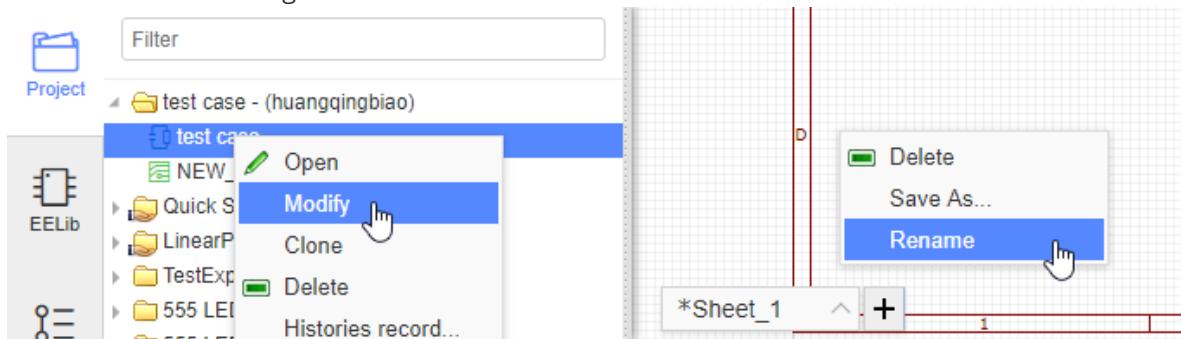
Using: "Menu - Design - Update PCB".

Alternatively, you can import changes from the schematic from within the PCB Editor:

<https://docs.easyeda.com/en/PCB/Import-Changes/index.html>

How to rename a Sheet/Page or modify description.

In this menu, there is a `Modify` option, so you can rename your files. Double click or right-click the sheet tab can change the sheet title too.



What is the unit of the schematic sheet? How to change schematic unit?

The basic unit of the schematic sheet is the pixel. 1 pixel is about 10mil (0.001 inch) but please note that this use of the pixels as a unit in a schematic is just for reference.

For a complex project, I want to split the schematic over several sheets. Does EasyEDA support hierarchy?

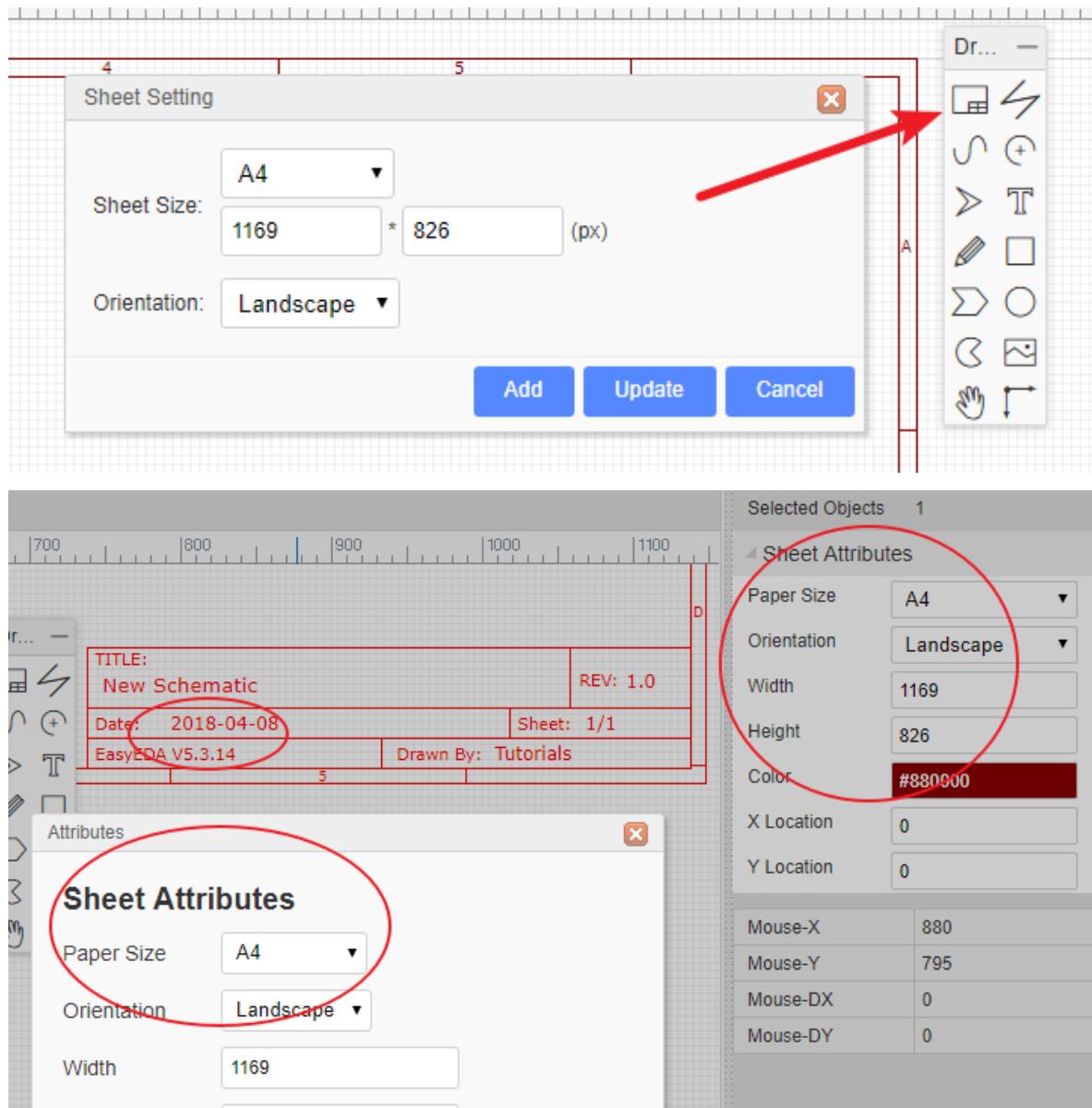
EasyEDA don't support hierarchy, but support multi-sheets.

Please check out this link <https://docs.easyeda.com/en/Schematic/Multi-Sheet/index.html>

How to change the sheet size and modify the design information.

To change the sheet size, move the mouse anywhere over the lower right area of the drawing border or frame until the whole border highlights red and then right-click on it. Paper size and orientation can then be changed in **Sheet Attributes** in the right hand panel.

To modify the design information, left-click on the relevant blue text in the lower right area of the drawing border or frame to change it in **Text Attributes** in the right hand panel. Double left-clicking the blue text will allow you to type new information directly into the field.



How to indicate low electronic level in the Schematic Pin or Netlabel

You can add a `#` character in the pin name/netlabel last text. You can use symbols that you are familiar with. You do not have to add a line above the netlabel name.

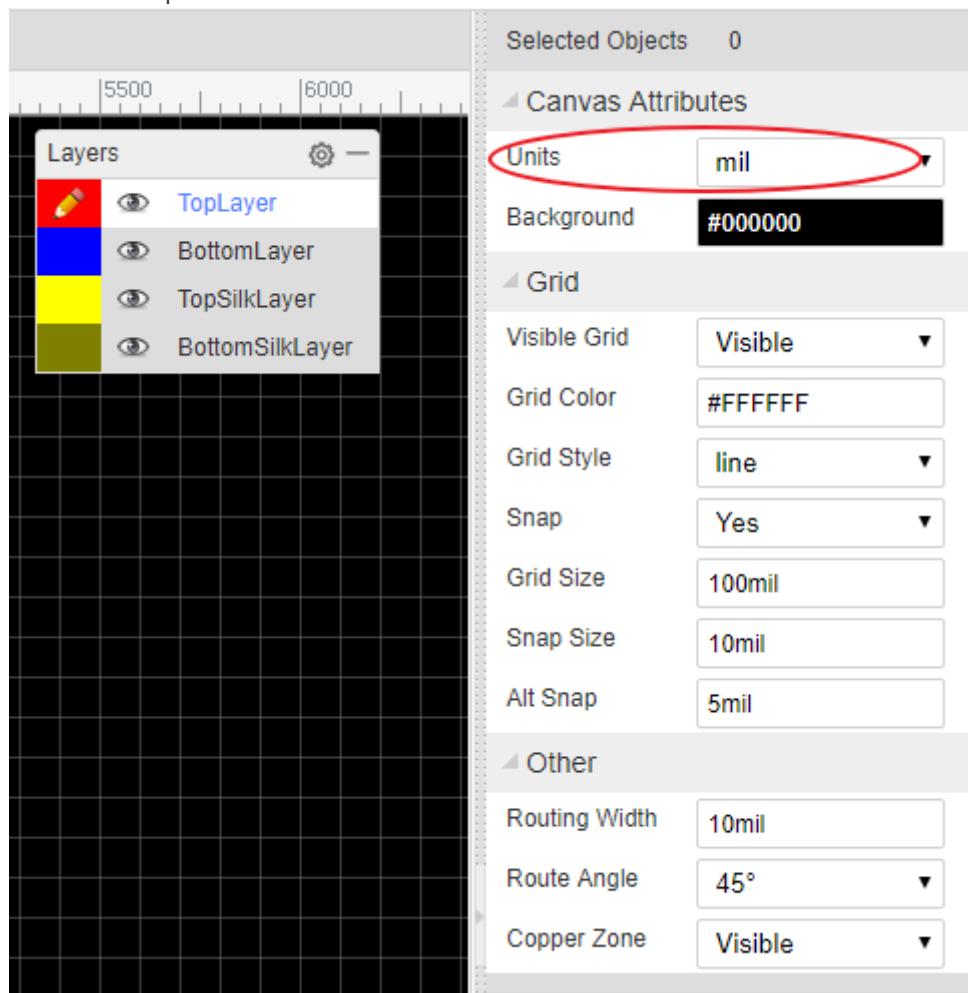
I can't convert schematic to PCB. Why is this?

1. You have not set the right footprints for your components.
2. [Prefix Conflict Error](#)
3. [Invalid footprints](#)

PCB

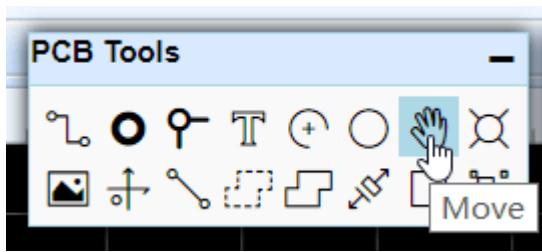
How to change the Units of PCB from mil to mm or inch.

There is an option for that in PCB canvas attributes:



How to pick and move the components on the PCB canvas quickly.

Before routing the PCB, the components need to be positioned in suitable places on the PCB. In the PCB Editor, it can sometimes be quite difficult to select components by clicking on the silkscreen outline or the pads. To select and move them more easily, please use drag mode (Hot Key `D`) or click the `Move` icon in the PCB Tools toolbar:



How to add test point in schematic or PCB?

Schematic: You can place a single pin connector from EElib, and then update its footprint.

PCB: You can place a top/bottom layer pad , and then route it with track.

Can I create a PCB without creating schematic?

Yes but for any but the simplest PCBs, please see:

[Layout PCB Without Schematic](#)

How to add more fonts for PCB.

You can refer to [Text](#) of PCB section.

How to insert an Image/Logo to PCB.

You can refer to [Image](#) of PCB section.

How to insert a DXF as board outline.

You can refer to [Import DXF File](#) of Import section.

How to create non rectangular pcb outline such as round?

You can import a DXF file for the board outline. For a round board outline, you can use an arc to do that, you just need to change to the board outline layer, then draw 1 arc like in the image below (need to adjust a bit later), you can use lines and arcs to create complex board outlines.

How to add a slot and cut out.

Please use solid region <https://docs.easyeda.com/en/PCB/PCB-Tools/index.html#Solid-Region>

Or draw a track and right-click it, use the "Convert to Board Cutout" option.

How to measure dimensions on a PCB.

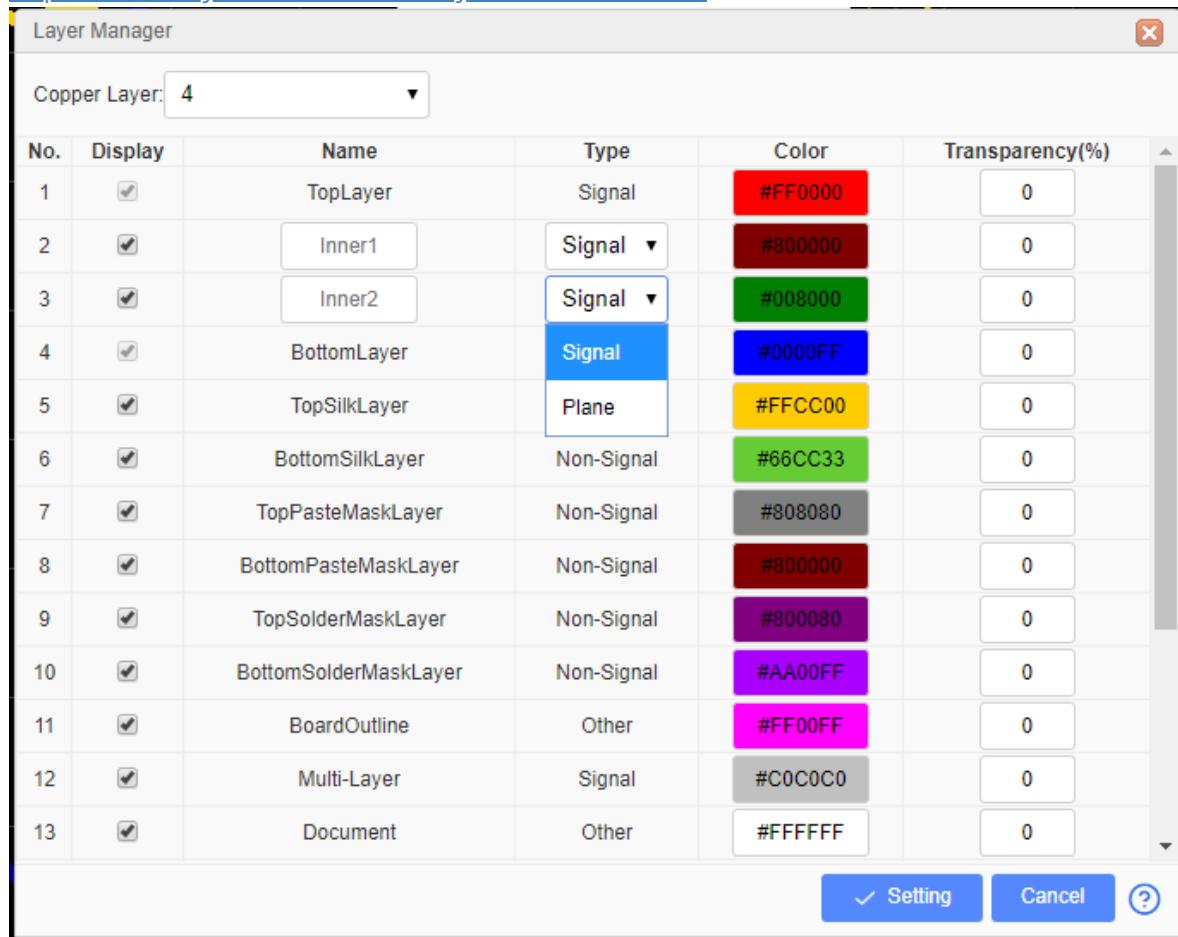
<https://docs.easyeda.com/en/PCB/PCB-Tools/index.html#Measure-Dimension>

use Hotkey M

How to add more layers.

Click the layer options button, then tick the extra layers in the dialog that opens.

<https://docs.easyeda.com/en/PCB/Layers-Tool/index.html>



The screenshot shows the 'Layer Manager' dialog box from EasyEDA. At the top left, it says 'Copper Layer: 4'. Below is a table with columns: No., Display, Name, Type, Color, and Transparency(%). The table lists 13 layers:

No.	Display	Name	Type	Color	Transparency(%)
1	<input checked="" type="checkbox"/>	TopLayer	Signal	#FF0000	0
2	<input checked="" type="checkbox"/>	Inner1	Signal	#800000	0
3	<input checked="" type="checkbox"/>	Inner2	Signal	#008000	0
4	<input checked="" type="checkbox"/>	BottomLayer	Signal	#0000FF	0
5	<input checked="" type="checkbox"/>	TopSilkLayer	Plane	#FFCC00	0
6	<input checked="" type="checkbox"/>	BottomSilkLayer	Non-Signal	#66CC33	0
7	<input checked="" type="checkbox"/>	TopPasteMaskLayer	Non-Signal	#808080	0
8	<input checked="" type="checkbox"/>	BottomPasteMaskLayer	Non-Signal	#800000	0
9	<input checked="" type="checkbox"/>	TopSolderMaskLayer	Non-Signal	#800080	0
10	<input checked="" type="checkbox"/>	BottomSolderMaskLayer	Non-Signal	#AA00FF	0
11	<input checked="" type="checkbox"/>	BoardOutline	Other	#FF00FF	0
12	<input checked="" type="checkbox"/>	Multi-Layer	Signal	#C0C0C0	0
13	<input checked="" type="checkbox"/>	Document	Other	#FFFFFF	0

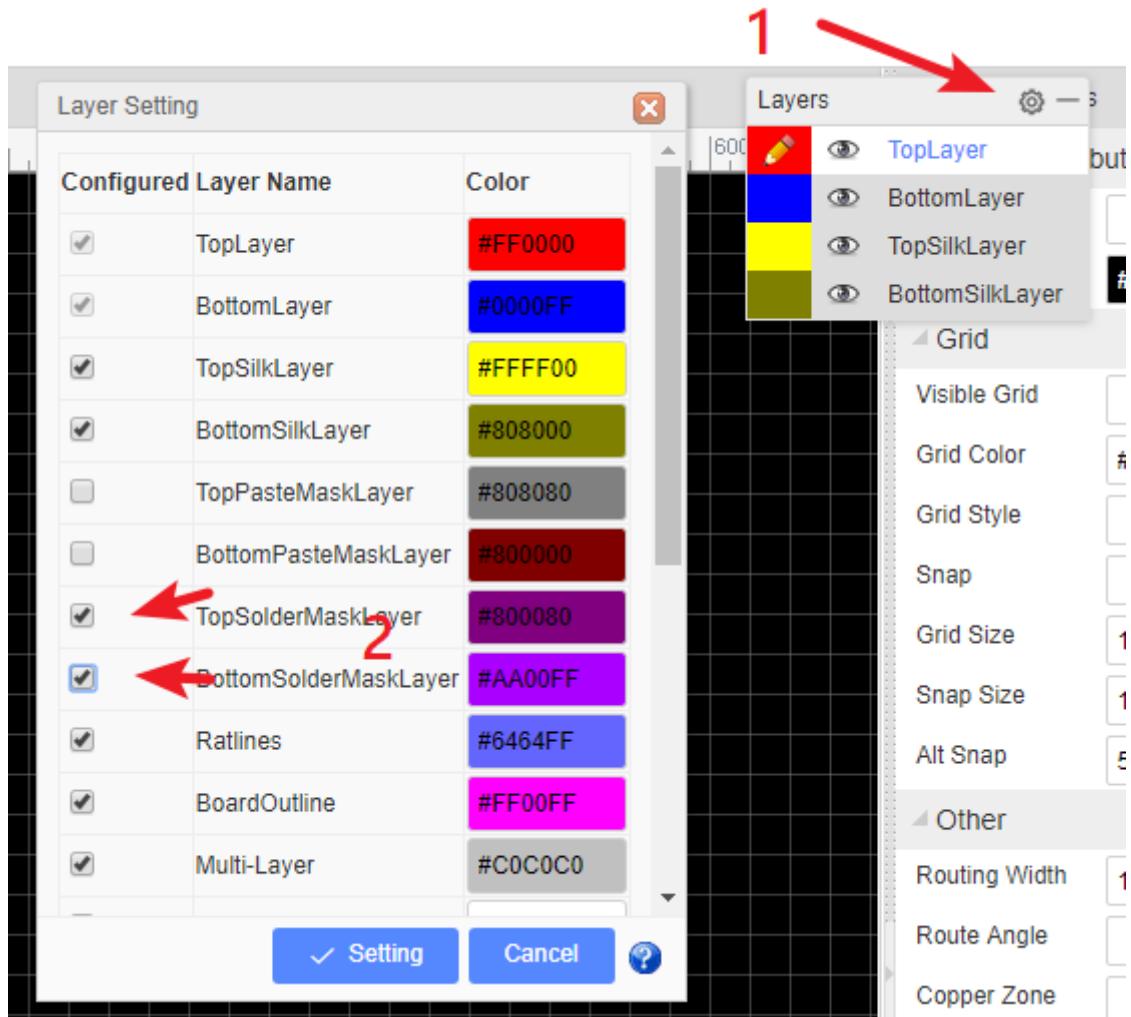
At the bottom right are buttons: a blue 'Setting' button with a checkmark icon, a white 'Cancel' button, and a blue question mark icon.

How to add solder mask aperture.

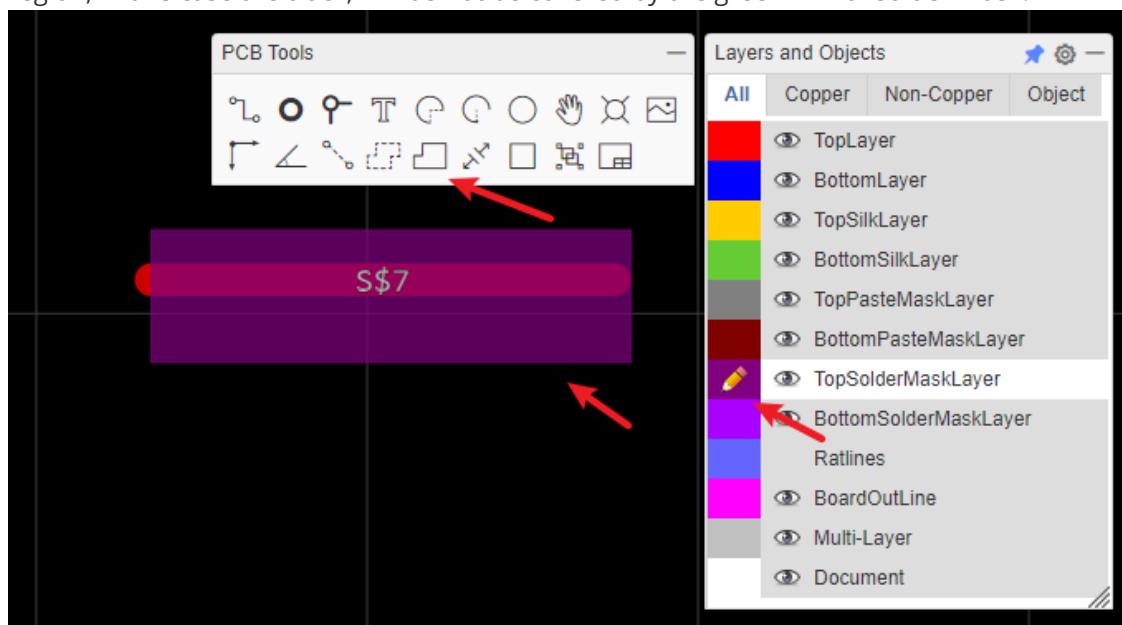
It is possible to get boards with the copper exposed so that you can apply a layer of solder over those tracks to further increase their current carrying capacity. In this case, you need to add solder mask over a copper (copper area, track, solid region).

EasyEDA will add solder mask for pads automatically. Sometimes however, you may need to add an aperture in the solder mask to expose an area of copper.

1. First, add a top or bottom solder mask layer, as required.

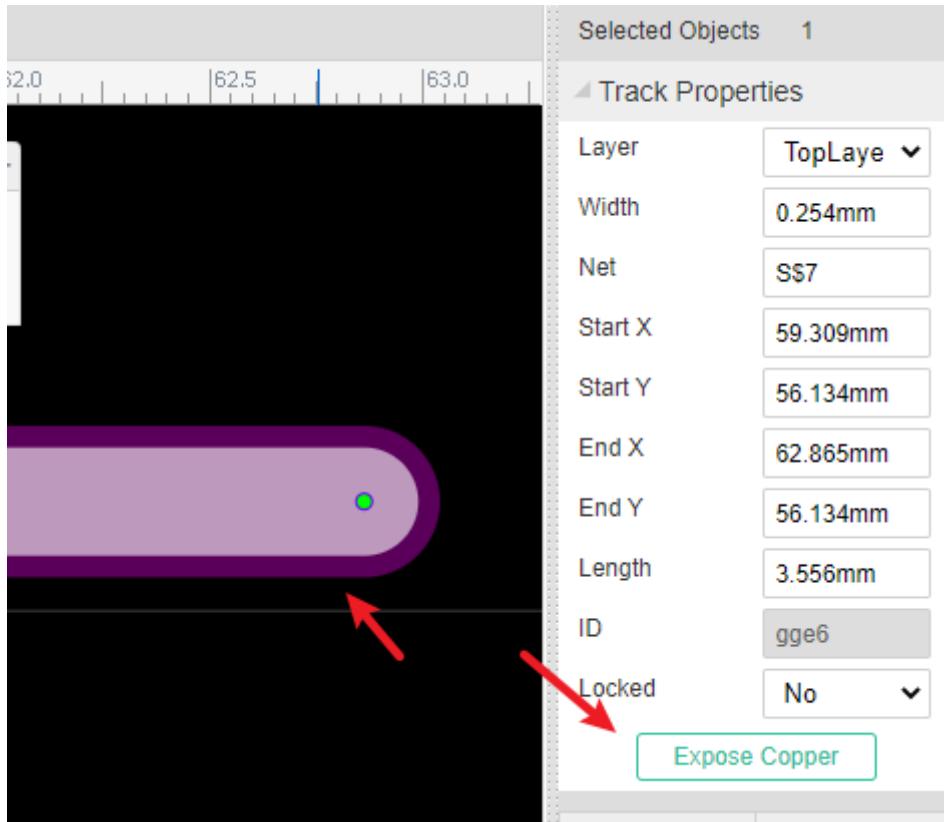


2. Next, draw a region in the solder mask layer over a copper item as illustrated in the image below. This in effect draws an aperture in the solder mask so that the copper item inside the region, in this case the track, will be not be covered by the green film of solder mask.



A common mistake is to just draw a solder mask, without a copper area, like the track pointed to by the yellow arrow. That is incorrect and does not produce the desired result.

Or you can click the track, and then click the `Expose Copper` button at the right-hand panel.



How do I set the dimensions of my PCB in the layout?

PCB's dimension/size depends on the board outline, you can create your board outline, please refer to [Board Outline](#) of the PCB section.

My PCB is complex, how can I be sure that I have routed all of the tracks?

Please refer to [Design Manager](#) of PCB section.

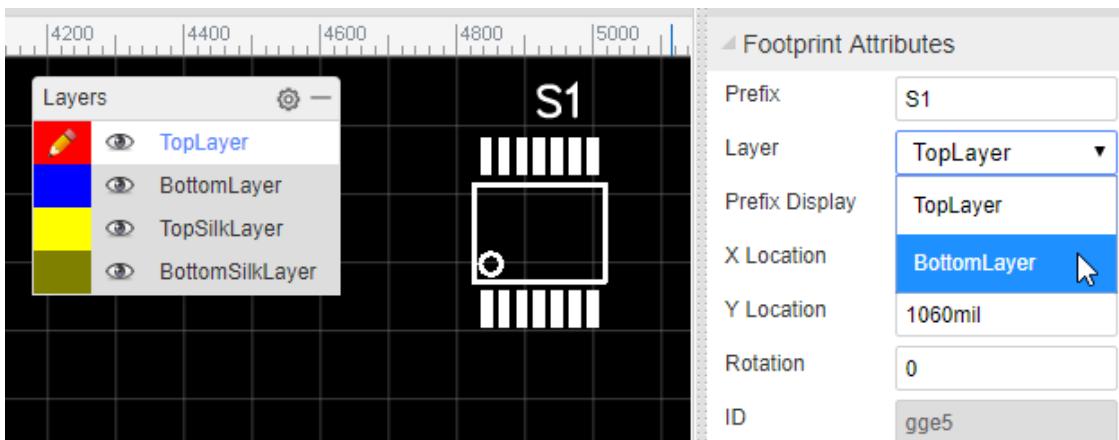
I need to start my layout again, how can I remove all of the tracks?

You can use "Menu - Route - Unroute All" and "Menu - Edit - Global Delete".

How to put a component on the bottom layer?

There are two ways to do this.

1. If your active layer is the bottom layer, then every component you place will be placed on the bottom layer automatically.
2. You can place a component then select it and change its layer attribute to `Bottom Layer` in the right hand panel.



How to panelize the PCB

Please refer at [PCB: Panelize](#)

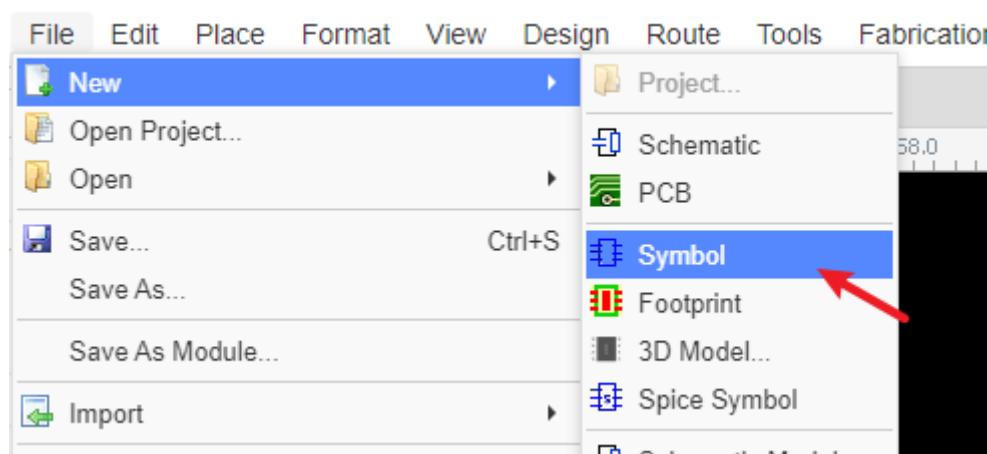
What does Warning copper area do not allow self intersection

Please refer at [Forum: What does Warning copper area do not allow self intersection](#)

Library and Parts

How to create a schematic symbol library.

File > New > Symbol



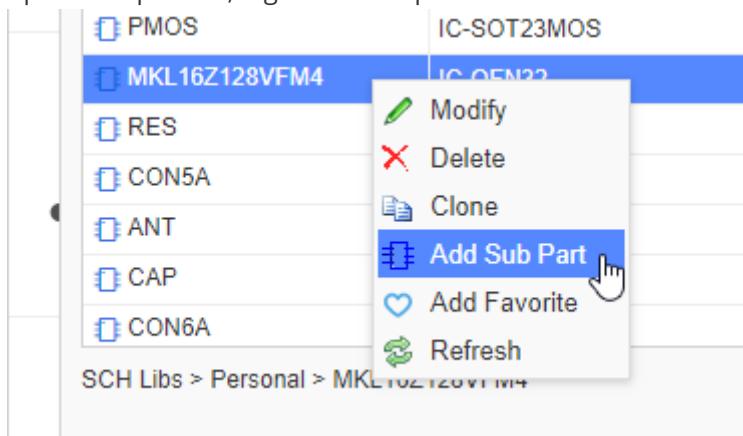
How to tag my schematic library symbol.

After creating the library and saving, you can add a tag for it, and you can add and edit the tag at "Library":

The screenshot shows the EasyEDA library interface. On the left, there's a sidebar with sections for 'My Libraries' (containing 'All' and 'DEVKIT'), 'My Favorites' (containing 'All'), 'thisateamfortest' (containing 'All'), and 'EasyEDA Team' (containing 'All'). The main area has tabs for 'Search Engine' (EasyEDA), 'LCSC Electronics', 'Types' (Symbol selected), 'Classes' (Work Space selected), and navigation buttons like 'Follow'. A search bar at the top right says 'Search symbol, footprint etc.' with a magnifying glass icon. Below the search bar are buttons for 'Help Verify', 'Footprint', 'Spice Symbol', 'SCH Module', 'PCB Module', and '3D Model'. The main content area displays a table of components. One row for 'NCP1117ST18T3G' is selected, and a context menu is open over it. The menu options are: Edit (highlighted with a red arrow), Modify (highlighted with a red arrow), Delete, Clone, Add Sub Part, Add Favorite, Refresh, View Datasheet..., Report Error..., View Owner, and View Detail. A modal dialog box is overlaid on the table, titled 'Modify file info'. It contains fields for 'Title:' (set to 'NCP1117ST18T3G'), 'Description:', and 'Tags:'. A note at the bottom says 'Split by ":" for multi tags'. At the bottom right of the modal are 'OK' and 'Cancel' buttons, with a red arrow pointing to the 'OK' button.

How to create sub parts for multi-part components.

In personal part list, Right click the part then select **Add Sub Part** from the menu that opens:



How to change the footprint for a component.

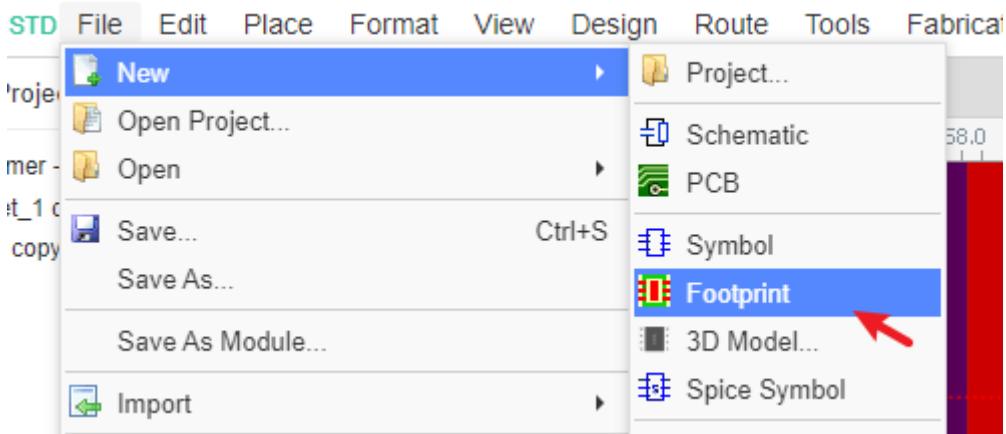
<https://docs.easyeda.com/en/Schematic/Footprint-Manager/index.html>

How to add sub parts to a schematic.

You can add sub parts to a schematic one by one but please note that the sub parts prefix must be in the form of U1.1 U1.2 etc, and not U1.A U1.B.

Work Space	LCSC	JLCPCB Assembled	System	Follow
	Title(PartNO)	Footprint	SMT	Type
	LT1881AIN8#PBF	<input checked="" type="checkbox"/>  PDIP-8_L10.2-W5.9-P2.54-LS7.6-BL		
	LT1881AIN8#PBF.1			
	LT1881AIN8#PBF.2			
	LT1882MPS#TRPBF	<input checked="" type="checkbox"/>  SOP-14_L8.6-W3.9-P1.27-LS6.0-BL		
	LT1882MPS#TRPBF.1			
	LT1882MPS#TRPBF.2			
	LT1882MPS#TRPBF.3			
	LT1882MPS#TRPBF.4			
	LT1884IS8#TRPBF	<input checked="" type="checkbox"/>  SO-8_L4.9-W3.9-P1.27-LS5.9-BL		
	LT1884IS8#TRPBF.1			

How to create a PCB footprint.

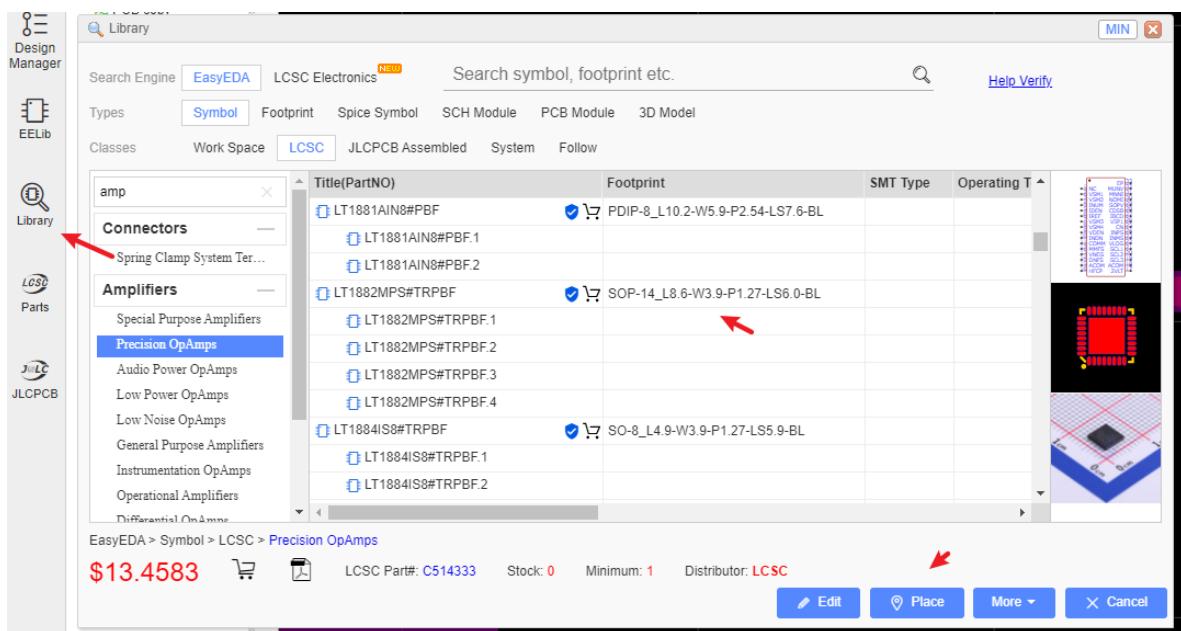


How to change a component's footprint?

please refer at [Footprint Manager](#)

How to find components/parts/libraries?

The component search function has been significantly improved to make finding part symbols and footprints quicker and easier. Press **SHIFT+F** or click on the **Libraries** icon on the left navigation panel:



In the new components dialog, it is easy to select the right components via tags and you can set tags for your own components.

Fabrication and Order

How to Order PCB

1. Before ordering, please check this Gerber first:
<https://docs.easyeda.com/en/PCB/Gerber-Generate/index.html#Gerber-View>
2. Visit and login at <https://lcpcb.com/quote>
3. Add this Gerber file(compressed file) on the page and type the order options
4. Save to Cart, and then submit the payment

If you want to combine the components order with the PCB order at <https://lcsc.com> , please refer:

<https://support.lcsc.com/article/24-do-you-offer-combine-shipment-with-pcbs>

please refer at [Order PCB](#)

How export BOM and order parts?

please refer at [Export BOM](#)

Import and Export

How to import Alitum/Eagle/Kicad File?

Please refer at [Import Altium Designer](#)

Can I export my design?

Yes, you can export your design as EasyEDA format or Alitum Designer format. Please refer at:

[Export EasyEDA format](#)

[Export Altium Designer Format](#)

How to export or print the schematic or PCB?

please refer at [Export](#)

Save and Backup

Where are my files?

Your files are stored on EasyEDA servers, so you can access them anywhere and share them with your partners.

if you using EasyEA desktop client, you can set the running mode as "Project Offline Mode", it will save your project to local.

How to save my file to the local?

You can download the project via :

- Right-click project and download;
- Download the [EasyEDA Source](#)
- Export to Altium file [Export: Export Altium](#)

How to recover the deleted file?

1. Check "Recycle Bin" at the editor bottom-left icon, find and recovery
2. Find it back at "Document Recovery" function. Via: - Advanced - Document Recovery.

I don't like others seeing my design. How can I stop that happening?

Set your project as Private. For extra security you can even save your work locally.
as above to save your file to locad as EasyEDA format.

Is EasyEDA safe?

There are no absolutely secure things in the world but even if you have the misfortune - as happened to one of our team - of losing one laptop and having two hard drives break, EasyEDA will try to protect your designs in following ways:

1. We utilize SSL throughout the entire domain EasyEDA.com. Secure Socket Layer (SSL) technology encrypts all data transferred between your computer and our servers. Your data is for your eyes only.
2. You can save your files locally.
3. Multiple copies of every file are saved in your local database.
4. EasyEDA servers backup your designs frequently.

What if EasyEDA cannot become self sustaining and has to close down?

We promise to do our best to ensure that neither of these things will happen; we have spent so much of our time to get to this point. We promise that if we cannot make enough money out of EasyEDA to keep it alive or to fund further development, we will not simply abandon our baby or our community but we will consider donating the code to the Open Source Community to let them build on our efforts. There are no companies who can stay forever, so if a time comes when we have to close down, we will follow the steps below:

1. Give our users six months warning prior to closure;
2. Ensure all our users can backup their designs;
3. Ensure that user's designs can be exported to some other EDA tools, such as Kicad, Altium Designer and others.
4. Package our codes, so that users can install an EasyEDA in their own OS (Windows, Linux, Mac). Users can then build their own cloud EDA.
5. Upload our codes to github.com and make them open source.

So, nothing will be lost and our users can continue to enjoy an awesome web based EDA tool that lets them stay in charge of their designs: anywhere, anytime and on any OS.

How to backup my project?

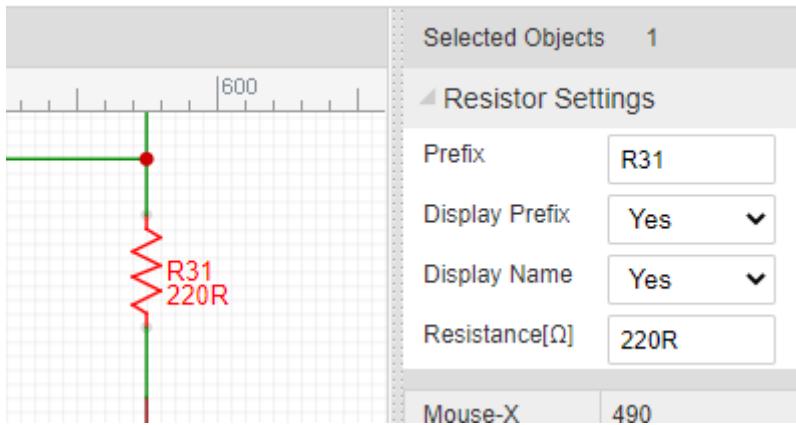
Please refer at: [Saving Your Work Locally](#).

Spice Simulation FAQ

EasyEDA's main target is schematic and PCB, not simulation. EasyEDA only support simple schematics simulation.

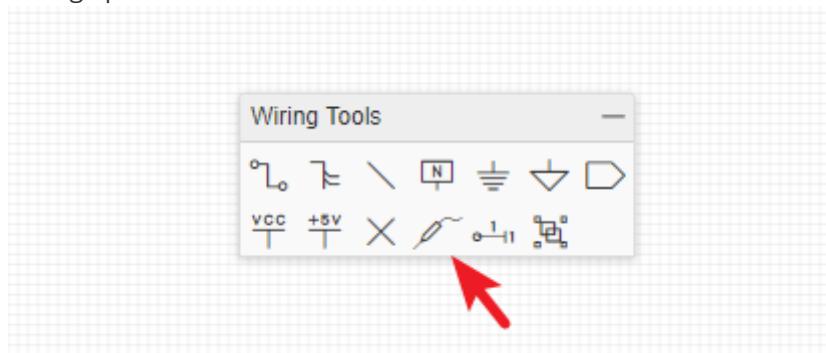
How to set the resistance of a resistor

You can use the name attribute. Just set the name or double click the value text.



Where Can I find the Probe/voltage probe?

Voltage probe



Why I can't simulate my schematic

EasyEDA only has very few simulation models, EasyEDA is powered by LTSpice, please check LTspice to know what can be simulated.

How to ask for help and get an answer

[\[Must read\] How to ask for help and get an answer](#)

Others

Can I use EasyEDA in my company?

You are free to use EasyEDA for individuals, business and education.
If you add our Logo and link on your PCB/Video we will appreciate.

What happens if EasyEDA service is offline for some reason?

EasyEDA can be run as an offline application. You can export your design first, when the service back, you can import the design and save to EasyEDA server.

Or you can use EasyEDA desktop client "Project Offline" mode.

How to find the list of hotkeys.

Please refer at [Introduction: Shortcut Keys](#)

Why does EasyEDA focus on Cloud based EDA?

EasyEDA is built for people who like to work anywhere, who like to build projects together with other team members, who like to share their projects, who like something that operates like a github for hardware design. The only way to meet these needs is to build a Cloud version EDA.

How can I work if there is no internet?

Although most of the time there are ways to access the internet easily and cheaply there may be times when, for whatever the reason, internet access is simply not possible. For times like this, EasyEDA is working to provide a desktop client soon.

Does EasyEDA have a desktop version?

Yes, please refer at: <https://easyeda.com/page/download>

Which Browser is best for EasyEDA?

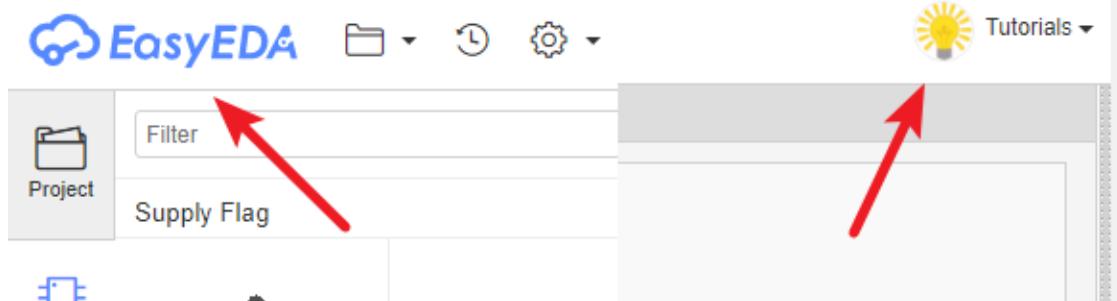
The latest **Chrome** and **Firefox**. If you are restricted to using other browsers, it would be better to download the EasyEDA desktop client.

How to go to your dashboard.

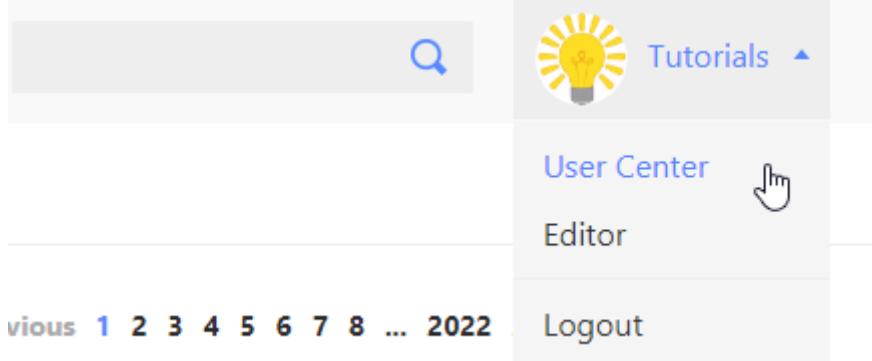
In the [User Center](#), you can check all your Projects, Modules, Libraries and Friends, Messages etc.

There are two ways to arrive there.

1. From the Editor, you can click on user logo:

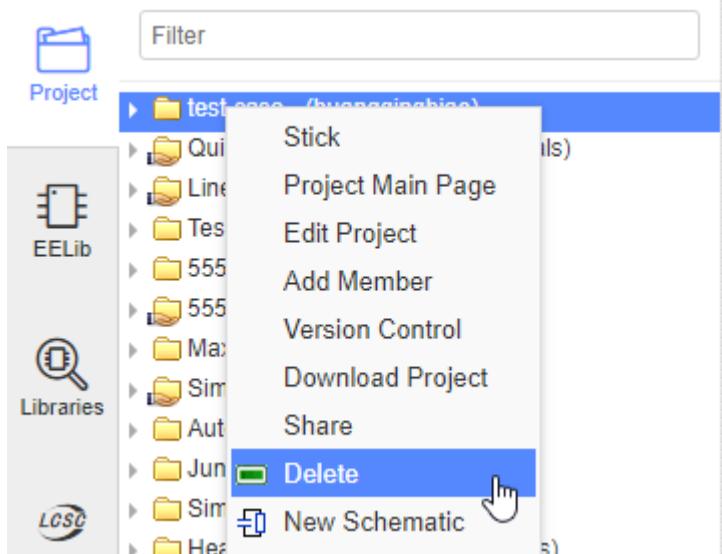


2. From the homepage, you can click My Projects:



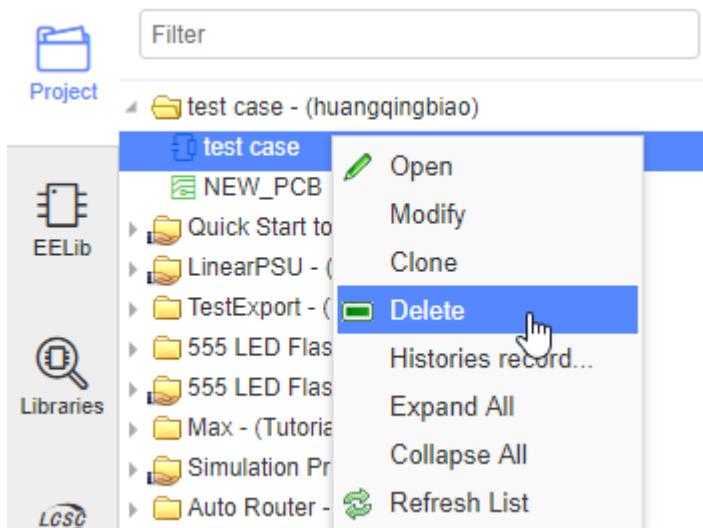
How to delete a project.

Select it and right click to open a context menu, like the image below.



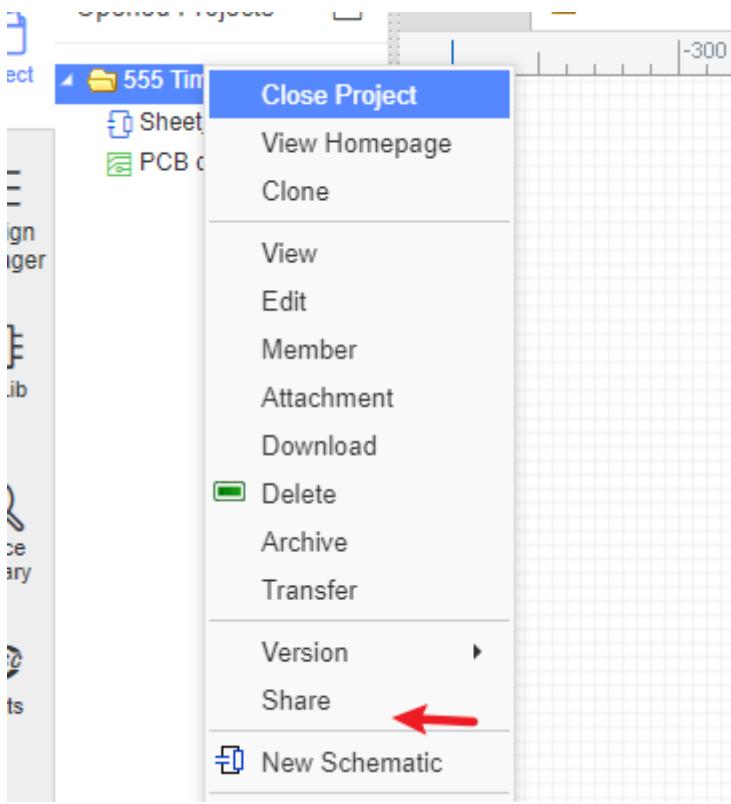
How to delete a schematic or PCB.

Select it and right click to open a context menu, like the image below.



How to share a project with others.

1. Make your project public.



2. To share a project privately with only selected collaborators via:

[Add Member](#)

How to find the version history of schematics and PCBs.

The version history of your EasyEDA schematics and PCBs can be accessed by right-clicking on the file you wish to query to open the context menu as shown in the image below:



Then click on the version number that you wish to view.

Note: *saving a previous version will restore that version to being the current version of the file.*

Does EasyEDA canvas use the Cartesian coordinate system?

Yes and no.

It uses X and Y coordinates where the horizontal X coordinate is positive to the right of the origin and negative to the left but the vertical Y coordinate is positive **below** the origin and negative above it.

Actually, we think our coordinate system is not very good but it is hard to change.



How to update editor to latest version and how to remove editor cache?

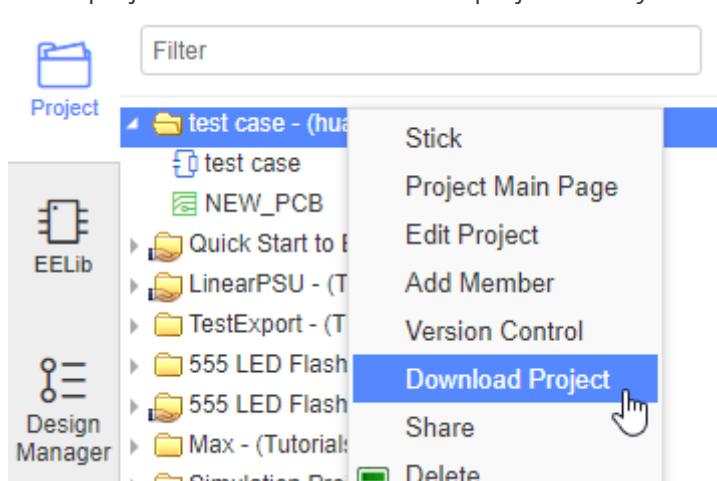
please refer at: [How to Update](#)

Essential checks before placing a PCB order

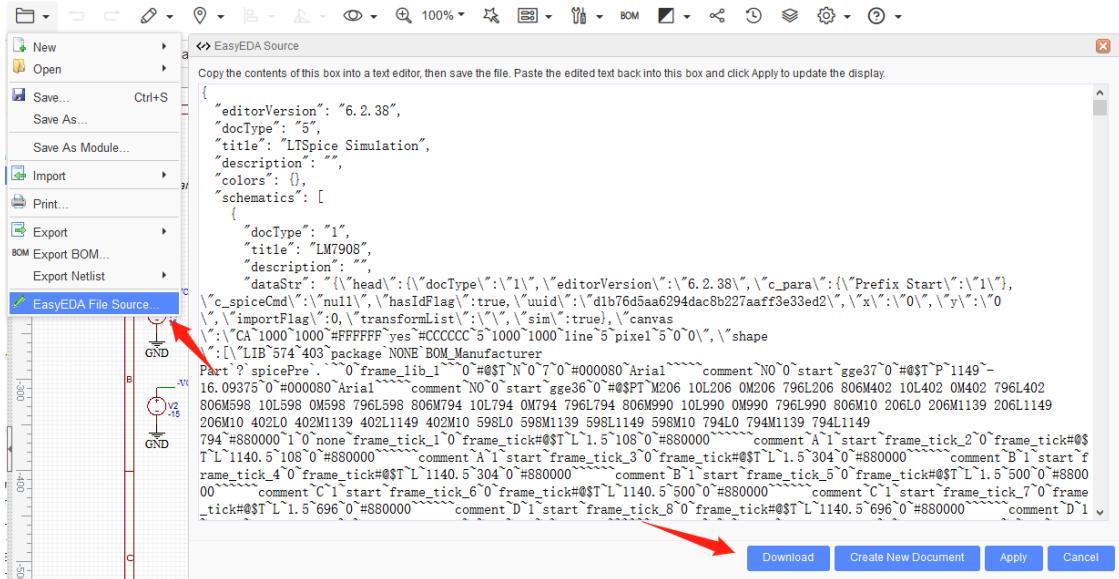
Please refer [Essential checks before placing a PCB order](#)

Keep in Mind

1. After the first save of any file, EasyEDA will back up all saved files automatically under the [Version Control](#). If you want to back up your files locally, you can download a copy of the whole project or of individual files in a project in EasyEDA Source (JSON) format:



and File > EasyEDA File Source > Download



2. If you need help, you can contact us email or ask via our [Support Forum](#); we will respond ASAP.

support@easyeda.com

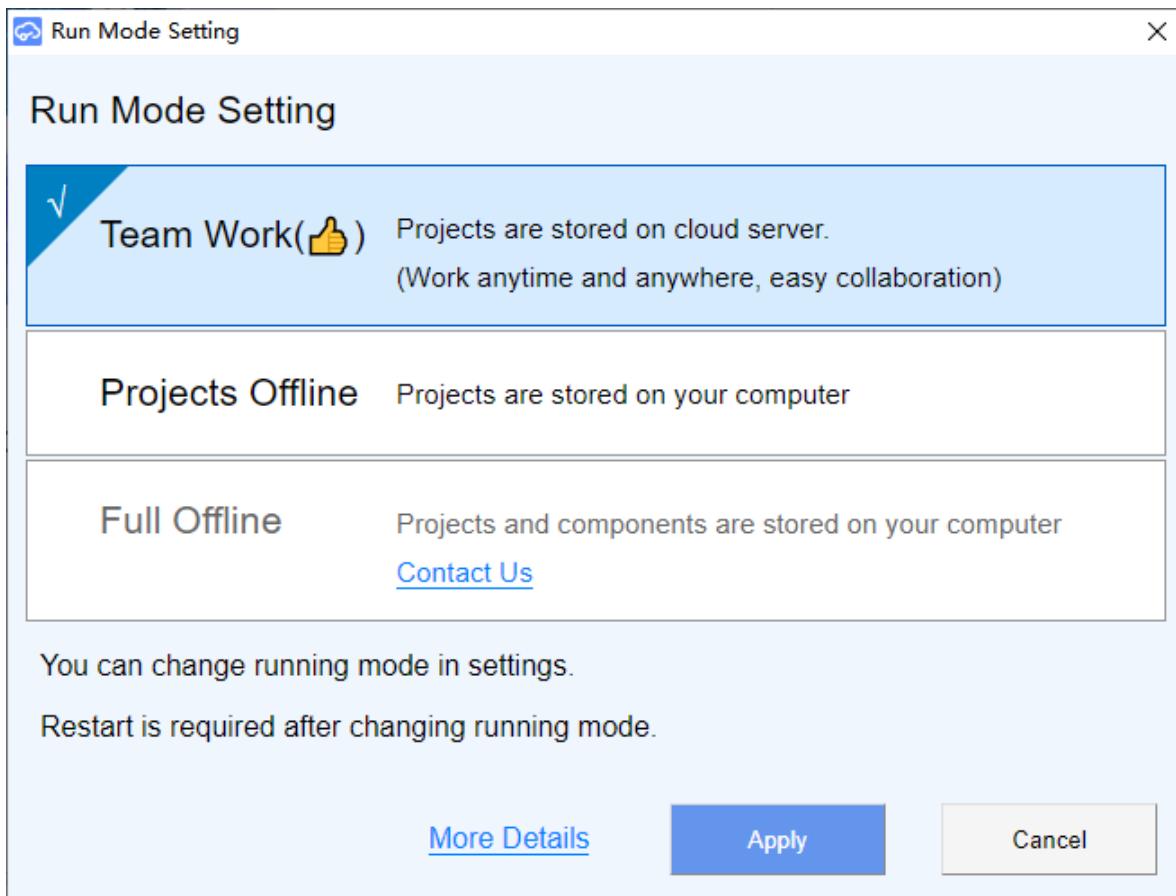
EasyEDA Desktop Client

Download

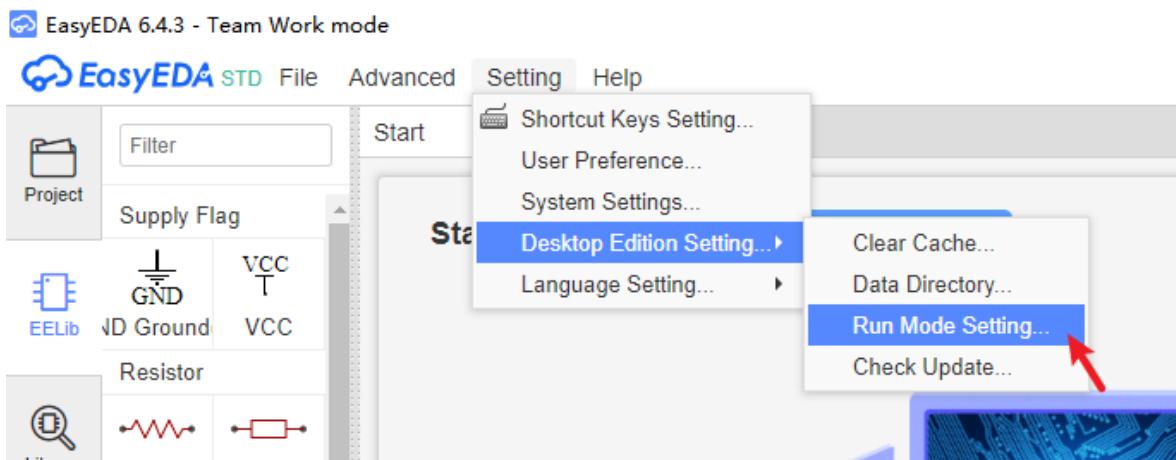
Download address: <https://easyeda.com/page/download>

Client Running Mode

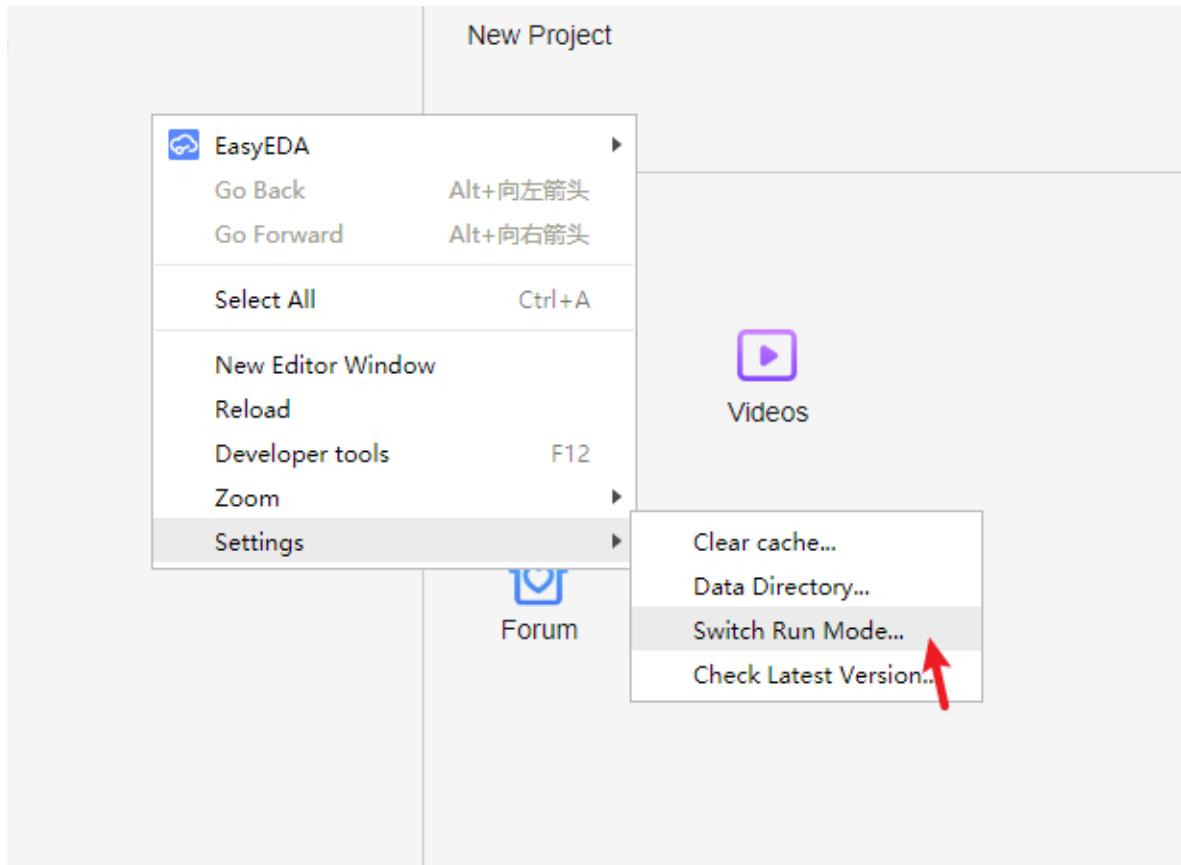
When you install the client first time, you can set the running mode:



if you want to change the running mode after installation, you can via: Top Menu - Setting - Desktop Client Setting - Running Mode Setting



or right-click the start page, via: Setting - Switch Run Mode



Team Work Mode

This version is full function, such as team work, work any time any where.
Project and library are saving at cloud server.

Projects Offline Mode

Project save at local, the library save at the cloud. Only few option needing internet, such as: library searching, library saving, schematic convert to PCB, import changes, etc.

Full Offline Mode

Doesn't provide yet.

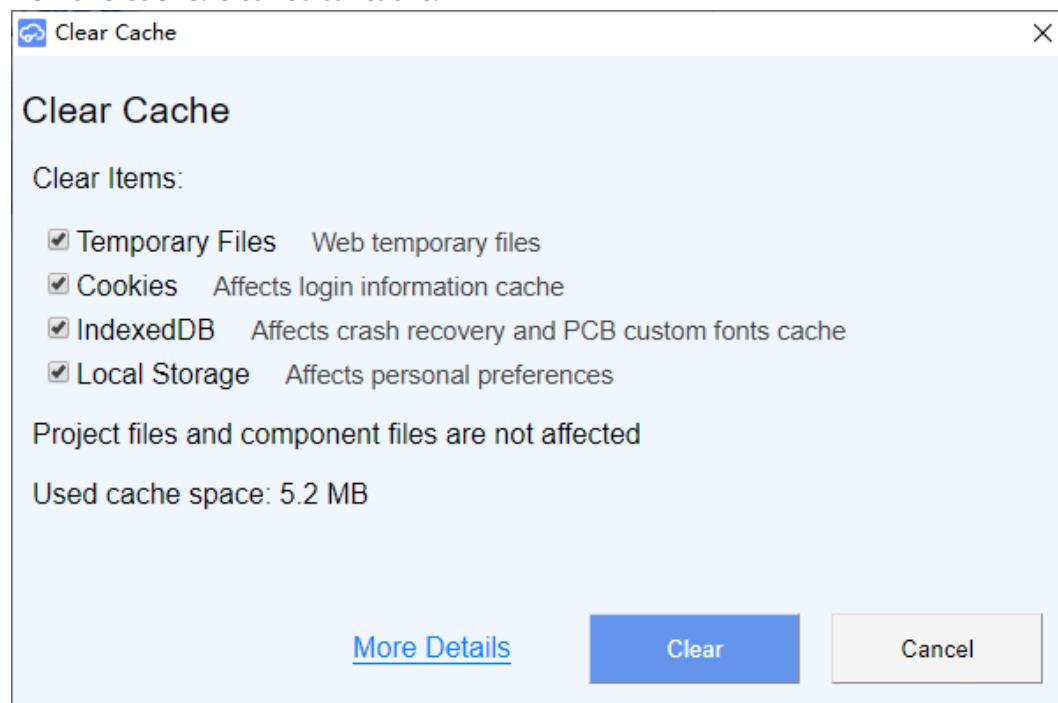
Projects and libraries are saving at local. It is only provide for company. That will take some cost.

Client Setting

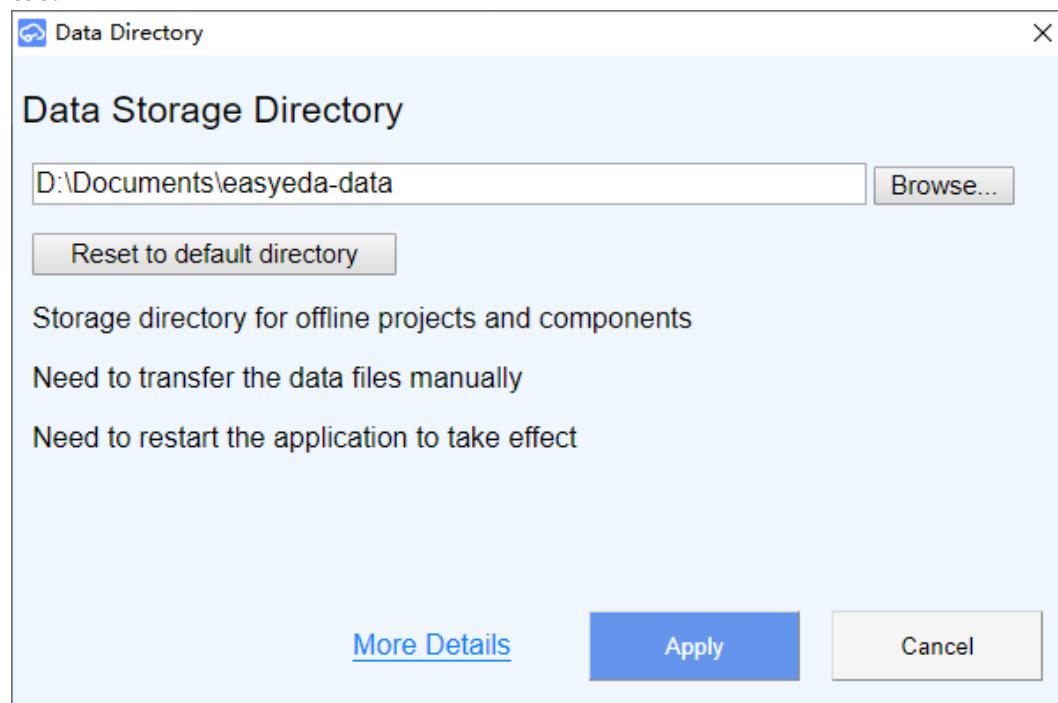
Right-click the start page, visit the setting menu. or at: Top Menu - Setting - Desktop Client Setting

- **New Editor Window:** Create a new editor window.
 - **Reload:** You can reload the editor.
 - **Zoom:** Zoom in or zoom out the editor windows.
 - **Setting:**

- **Remove Cache:** Clear editor cache.



- **Data Saving Directory:** Including offline projects and auto-backup projects saving directory. When you using "Team Work Version", the client will auto-backup your file to this directory, which is named "projects_backup", each signle file you saved will be saving in this directory, if you want to recovery the file from this directory, you can open the backup file at the editor. Or you can use editor Document Recovery function too.

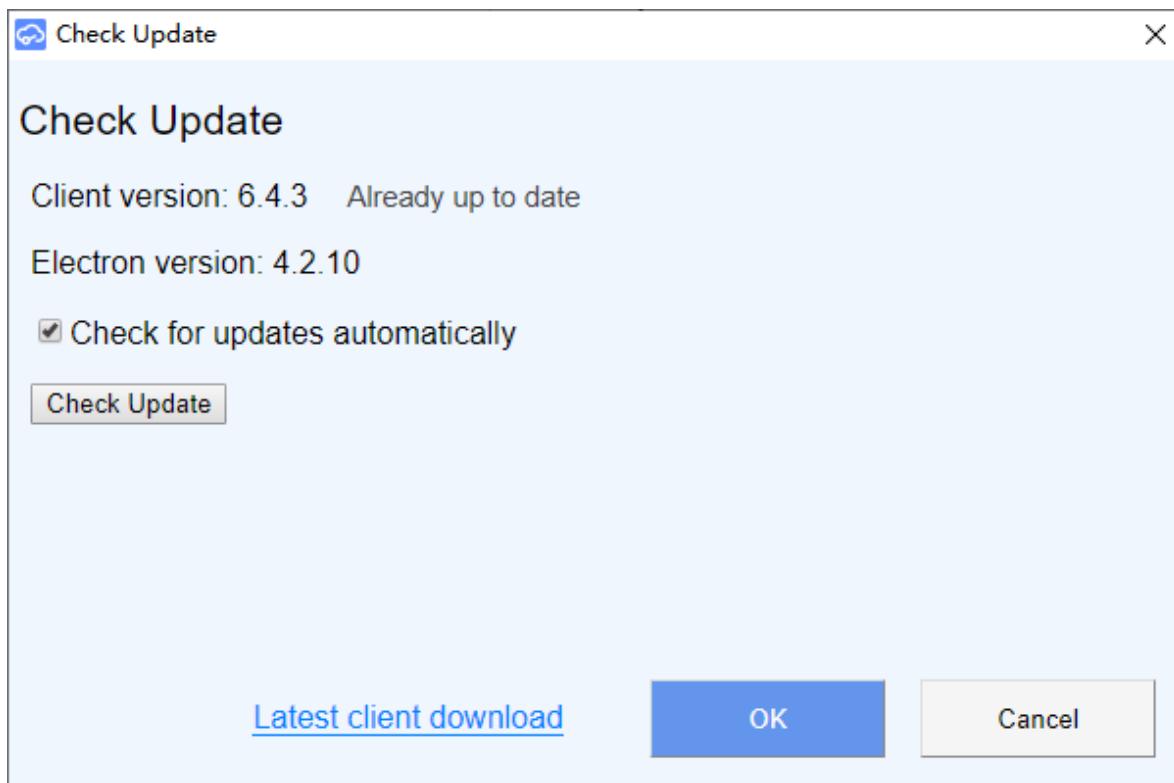


名称	修改日期	类型	大小
components	2020/3/9 17:22	文件夹	
projects	2020/3/9 19:20	文件夹	
projects_backup	2020/8/3 19:15	文件夹	

- ```

1 | - **version setting**: Modify the running mode you need.
2 |
3 | - **Check Update**: Check the client version.

```



How to import online project into project offline version in batch?

1, first download project backup to local: [backup project](#)

2, after downloading and then decompression, to get the projects separate compression files, each compression file separately decompression in a folder.

3, copy the unzipped project folder to the offline project save directory.

4, then open the client, the client will automatically scan the newly added directory to generate a list of projects.

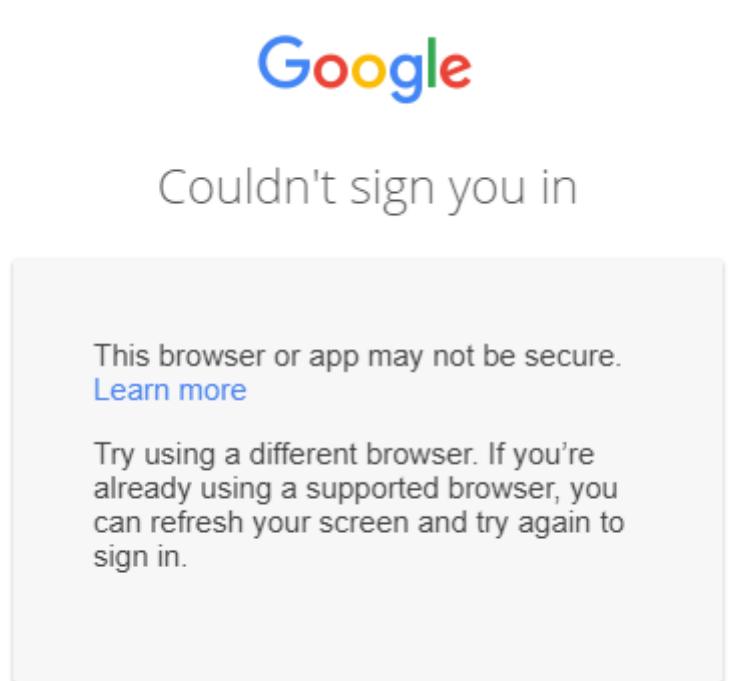
Note:

- Please do not directly modify the name of the document in the folder in explorer, and do not directly copy and paste the new document in the folder, otherwise the editor will not be able to properly recognize the newly added document. Please go through the editor " - File - EasyEDA File Source..." Proceed to add new documents into the project.
- The too old client version doesn't allow to use, the dialog will tell you the version is expired, please download the new version to install.
- Doesn't support to upgrade automatically yet, please download and install manually.

## Known Issue

1. When client running mode is "Project offline", it doesn't support to open the public project at Explore, and can not open the cloud project too.

2.If you login with Google account, it will show the client is not secure, please refer at this post  
[Can't login via Google Account](#)



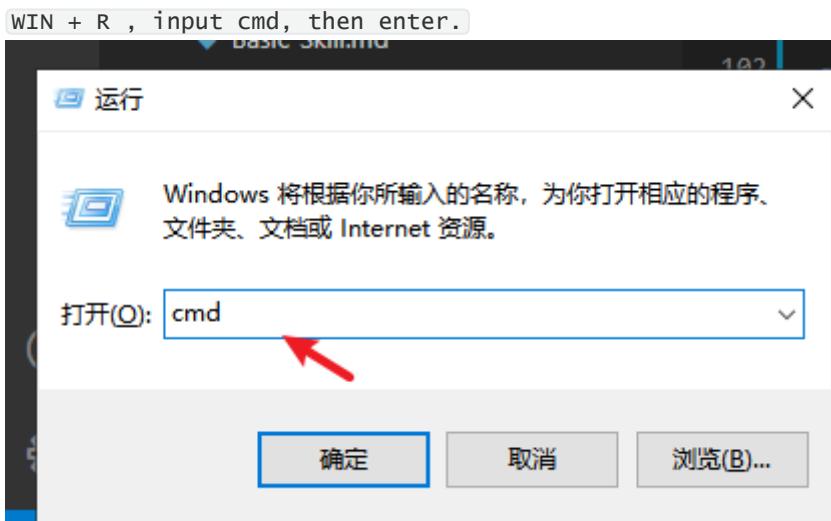
Please reset your password to get the password [Reset Password](#)

- 1.Hit password reset link above
- 2.enter gmail email address and hit reset, keep track of your new password
- 3.now log in "normally", typing in your gmail address and password, not hitting the "login with Gmail" button.
- 4.this issue is Google block other browsers, you can search this issue at Google

3.Windows: Some windows systems can not run EasyEDA client well, or some PCs need some times to loading the login page,

if you met the dialog blank screen all the time when open the client, please try below steps:

- Close client
- 1.Open CMD window dialog by administrator



- 2.Input this at cmd window: `netsh winsock reset`

C:\WINDOWS\system32\cmd.exe  
Microsoft Windows [版本 10.0.18363.959]  
(c) 2019 Microsoft Corporation. 保留所有权利。  
C:\Users\ASUS>netsh winsock reset.

- 3.Enter
- 4.Open client again. Maybe need to restart the computer.

4.Linux OS: Show segement fault while runing the client. That is system capability issue, any chance to upgrade the OS version.

5.Mac OS: Can't install isse: [How to open apps from unidentified developers on macOS Sierra](#)

## How to Update

---

### Version Rule

---

EasyEDA version number is

`ReleaseCountsOfThisYear.MajorVersion.ReleaseCountsOfThisMajorVersion`. For example, v4.9.3 is the fourth year released of EasyEDA, and nine major versions are released in this year, EasyEDA had released 3 times in this major version.

### Version Upgrade

---

If you use EasyEDA online, it can seamlessly upgrade by itself. However,EasyEDA uses an App Cache technique to allow you to use EasyEDA offline ([W3C HTML5 Offline Web Applications](#)) which may delay the automatic upgrading process. Therefore, if you want to upgrade to the latest version immediately, you can follow the two simple steps below.

1. Check the About... dialog;
2. If the Built Date is older than 2017/06/01:  
Close your browser open EasyEDA again.  
If the Built Date is still showing older than 2017/06/01:  
Close your browser and open EasyEDA again.  
If the Built Date is at or newer than 2017/06/01, you don't need to do anything.

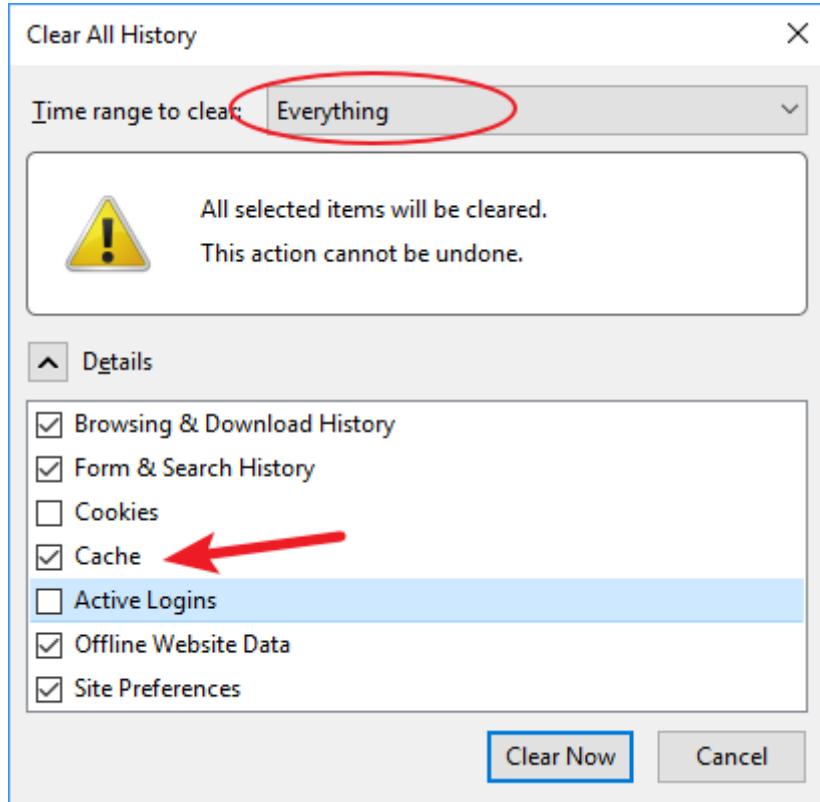
**Note:** 2017/06/01 is just an example.

If those two steps don't work, you may need to clear your browser's cache:

### Mozilla Firefox

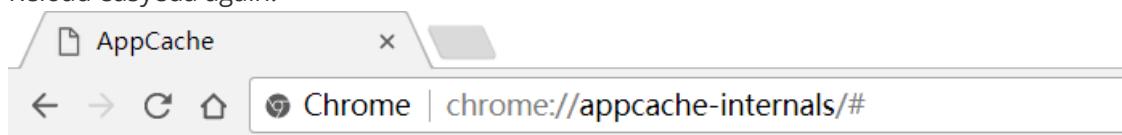
- Close the editor, Go to "Preferences... > Privacy & Security > History > clear your recent history" or use **Ctrl+shift+Delete**,
- Click on "Clear now",

- Reload easyeda again.



## Chrome

- Close the editor, Open the following URL: `chrome://appcache-internals/`.
- Look for `easyeda.com` and click "Remove".
- Reload easyeda again.



## Application Cache

Instances in: C:\Users\AppData\Local\Google\Chrome\User Data\Default (2)

<https://easyeda.com/>

Manifest: <https://easyeda.com/editor.appcache>

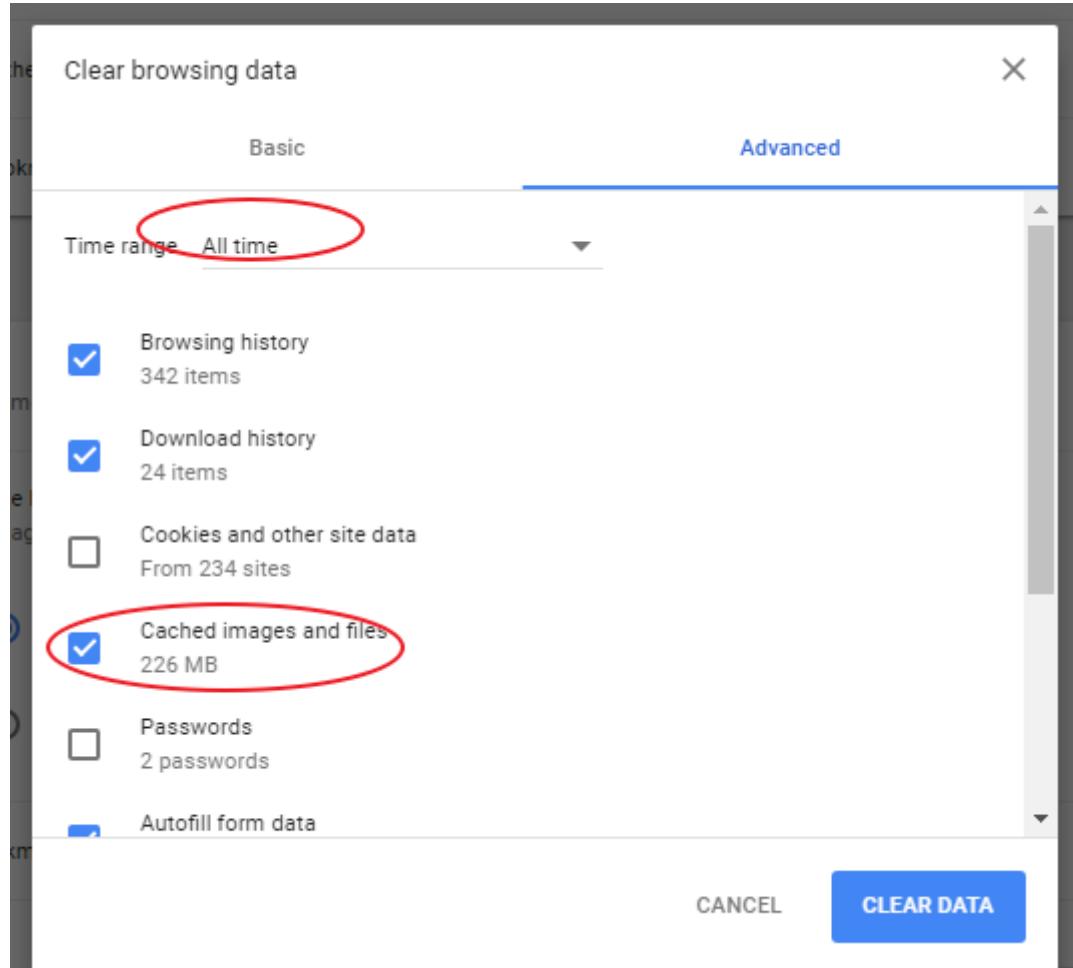
Size: 3.5 MB

- Creation Time: Tue Jun 13 2017 12:43:57 GMT+0800 (中国标准时间)
- Last Access Time: Tue Jun 13 2017 14:46:52 GMT+0800 (中国标准时间)
- Last Update Time: Tue Jun 13 2017 12:43:57 GMT+0800 (中国标准时间)

[Remove Item](#)

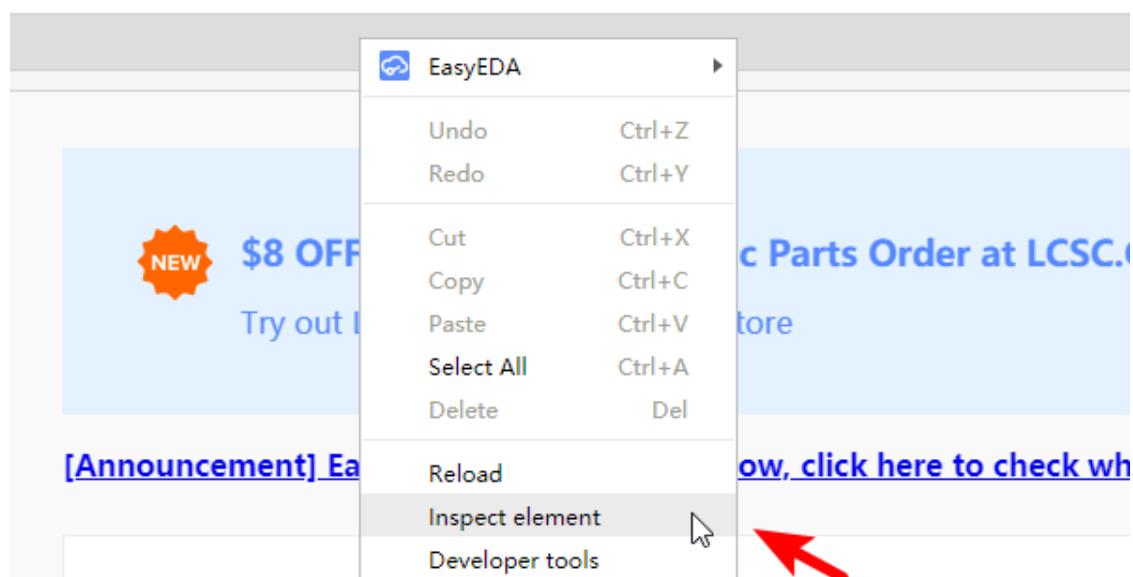
[View Details](#)

- Or you can use hotkey **Ctrl+shift+Delete** to delete Chrome caches.



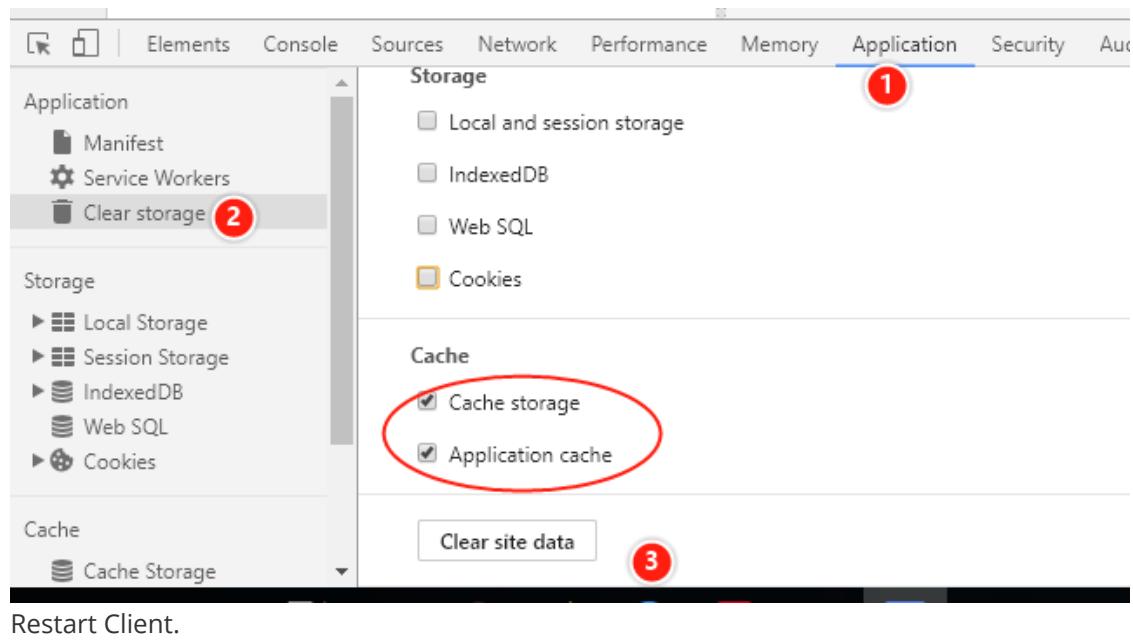
## Desktop Client

- Close client and re-open it.
- If doesn't work, right-click start page, use "Inspect Element".



[What's new in latest version NEW](#)

- Switch to "Application" - "Clear storage", enable "Cache storage" and "Application cache", then click "Clear site data".



- Restart Client.

## User Center FAQ

---

### How to change password

---

Via: User Center > Account > Password Setting

### How to recover the deleted file

---

Via: User Center > Recycle Bin

Find the file you want and then recover it.

### How to transfer project or library to the team

---

Transfer project: Enter the project, via: Setting > Advance Setting > Transfer Project

Transfer library: Move the mouse to the library, and then click the transfer icon.

### How to delete project

---

Via Project > Manage > Setting > Advanced > Delete Project

## Contact Us

---

## Contact

---

## PCB Order Problems:

---

- [support@jlpcb.com](mailto:support@jlpcb.com)
- At present, EasyEDA PCB service is transfer to [JLCPCB.com](http://JLCPCB.com), we are the same company group, any PCB orders problem please contact with [JLCPCB](http://JLCPCB).

## Parts Order Problems:

---

- [support@lcsc.com](mailto:support@lcsc.com)
- EasyEDA provides direct links to [LCSC](http://LCSC) thousands of components. Please order at [LCSC.COM](http://LCSC.COM), any parts order problem please contact with [LCSC](http://LCSC).

## All Other Inquiries About EasyEDA:

---

- Tutorials: [EasyEDA tutorial](#)
- User forum: [EasyEDA forum](#)
- If you met some problems with design, please attach your design by [EasyEDA source file](#) and how to repeat the issue.

[support@easyeda.com](mailto:support@easyeda.com)

## Notice

---

*EasyEDA team may not have the time or resources to help you fix all your problems; we may just be able to help you to fix problems commonly encountered by newbies, such as using a drawing polyline in place of a wire, finding a spice model for a simulation or selecting the right PCB footprint.*

- *Please note that although some browsers or plug-ins allow you to use gestures, EasyEDA does not work with gestures, so you should disable this function.*
- *Simulation editing is not yet fully supported: care must be taken because the last save by any collaborator overwrites all previous saves.*
- *It can also find the value text but it cannot step through multiple components with the same value.*
- *Take a few moments to think about your username because this is the name that other users will see on your designs and posts if you choose to share them or make them public. Once you have created an account, you cannot change your username.*
- *You can use upper and lower case letters, numbers and symbols to make a strong password but don't forget that the password entry is case sensitive.*
- *Except ordering of PCBs directly from EasyEDA.*
- *If you always open EasyEDA in the same browser on the same machine, your Anonymous files will appear under the Anonymous Files folder in the left hand panel but you should not rely on this as a way of keeping track of Anonymous files.*

# **Business Development/Cooperation About EasyEDA:**

---

please contact

[dillon@easyeda.com](mailto:dillon@easyeda.com)

## **Address:**

---

- F5, Tianjian Building, No.7 Shangbao Road, Futian District, [Shenzhen](#), Guangdong, 518000, China



# **Introduction to EasyEDA**

---

Welcome to EasyEDA, a great web based on EDA(Electronics Design Automation) tool for electronics engineers, educators, students, makers and enthusiasts.

There's no need to install any software. Just open EasyEDA in any HTML5 capable, standards compliant web browser.

Whether you are using Linux, Mac or Windows, Highly recommend to use Chrome or Firefox as your browser ,you can also download [EasyEDA client](#).

EasyEDA has all the features you expect and need to rapidly and easily take your design from conception through to production.

### **EasyEDA Editor:**

<https://easyeda.com/editor>

### **Instruction:**

- This tutorial document will be updated according to the updated EasyEDA editor.

### **Tutorial for PDF**

[EasyEDA-Tutorials.pdf](#)

### **EasyEDA Provides:**

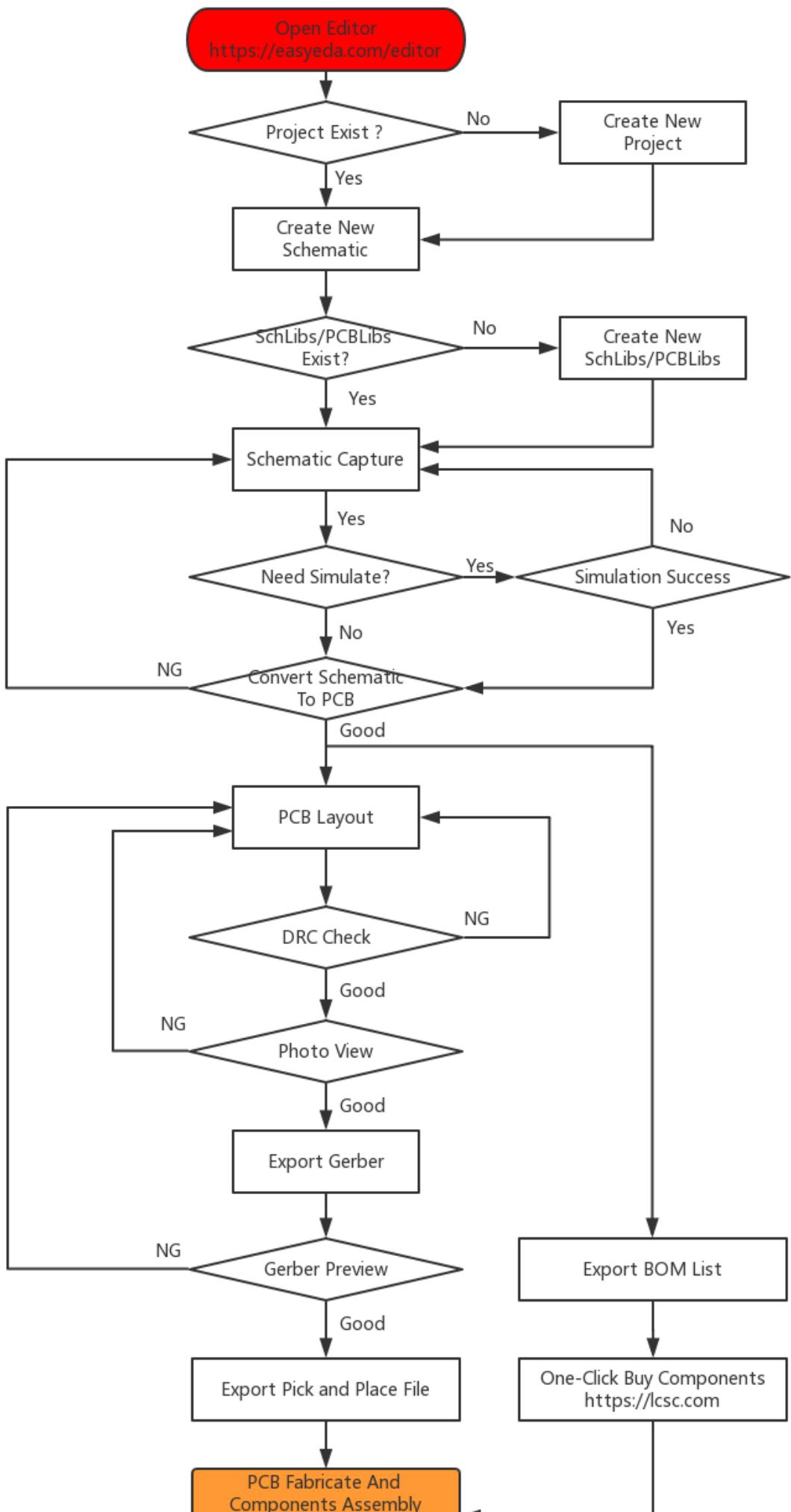
- Simple, Easier, Friendly, and Powerful general drawing capabilities
- Working Anywhere, Anytime, Any Devices
- Real-time Team Cooperation
- Sharing Online
- Thousands of open source projects

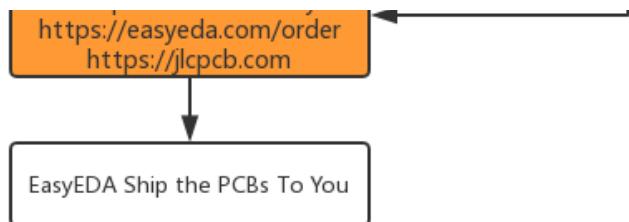
- Integrated [PCB fabrication](#) and [Components purchase](#) chain
- API provide
- Script support
- Schematic Capture
  - [LTSpice-based](#) Simulation
  - Spice models and subcircuits create
  - WaveForm viewer and data export(CSV)
  - Netlist export(Spice, Protel/Altium Designer, Pads, FreePCB)
  - Document export(PDF, PNG, SVG)
  - EasyEDA source file export(json)
  - Altium Designer format export
  - BOM export
  - Mutil-sheet schematics
  - Schematic module
  - Theme setting
  - Document recovery
- PCB Layout
  - Design Rules Checking(DRC)
  - Mutil-Layer, 6 copper layer supported
  - Document export(PDF, PNG, SVG)
  - EasyEDA source file export(json)
  - Altium Designer format export
  - BOM export
  - DXF export
  - Photo view
  - 3D View
  - Generate the fabrication file(Gerber)
  - Export Pick and Place file
  - Auto Router
  - PCB module
  - Document recovery
- Import
  - Altium/ProtelDXP ASCII Schematic/PCB
  - Eagle Schematic/PCB/Libraries
  - KiCAD Schematic/PCB/Libraries
  - DXF
- Libraries
  - More than 1000,000 public Libraries(Symbol and Footprint)
  - Libraries management
  - Symbol/Subpart create and edit
  - Spice symbol/model create and edit
  - Libraries management
  - Footprint create and edit

# Design Flow by Using EasyEDA

---

You can create circuits design easily by using EasyEDA. The design flow as below:





# UI Introduction

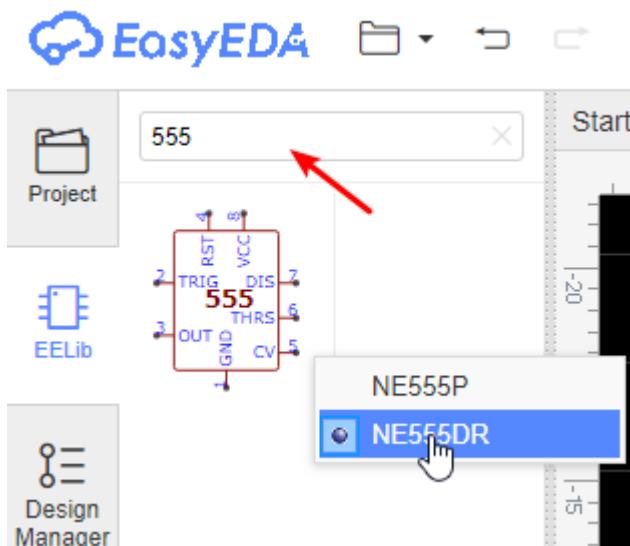
---

EasyEDA Editor has a clearly and friendly user interface. You can use its every function very easily when you become familiar with EasyEDA.

## Filter

---

Before using the Filter, you need to select what module you need in the left navigation panel, and then you can find projects, files, parts and footprints quickly and easily just by typing a few letters of the title. For example, if you want to find all files containing “NE555” in the title, just type “555”, it is non-case-sensitive.



The Filter could only find projects, files and part titles and names. It does not support the Descriptions and Content fields.

Click the X to clear the filter.

## Navigation Panel

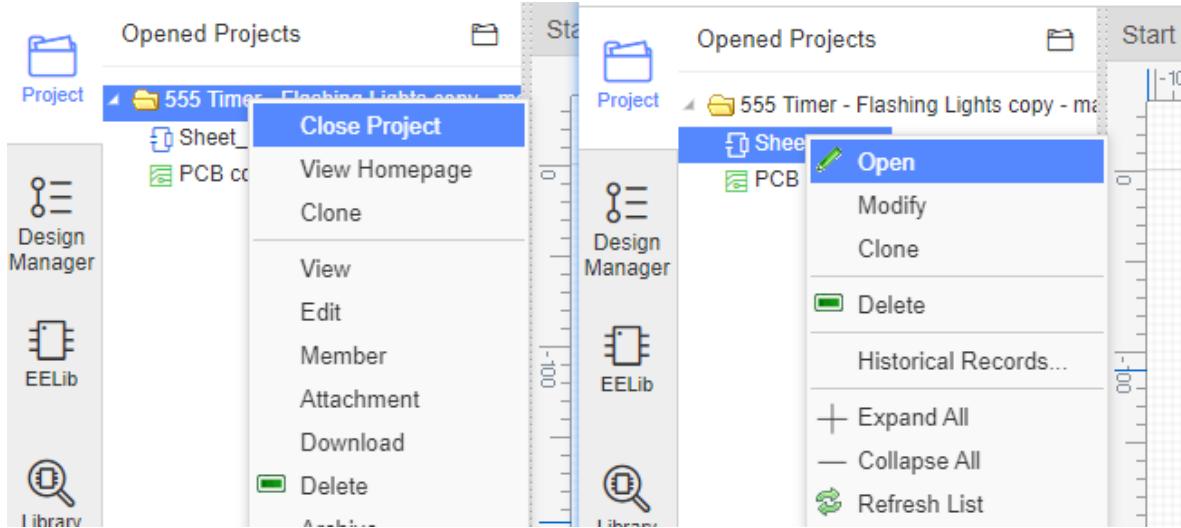
---

The Navigation panel is very important for EasyEDA: The part that you can find all your projects, files, parts and footprints.



## Project

Here, You can find all of your projects that are private or shared with the public, or fork from someone else's. These options have a content menu when you drop down to Projects and right click an item, you will get a tree menu like :

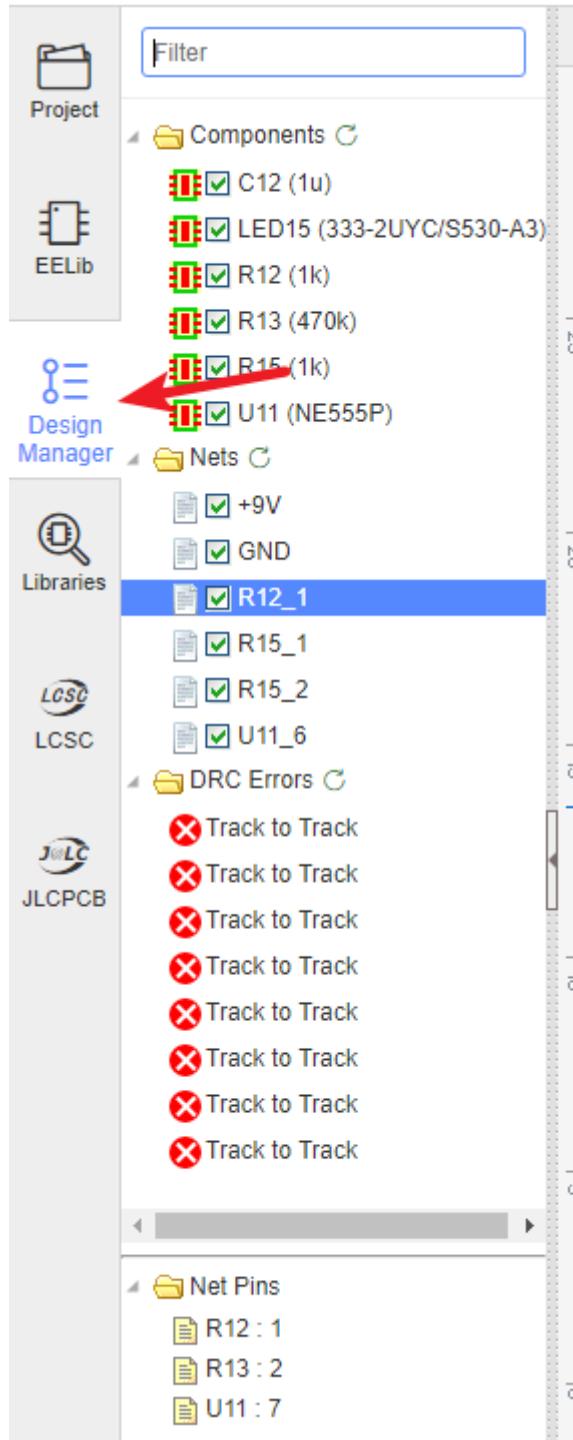


## EELib

EElib means EasyEDA Libraries, It provides lots of components completed with simulation models, many of which have been developed for EasyEDA to make your simulation experience easier.

## Design Manager

Design Manager, you can check each component and net easily, and it will provide DRC(Design rule check) to help your design better.



## Library

Contains schematic symbols and PCB footprints for many available components and projects and your own libs and modules will show up here.

The screenshot shows the EasyEDA component library search interface. On the left, there's a sidebar with icons for Design Manager, EELib, Library, Parts, and JLCPBCB. The 'Library' icon has a red arrow pointing to it. The main area has tabs for Search Engine (EasyEDA), LCSC Electronics, and a search bar for 'Search symbol, footprint etc.'. Below the search bar are tabs for Types (Symbol, Footprint, Spice Symbol, SCH Module, PCB Module, 3D Model) and Classes (Work Space, LCSC, JLCPBCB Assembled, System, Follow). The 'Work Space' tab is selected. The main content area shows a table of components under the heading 'amp'. The first row is highlighted in blue and labeled 'NCP1117ST18T3G'. To the right of the table are two small preview images: a schematic diagram and a PCB layout. At the bottom, there are buttons for Edit, Place, More, and Cancel.

- **LCSC**

If you want to buy components to finish your PCBA, you should try the **LCSC** module, LCSC.com and EasyEDA are the same company.

EasyEDA partners with China's largest electronic components online store by customers and ordering quantity launch <https://lcsc.com>.

LCSC means Love Components? Save Cost! We suggest to our users to use LCSC parts to design. Why?

- Small Quantity & Global Shipping.
- More Than 25,000 Kinds of Components.
- All components are genuine.
- It is easy to order co after design.
- You can save 40% cost at least.
- You can use our components' symbols and footprint directly in EasyEDA editor.

- **JLCPBCB**

JLCPBCB.com, LCSC.com and EasyEDA are the same company group. <https://jlcpcb.com>

More than 200,000 customers worldwide trust JLC, 8000 + online orders per day, JLCPBCB (Shenzhen JIALICHUANG Electronic Technology Development Co.,Ltd.), is the largest PCB prototype enterprise in China and a high-tech manufacturer specializing in quick PCB prototype and small-batch production. Affordable, series quality boards fully manufactured in China. Fully e-tested. Transparent pricing.

## Top Menu

A most of EasyEDA features can find out at top menu:



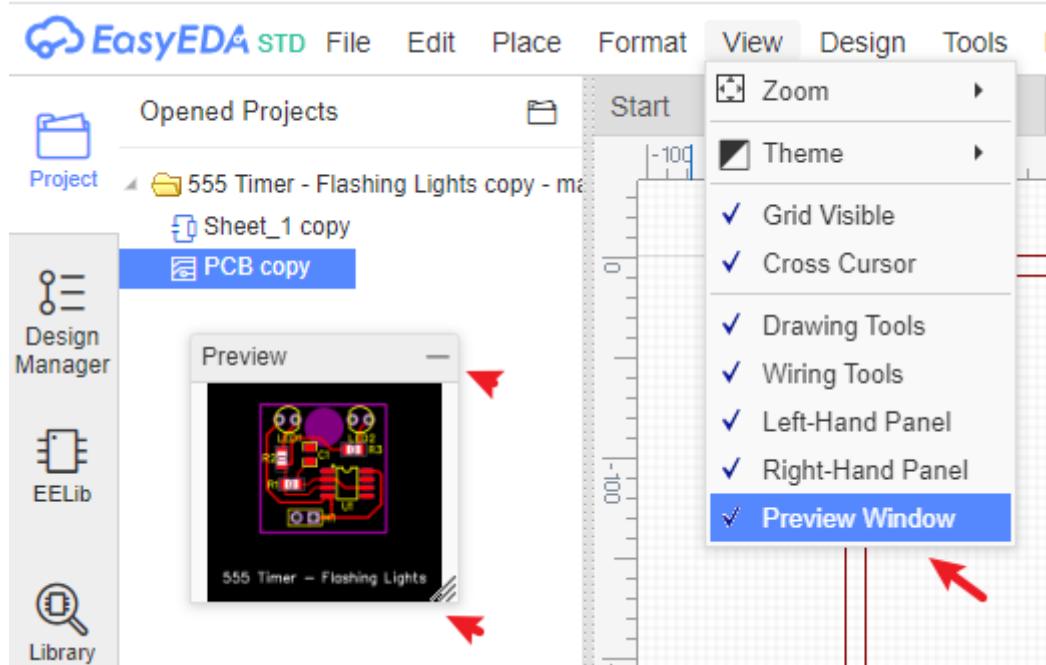
You can find what you need easier and clearly.

## Preview Dialog

The Preview dialog will help you choose components and footprints and can help you to identify schematics and PCB layouts.

You can close or open this dialog via:

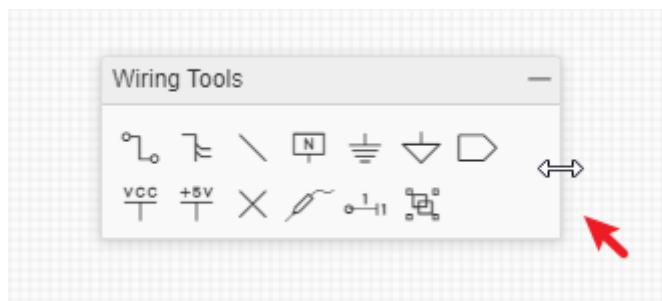
**Top Menu > View > Preview Window.**



- The Preview Dialog has a resizing handle in the bottom right corner.
- The Preview Dialog can't be closed but double clicking on the top banner will roll up the panel or you can click the top right corner . Double clicking top banner again toggles it back to the selected size.

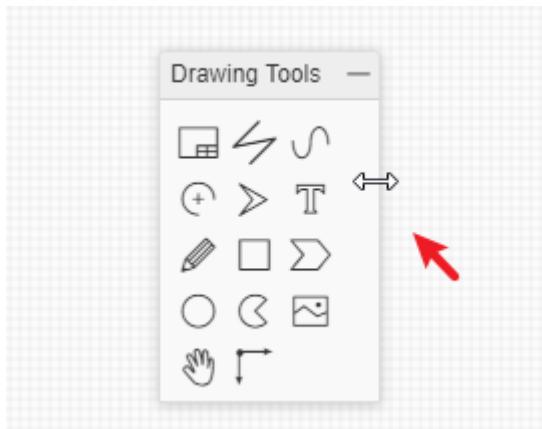
## Wiring Tools

Wiring Tools are document type sensitive: different document types have different tools.



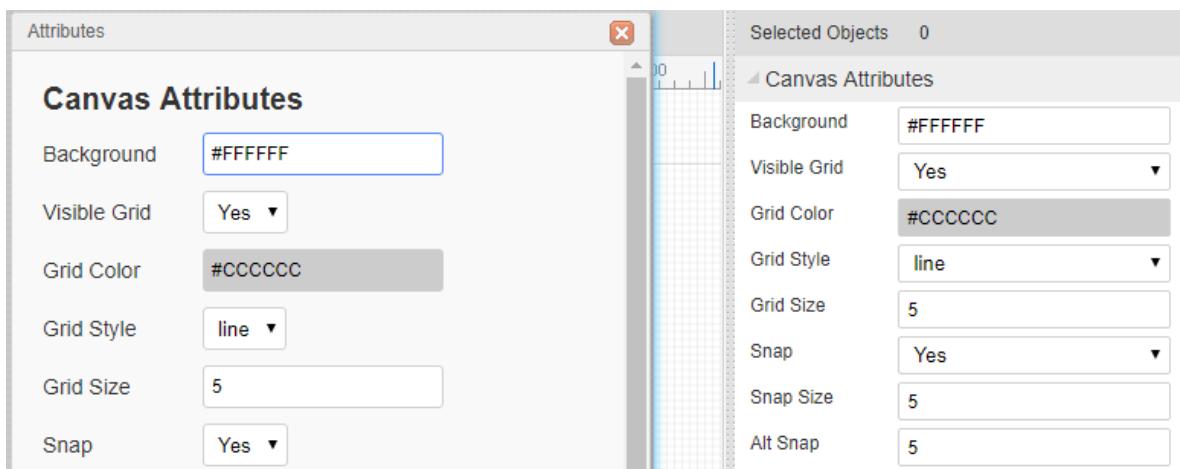
## Drawing Tools

To keep EasyEDA's UI clean and sharp, the Wiring and Drawing tools palettes can be resized horizontally, rolled up or hidden so if you want to focus on drawing, you can roll up or hide the others to make more space and reduce the clutter.



## Canvas Attributes

You can find the canvas Properties setting by clicking on any of the blank space in the canvas.

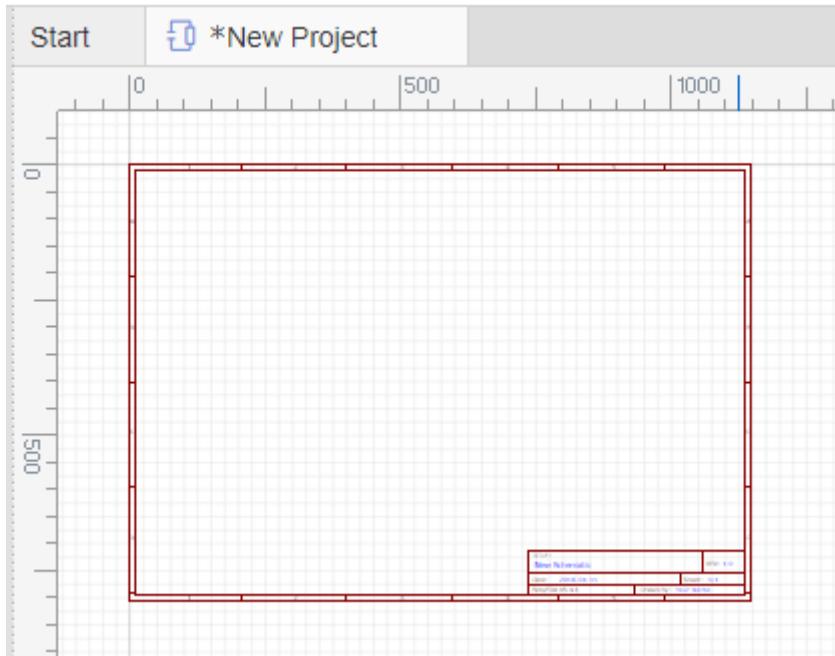


Background and grid colors and the style, size, visibility and snap attributes of the grid can all be configured.

The canvas area can be set directly by the Width and Height or from available preset frame sizes.

## Canvas

This is where it all happens! This the area where you create and edit your schematics, PCB layouts, symbols, footprints and other drawings, run simulations and display WaveForm traces.

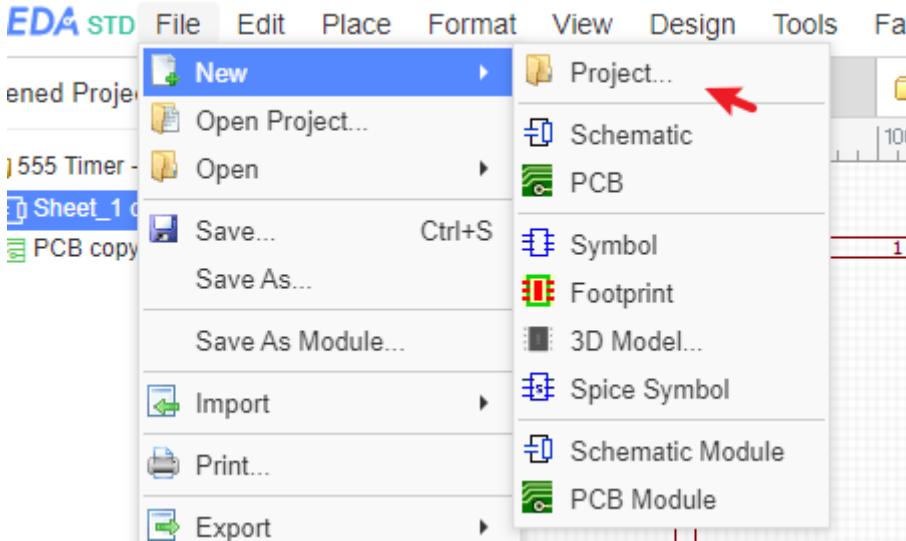


# How to Create a New Project or File

## Create Project

After logging in, you can create a new project:

**File > New > Create a new project/Schematic..etc**



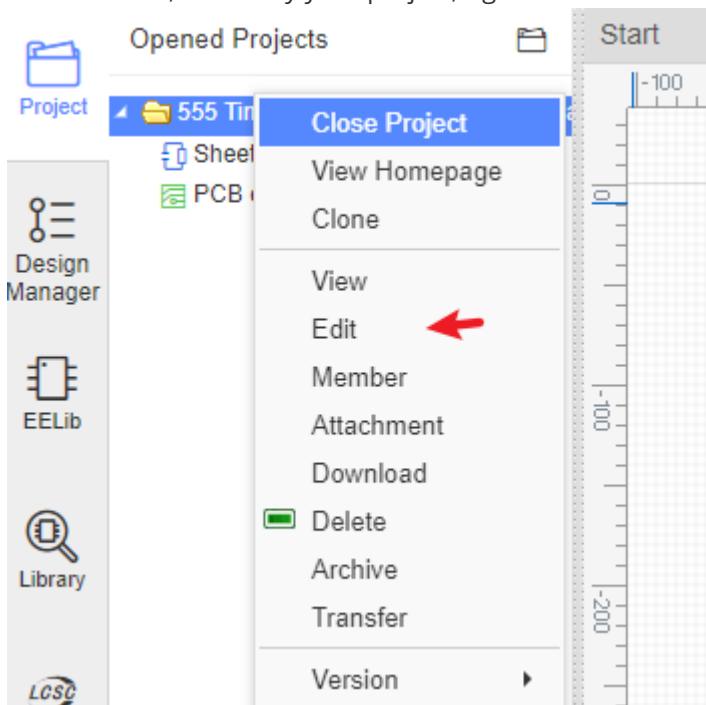
The Project concept is important in EasyEDA because it is the foundation of how to organize your designs.

New Project

|                                                                             |                                  |                             |
|-----------------------------------------------------------------------------|----------------------------------|-----------------------------|
| Owner:                                                                      | UserSupport                      | <a href="#">Create Team</a> |
| Title:                                                                      |                                  |                             |
| Path:                                                                       | https://easyeda.com/UserSupport/ |                             |
| Description:                                                                |                                  |                             |
| <input type="button" value="✓ Save"/> <input type="button" value="Cancel"/> |                                  |                             |

- **Owner:** You can change the owner of this project, you can change the owner to the team if you have joined.
- **Title:** Give it a title: this will show in the project tree in the left hand panel.
- **Path:** EasyEDA allows you set the path for the project, if you want to share with your friend, it will be useful. It can't be editable when it is created.
- **Description:** Adding a short description helps you and anyone you are sharing this project with understand what the project is about.

Once created, to modify your project, right click on it in the project tree in the left hand panel:



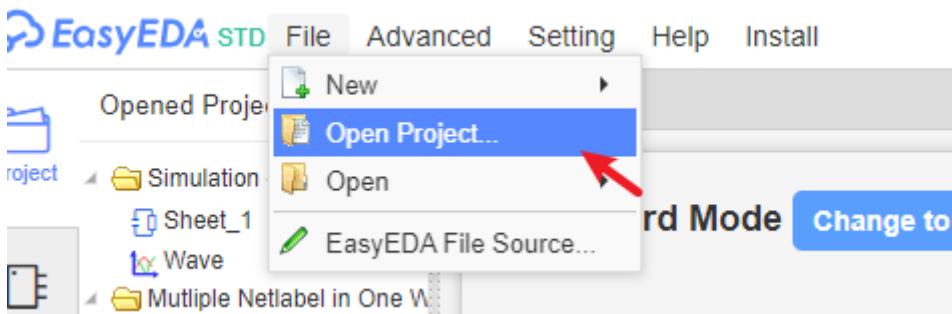
then will open a web page in which you can edit your project:

From here, you can change the publish or not, allow other people to comment on your project and type a more detailed description of the project content. To help you make your project stand out or to maybe simply make a detailed description of your project easier to read, you can use Markdown syntax.

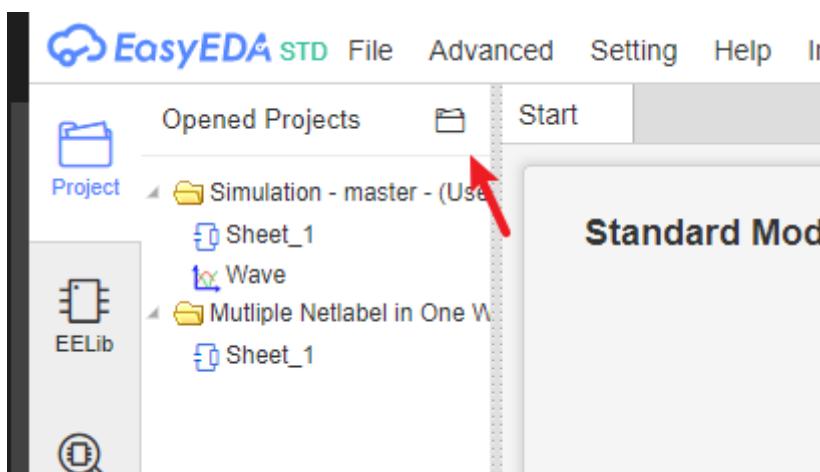
## Open Project

You can open your created project via:

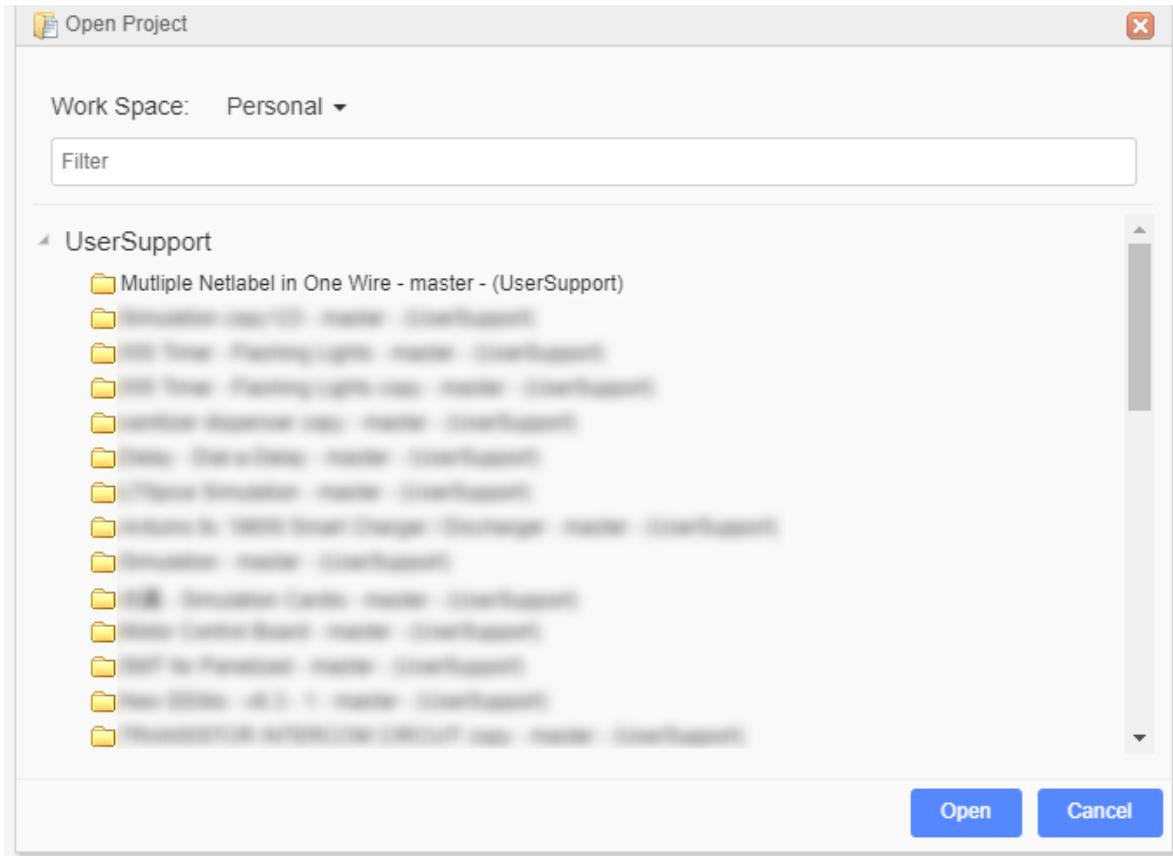
Top Menu - File - Open Project



Or click the Opened Project "open project" icon

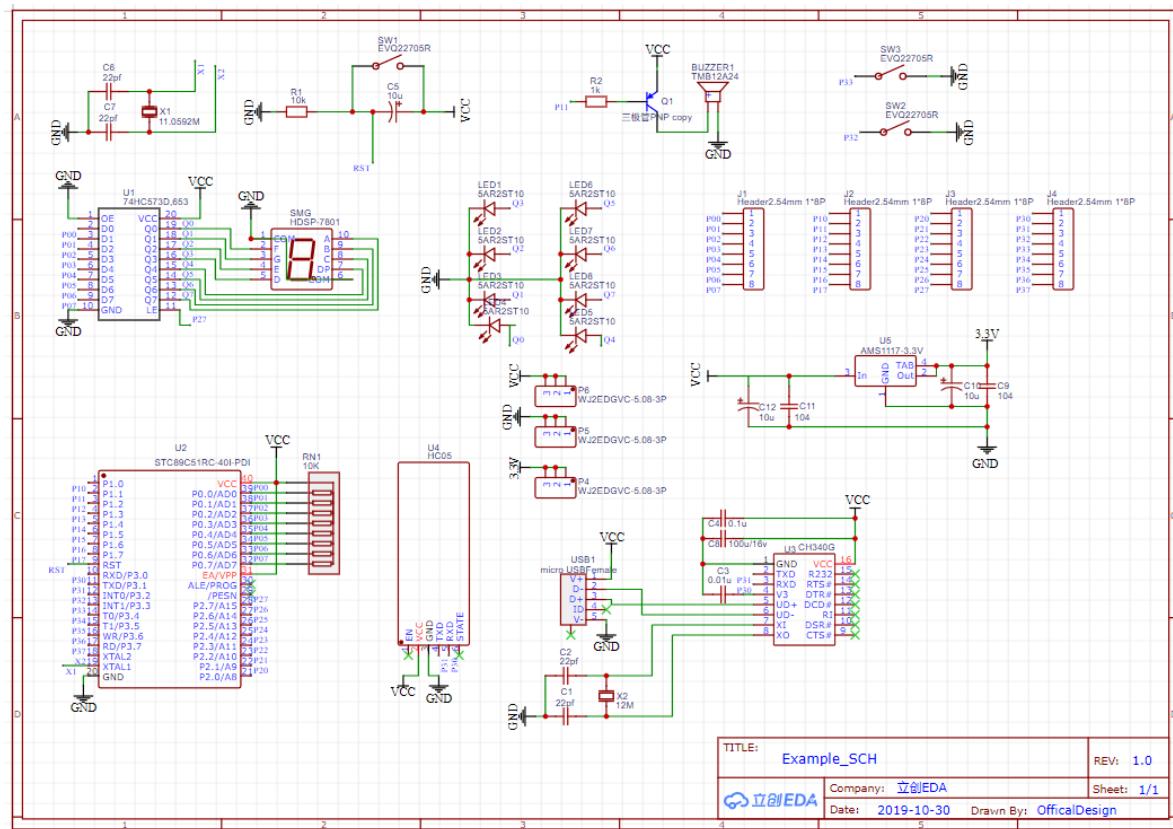


Select the project and open it.



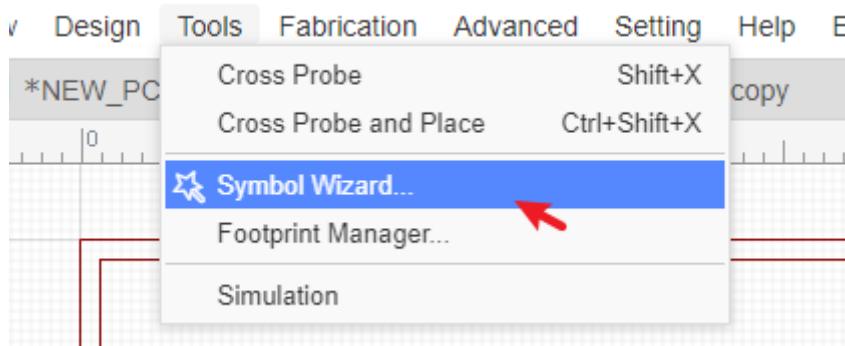
## Schematic Capture

EasyEDA can create highly professional looking schematics.



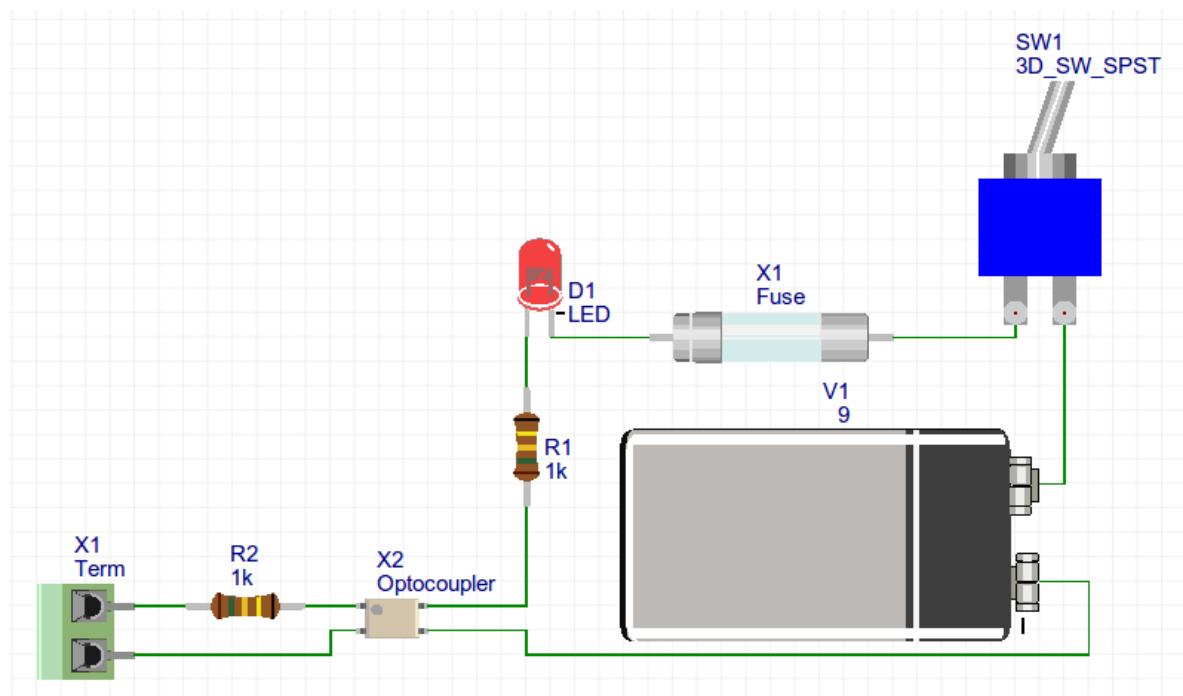
Because EasyEDA has some simple but powerful drawing capabilities, you can create your own symbols either by copying existing symbols into your own library and then editing and saving them, or by drawing them from scratch.

There is also a **Symbol Wizard** to quickly draw new symbols for [DIP](#), [QFP](#) and [SIP](#)



A feature of EasyEDA is that as well as extensive libraries of the usual simple "2D" graphical schematic symbols, it has a library of drawn [3D](#) component symbols, i.e. symbols that look like the physical components that they represent.

If you have enough time and patience using the drawing features to full effect in symbol creation, your schematic can be built like this:



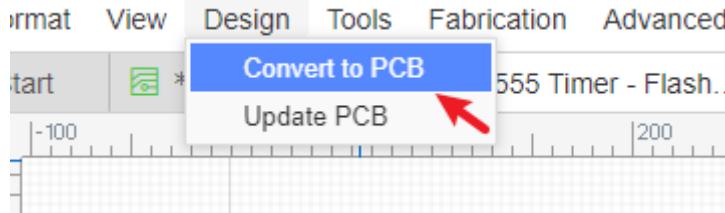
Another powerful feature is that it is also possible to import symbols from [Kicad](#), [Eagle](#) and [Altium](#) libraries.

## PCB Layout

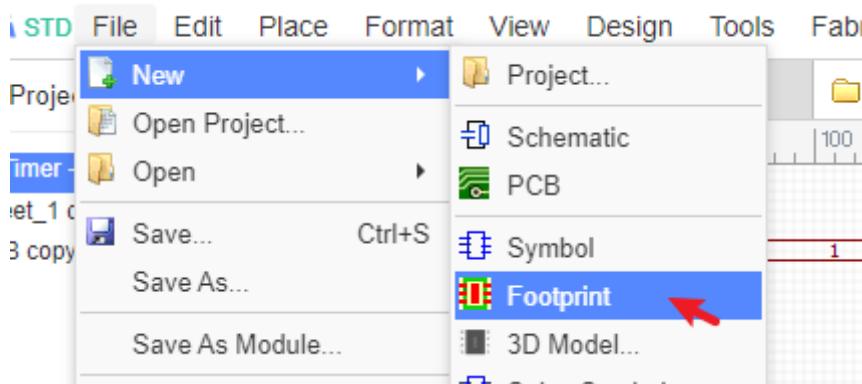
When you are satisfied with your schematic design and simulation results, you can then quickly proceed to produce your finished and populated PCB without leaving EasyEDA.

EasyEDA's PCB Design canvas helps you to quickly and easily lay out even complex multilayer designs from schematics you have already created in the Schematic canvas or directly as a layout with no schematic.

- Passing an EasyEDA Schematic into the PCB Design editor is as easy as clicking a button: Just click the **Convert to PCB** via: "Menu - Design - Convert to PCB".



- EasyEDA has extensive footprints. You can also build up your own library of unusual and specialized parts by copying and modifying existing parts or from scratch using EasyEDA's powerful footprint creation and editing tools.

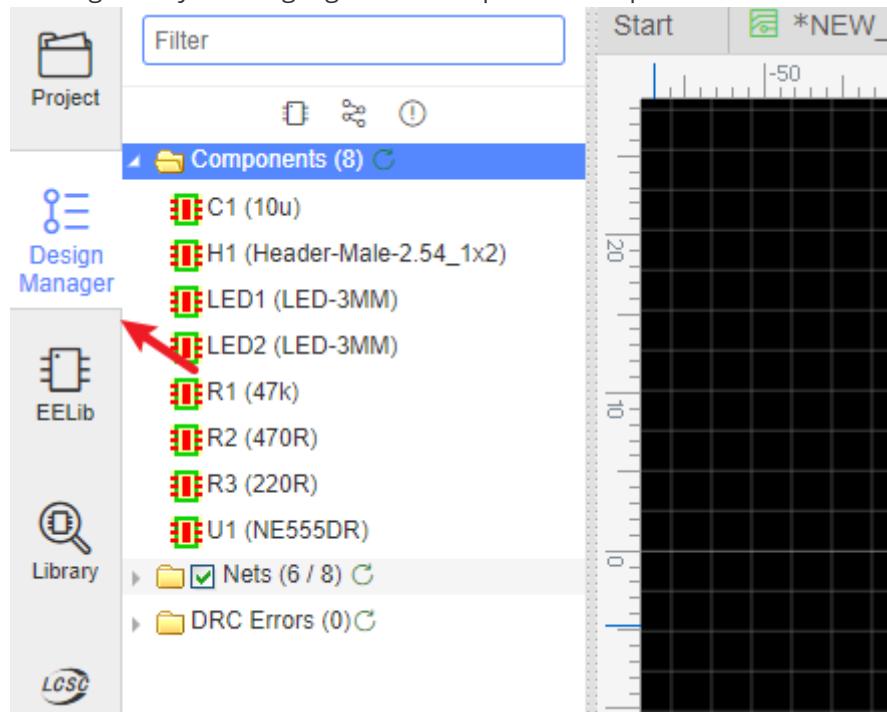


- In a similar way as in the Schematic design canvas, to help you locate items and navigate your way around when working in the PCB Design canvas there is a PCB Design Manager.

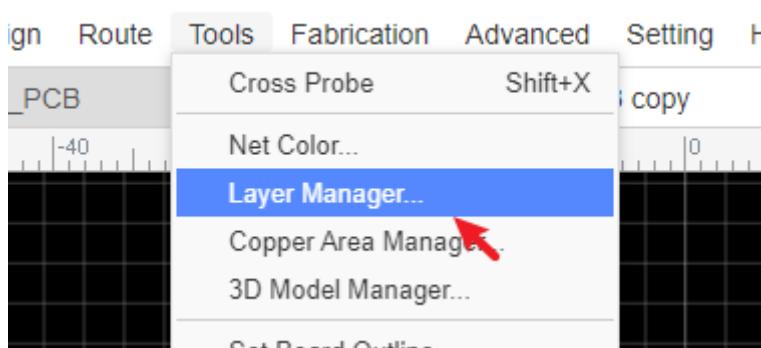
#### **Left Navigation Panel > Design Manager**

The PCB Design Manager is a very powerful tool for finding components, tracks (nets) and pads (Net Pads).

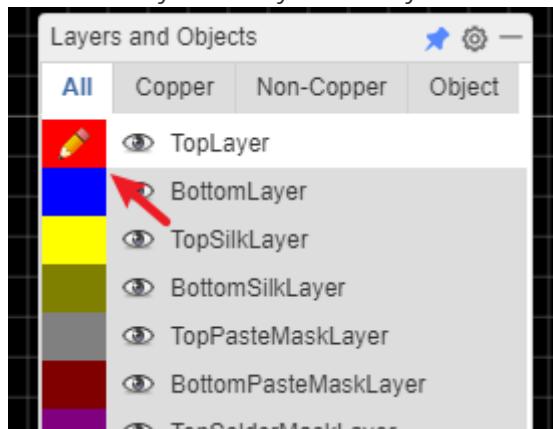
Clicking on any item highlights the component and pans it to the center of the window.



- You can set up layers used in the PCB and their display colours and visibility using **Top Menu - Tools - Layer Manager...**

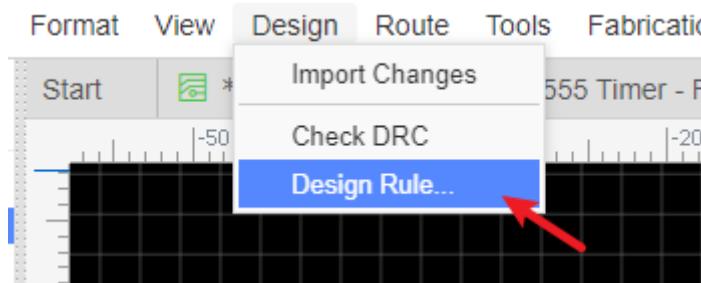


The active layer and layer visibility can be selected using the Layers Toolbar.



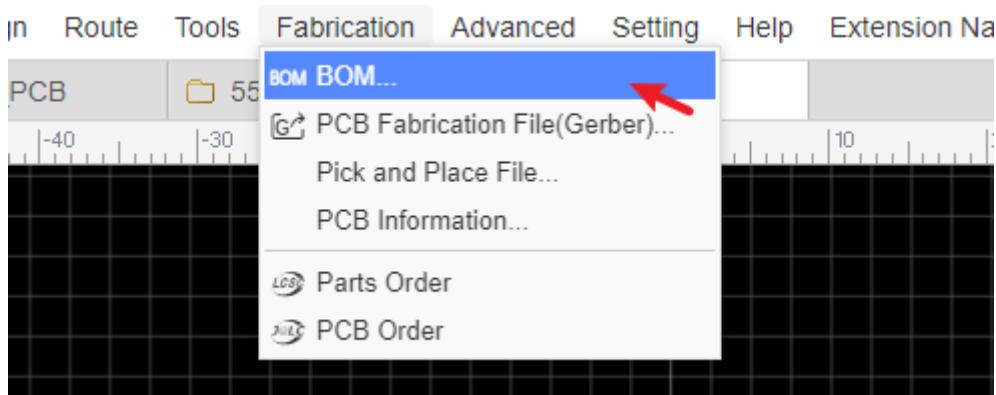
- Default track widths, clearances and via hole dimensions can all be configured in the Design Rule Check dialog which is opened via:

**Top Menu > Design > Design Rule...**



From first setting up the Design Rule Check (**DRC**) at the start of your board layout, running a DRC is almost the last step in checking your PCB design before you generate **Gerber** and **Drill** files for board manufacture ready to place your order for a finished PCB.

- The last step is to check the Gerber and Drill files using an easy way it is to install and use Free and Open Source Software Gerber Viewer: [Gerbv: http://gerbv.geda-project.org/](http://gerbv.geda-project.org/)
- While you are waiting for your PCB to be delivered, you can create a Bill of Materials (BOM) via:



- And you can produce professional quality `.svg`, `.png` or `.pdf` output files for your documentation.

PCB Designs can be shared with colleagues and made public in the same way as Schematics. The size of PCB that you can produce using EasyEDA is almost unlimited: designs of over 100cm \* 100cm are possible ... but you might need a powerful computer for that.

EasyEDA supports up to 6 layer PCBs by default but it is capable of handling more, so if you need more layers then please contact us.

### Search footprints

Searching footprints is the same as searching symbols by using **Library** in the Schematic. You can place the selected footprints in the canvas after the search.

## Libraries Management

Thanks to the Free and Open Source Kicad Libs and some Open Source Eagle libs, EasyEDA now has 700,000+ components, which should be enough for most projects.

Now you can enjoy using EasyEDA without having to spend so much time hunting for or building schematic symbols and PCB footprints.

- **Library**

On the left hand Navigation panel you will find "**EElib**" and "**Library**", just type what components you want and search.

At Libraries:

Search Libraries

1kohm 0603

Types: SCH Libs (1), PCB Libs, SCH Modules, PCB Modules

Classes: Personal(0), LCSC(42) (4), LCSC Assembled(2), System(0), Team(0), Follow(0), User Contributed(1)

| Title(PartNO)    | Package | Tolerance | Power  | Resistance | Inductance | Manufacturer | Description     |
|------------------|---------|-----------|--------|------------|------------|--------------|-----------------|
| 0603WAJ0102T5E   | 0603    | ±5%       | 1/10W  | 1KΩ        |            | UniOhm       | 1KΩ (102) ±5%   |
| RTT03102JTP      | 0603    | ±5%       | 1/10W  | 1KΩ(102)   |            | RALEC        |                 |
| BLM18BD102SN1D   | 0603    |           |        |            | 1KΩ        | MuRata       |                 |
| 4D03WGF1001T5(E) | 0603_X4 | ±1%       | 1/16W  | 1KΩ(1001)  |            | UniOhm       |                 |
| MPZ1608S102ATA00 | 0603    |           |        |            | 1kΩ        | TDK          |                 |
| RN731JTTD1001B25 | 0603    | ±0.1%     | 0.063W | 1KΩ        |            | KOA          |                 |
| BLM18AG102SN1D   | 0603    |           |        |            | 1KΩ        | MuRata       | 1KΩ±25% @100MHz |

Steps:

- o 1. Choose the library type
- o 2. Typing the keyword such as "1k 0603"
- o 3. Click the search button
- o 4. Select the class you which is wanted of the result
- o 5. If you don't need the search you need to remove all the search keywords

#### • Create Library

EasyEDA supports creating symbols by yourself, after created you can find out your components at **Library > Symbols/Footprints > Workspace**, and it is easy to manage your libraries.

Search Components, Footprints, Modules

Personal (2) (3), LCSC (1), LCSC Assembled (1), System (1), Team (1), Follow (1)

Refresh

Created

Favorite

#### • Transfer Libraries

If you want to transfer your libraries to the team, you can do that in "User Center > Libraries"

> Personal".

The screenshot shows the user's profile page with a sidebar on the left containing links like Projects, Teams, Modules, Libraries (which is selected), Friends, Notifications, and Messages. The main area is titled 'Libraries' and shows a list of libraries the user has created, followed by 'Team Libraries', 'Favorite (12)', and 'Followed'. Below this is a 'Resource' section with links to Schematic Libraries and PCB Libraries. To the right, under 'Created', there is a list of libraries: 'test.2' (3 days ago) and 'test' (3 days ago). The 'test.2' entry has an 'Edit' button highlighted with a red arrow.

|   |      |       |    |
|---|------|-------|----|
| 1 | PIN1 | PIN12 | 12 |
| 2 | PIN2 | PIN11 | 11 |
| 3 | PIN3 | PIN10 | 10 |
| 4 | PIN4 | PIN9  | 9  |
| 5 | PIN5 | PIN8  | 8  |
| 6 | PIN6 | PIN7  | 7  |

To prepare for the final assembly stage you can create a Bill of Materials (**BOM**) via: **File > Export BOM...**

and you can produce professional quality **.SVG**, **.PNG** or **.PDF** output files for your documents.

All EasyEDA Schematic Symbol and PCB Footprint libs are public, so after you have created and saved a new symbol or footprint, others will be able to find your part and you will be credited as a contributor. <https://easyeda.com/page/contribute>

## Version-Control

EasyEDA provide a simple but powerful version control feature. Each version is independent, you can edit and save for every version.

When create the new project, it will be set the default version name as "master", you can edit the name at the "Project Manage - Version" page.

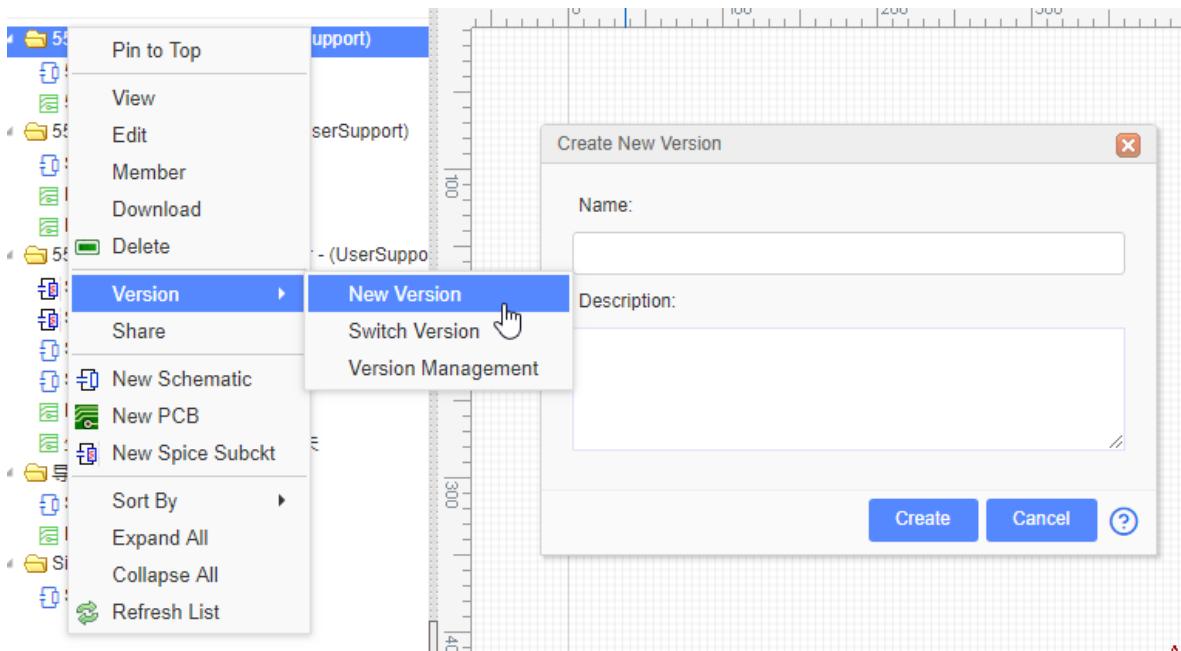
You can create 10 versions for every project. The more versions you need to delete the older first.

### Create New Verison

Via: Project folder - right-click menu - Version - New Version

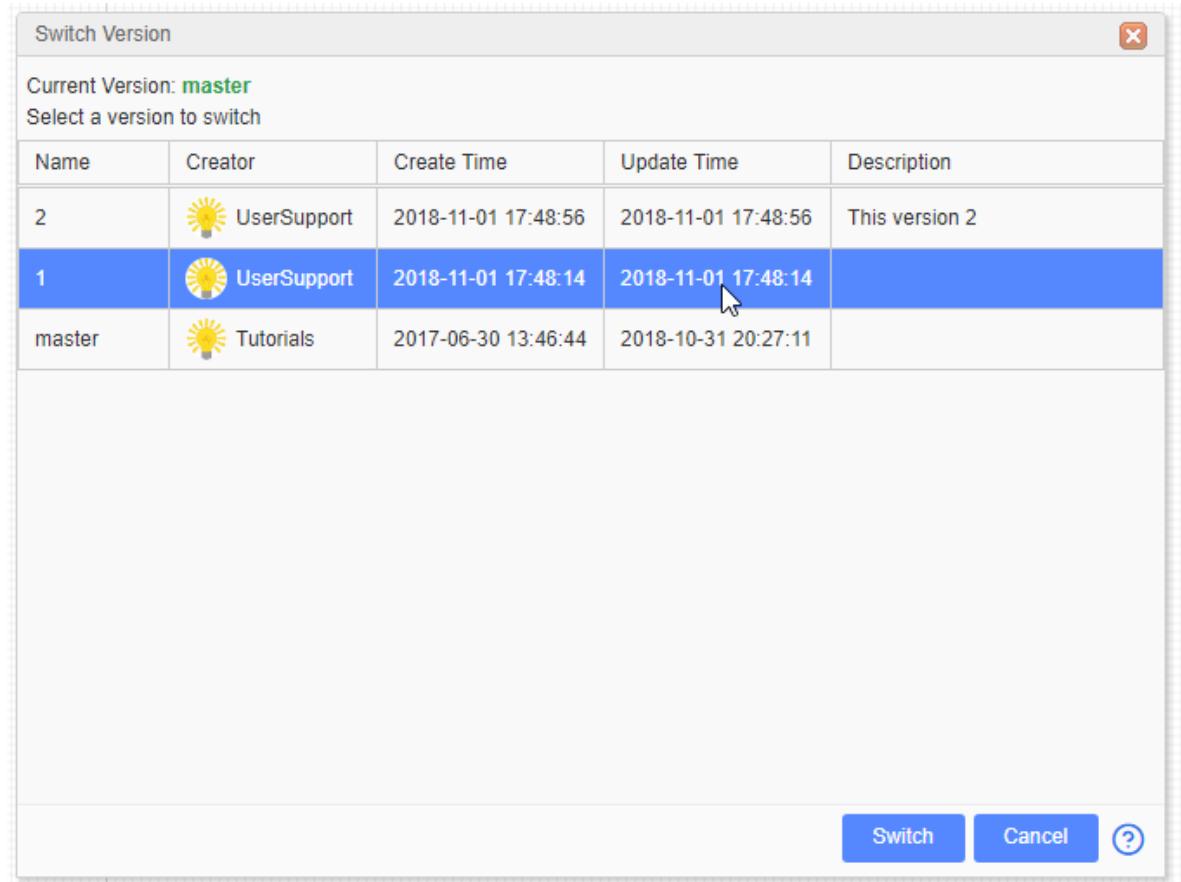
At the new version dialog, you need to type the version's name and description, and create it.

If you want to switch to new version, you have via "Version - Switch Version".



## Switch Version

Click "Switch", the dialog will list current version and all version for this project, you can select one and switch to it.

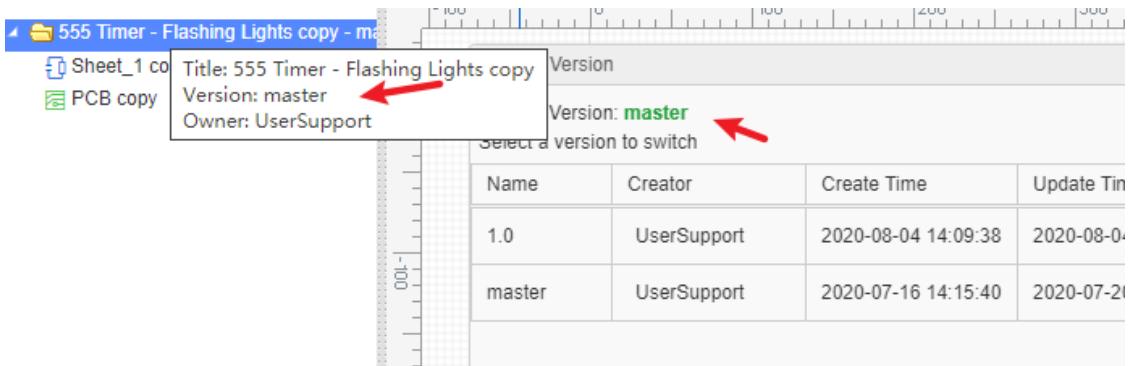


| Name   | Creator       | Create Time         | Update Time         | Description    |
|--------|---------------|---------------------|---------------------|----------------|
| 2      | 💡 UserSupport | 2018-11-01 17:48:56 | 2018-11-01 17:48:56 | This version 2 |
| 1      | 💡 UserSupport | 2018-11-01 17:48:14 | 2018-11-01 17:48:14 |                |
| master | 💡 Tutorials   | 2017-06-30 13:46:44 | 2018-10-31 20:27:11 |                |

**Switch**   **Cancel**   **?**

Notice:

- Before switching the other version, you have to close the current version's document manually first.
- You only can open the current version's document, if you want to open other's version's document, you have to switch the version first.
- If you not sure which version it is, you can check it at "Switch Version" dialog to check "Current Version", or hover the mouse cursor on the project folder.



## Version Management

Via "Version Management", will open the "Project Page - Version".

Version page will list all versions, you can edit the versions' name and description, or delete them. Current version can't be deleted.

The screenshot shows the 'Project Versions' page. On the left is a sidebar with icons for Back, Version (selected), Attachments, Members, and Settings. The main area has a header 'Project Versions'. Below it is a 'Current Version' section with fields for Name (master) and Description. Underneath is a 'Version List' table.

| Name   | Creator     | Create Time          | Update Time          | Description    | Operations |
|--------|-------------|----------------------|----------------------|----------------|------------|
| master | Tutorials   | 2017-06-3 0 05:46:44 | 2018-10-3 1 12:27:11 |                |            |
| 1      | UserSupport | 2018-11-0 1 09:48:14 | 2018-11-0 1 09:48:14 |                |            |
| 2      | UserSupport | 2018-11-0 1 09:48:56 | 2018-11-0 1 09:48:56 | This version 2 |            |

## Share with Public

Sharing your work with others is a big feature of web based EDA tools and EasyEDA is no exception in offering you some nice features.

Did you create a really cool project with EasyEDA? Show it off and be super helpful to other EasyEDA users, you just need to set your projects to public, so others can explore your circuits.

All projects in EasyEDA are set to private by default, your private project can not be shared with anyone.

i.e. to make it public, you should right click and edit your existing project to be a Public project:

- At the Workspace, click **Share** icon when the mouse hover the project cover, it will ask you to confirm.

Update Time operation

|            |                                |                        |                           |                          |                          |                       |                        |
|------------|--------------------------------|------------------------|---------------------------|--------------------------|--------------------------|-----------------------|------------------------|
| days ago   | <a href="#">Open in Editor</a> | <a href="#">Detail</a> | <a href="#">Members</a>   | <a href="#">Settings</a> | <a href="#">Clone</a>    | <a href="#">Share</a> | <a href="#">Delete</a> |
| 5 days ago | <a href="#">Open in Editor</a> | <a href="#">Detail</a> | <a href="#">Home page</a> | <a href="#">Members</a>  | <a href="#">Settings</a> | <a href="#">Clone</a> | <a href="#">Share</a>  |

Or enter project manage page, via "Workspace > Project > Manage > Settings > Basic > Project property:Public"

Project property

Public  Private

Public license

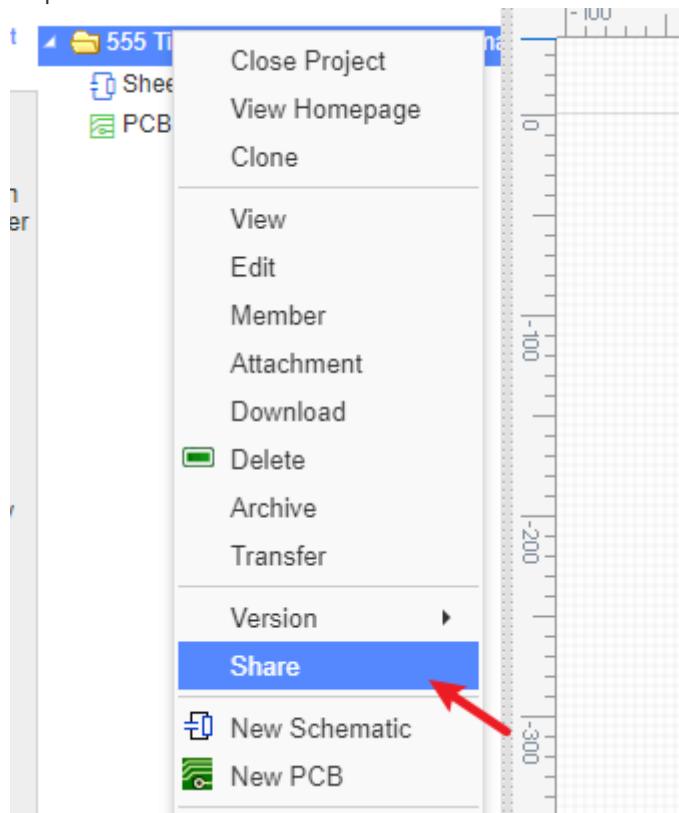
[Choose a public license](#)

Tags

[+ Tags](#)

**Save**

- At the editor, you can right-click the project, click the **Share** menu, after setting the project as public.



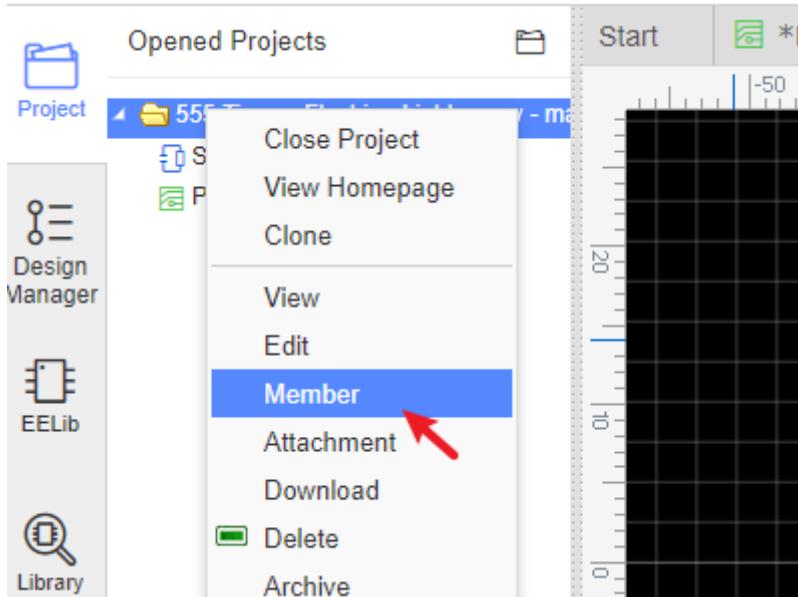
# Project Member

How to share project with selected people?

Can you share a private project with your partner? Can your partner modify your designs?

Yes, you can use **Member** to do this.

Right click the project and you will see the **Member** on the context menu; clicking on it will open the Member webpage.



So if you want to share a project with someone,

1. You just need to know their E-mail address which they have used to create an account with EasyEDA
2. The project member you can set as "Developer", "Manager", and "Observer".

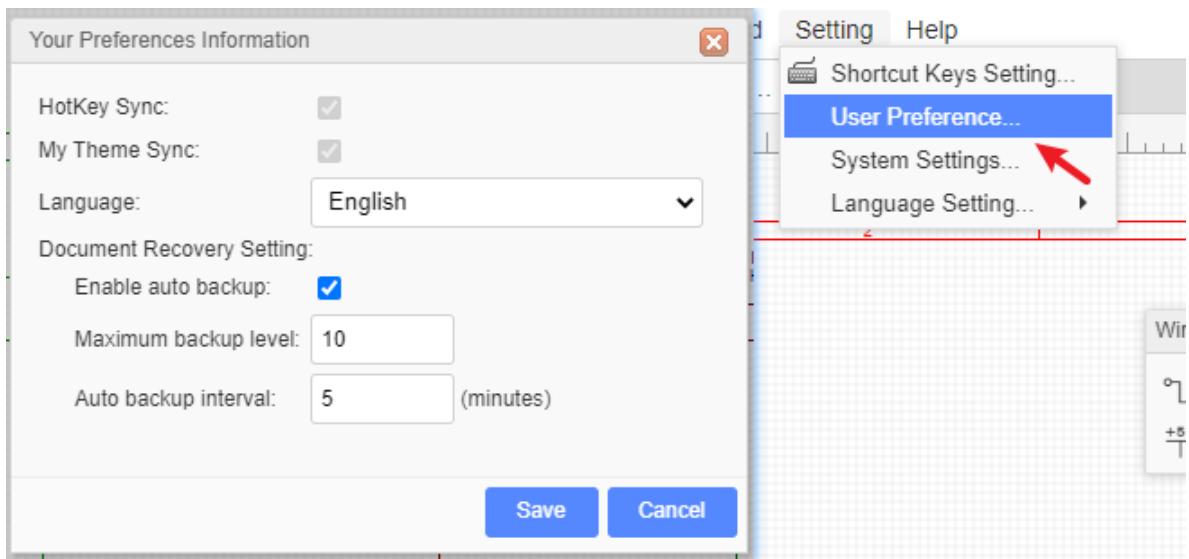
After setting up **Member** and Permissions, your partner will find your project in the **Open Project** when they login.

If your partner doesn't wish to accept the shared project, they can reject it by leaving the project when they enter this project "Member" function.

## User Preference

When EasyEDA shows up the login success pop up in the bottom right of the window, your user management menu will look like this:

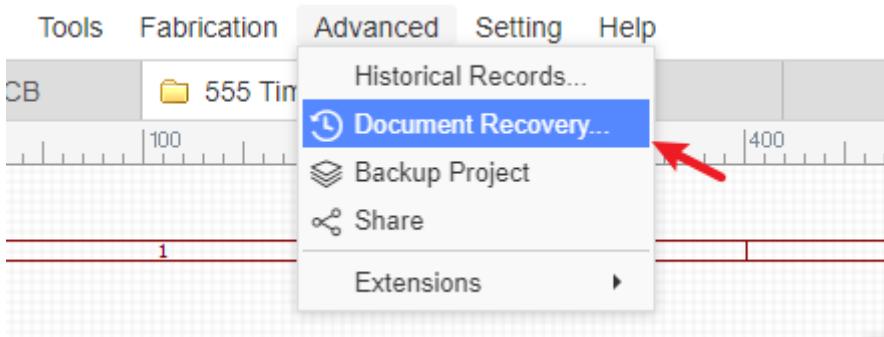
Click on **Top Menu - Setting - User Preferences**,



#### Document Recovery Setting:

- **Maximum backup level:** Every opened document can be saved as a backup to this number of different revisions.
- **Auto backup interval:** This is the time interval between auto saves of all your opened documents.

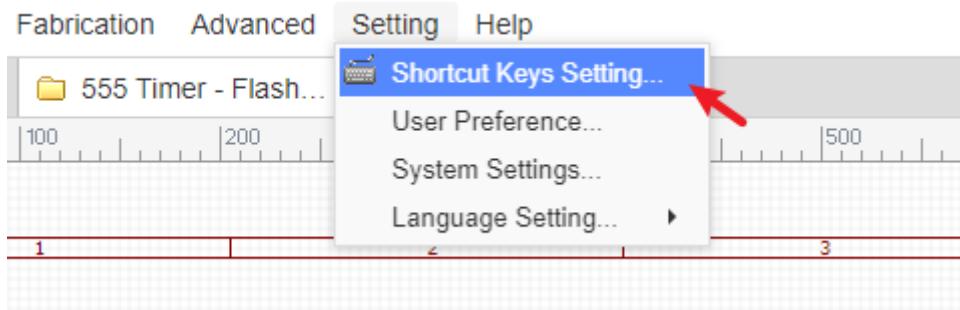
The Document Recovery function you can find at:



## Shortcut Keys

After a while of using an EDA tool suite, clicking all over the place with a mouse gets very tedious and seriously reduces your productivity. Keyboard shortcuts or Hotkeys avoid much of that. EasyEDA not only provides lots of hotkeys, but also every hotkey can be reconfigured.

Under the Setting menu, click the Hotkeys Setting... Menu which will open the Hotkey Setting dialog.



To change a Hotkey, click anywhere in the row for the hotkey you want to change and then press your new key.

For example, if you want to use **R** instead of **space** to rotate selected objects, click on the first row, then press **R**.

After you change the hotkey, don't forget to click Save Changes button.

The **docType** column describes which type of EasyEDA document each hotkey applies to.

**docType** has three types:

- **ALL**: any document type in EasyEDA.
- **SCH**: schematic and schematic libs
- **PCB**: PCB and Footprints.

The functions of some hotkeys may change between docTypes. For example, the hotkey **C** draws an Arc in SCH, but draws a circle in PCB.

A list of all the available default hotkeys is given below.

## All document

| <b>DocType</b> | <b>Shortcut</b> | <b>Function</b>                                                                  |
|----------------|-----------------|----------------------------------------------------------------------------------|
| All            | Space           | Rotate selected objects                                                          |
| All            | Right-Click     | Keep right-click to pan canvas; Open offset dialog when select one object        |
| All            | Left            | Scroll Or Move selected left                                                     |
| All            | Right           | Scroll or Move selected right                                                    |
| All            | Up              | Scroll or Move selected up                                                       |
| All            | Down            | Scroll or Move selected down                                                     |
| All            | TAB             | Change object's attributes when placing; Open offset dialog when select a object |
| All            | Esc             | Cancel current drawing                                                           |
| All            | Home            | setting new canvas origin                                                        |
| All            | Delete          | Delete Selected                                                                  |
| All            | F1              | Open tutorials                                                                   |
| All            | F11             | Full screen at browser                                                           |
| All            | A               | Zoom In                                                                          |
| All            | Z               | Zoom Out                                                                         |
| All            | D               | Drag                                                                             |
| All            | K               | Fit Window                                                                       |
| All            | R               | Rotate selected objects                                                          |
| All            | X               | Flip Horizontal(doesn't support footprint)                                       |
| All            | Y               | Flip Vertical(doesn't support footprint)                                         |
| All            | ALT+F5          | Full screen at browser                                                           |
| All            | CTRL+X          | Cut                                                                              |
| All            | CTRL+C          | Copy                                                                             |
| All            | CTRL+V          | Paste                                                                            |
| All            | CTRL+A          | Select All                                                                       |
| All            | CTRL+Z          | Undo                                                                             |
| All            | CTRL+Y          | Redo                                                                             |
| All            | CTRL+S          | Save                                                                             |
| All            | CTRL+F          | Find Component                                                                   |
| All            | CTRL+D          | Design Manager                                                                   |

| <b>DocType</b> | <b>Shortcut</b> | <b>Function</b>                             |
|----------------|-----------------|---------------------------------------------|
| All            | CTRL+Home       | Open canvas origin setting dialog           |
| All            | SHIFT+1         | Cycle forward to next open tabbed document  |
| All            | SHIFT+2         | Cycle backward to next open tabbed document |
| All            | SHIFT+X         | Cross Probe                                 |
| All            | SHIFT+F         | Search Library                              |
| All            | SHIFT+Drag      | Cursor snap to part's origin                |
| All            | SHIFT+ALT+H     | Align horizontal centers                    |
| All            | SHIFT+ALT+E     | Align vertical centers                      |
| All            | CTRL+SHIFT+L    | Align left                                  |
| All            | CTRL+SHIFT+R    | Align right                                 |
| All            | CTRL+SHIFT+O    | Align top                                   |
| All            | CTRL+SHIFT+B    | Align bottom                                |
| All            | CTRL+SHIFT+G    | Align grid                                  |
| All            | CTRL+SHIFT+H    | Distribute Horizontally                     |
| All            | CTRL+SHIFT+E    | Distribute Vertically                       |
| All            | CTRL+SHIFT+F    | Find similar objects                        |

## Schematic

---

| <b>DocType</b> | <b>Shortcut</b> | <b>Function</b>             |
|----------------|-----------------|-----------------------------|
| Schematic      | W               | Draw Wire                   |
| Schematic      | B               | Draw Bus                    |
| Schematic      | U               | Bus Entry                   |
| Schematic      | N               | NetLabel                    |
| Schematic      | P               | Place Pin                   |
| Schematic      | L               | Draw Polyline               |
| Schematic      | O               | Draw Polygon                |
| Schematic      | Q               | Draw Bezier                 |
| Schematic      | C               | Draw Arc                    |
| Schematic      | S               | Draw Rect                   |
| Schematic      | E               | Draw Ellipse                |
| Schematic      | F               | Freehand Draw               |
| Schematic      | T               | Place Text                  |
| Schematic      | I               | Edit Selected Symbol        |
| Schematic      | CTRL+Q          | NetFlag VCC                 |
| Schematic      | CTRL+G          | NetFlag GND                 |
| Schematic      | F8              | Run the Document Simulation |
| Schematic      | CTRL+J          | Open the Simulation Setting |
| Schematic      | CTRL+SHIFT+X    | Cross Probe and Place       |

## PCB

---

| <b>DocType</b> | <b>Shortcut</b> | <b>Function</b>                                             |
|----------------|-----------------|-------------------------------------------------------------|
| PCB            | W               | Draw Track                                                  |
| PCB            | U               | Draw Arc                                                    |
| PCB            | C               | Draw Circle                                                 |
| PCB            | N               | Draw Dimension                                              |
| PCB            | S               | Draw Text                                                   |
| PCB            | O               | Draw Connect                                                |
| PCB            | E               | Draw copperArea                                             |
| PCB            | T               | Change To TopLayer; Change selected part to toplayer        |
| PCB            | B               | Change To BottomLayer; Change selected part to bottomlayer  |
| PCB            | 1               | Change To Inner1                                            |
| PCB            | 2               | Change To Inner2                                            |
| PCB            | 3               | Change To Inner3                                            |
| PCB            | 4               | Change To Inner4                                            |
| PCB            | P               | Place Pad                                                   |
| PCB            | Q               | Change canvas unit                                          |
| PCB            | V               | Place Via                                                   |
| PCB            | M               | Measure                                                     |
| PCB            | H               | Highlight Net all the time, press it again cancel highlight |
| PCB            | L               | Change Route Angle                                          |
| PCB            | -               | Decrease Routing Width; Switch to the forward signal layer  |
| PCB            | +               | Increase Routing Width; Switch to the next signal layer     |
| PCB            | *               | Cycle switch to the next signal layer                       |
| PCB            | Delete          | Delete selected object; Undo the track when routing         |
| PCB            | ALT--           | Decrease Snap Size                                          |
| PCB            | ALT++           | Increase Snap Size                                          |
| PCB            | CTRL+R          | Depend on reference point for copy object repeatedly        |
| PCB            | CTRL+L          | Open layer manager                                          |
| PCB            | CTRL+Q          | Hide/show network text                                      |

| DocType | Shortcut           | Function                                                        |
|---------|--------------------|-----------------------------------------------------------------|
| PCB     | SHIFT+M            | Remove All Copper Area fill data                                |
| PCB     | SHIFT+B            | Rebuild All Copper Area                                         |
| PCB     | SHIFT+D            | Move Object(s) by reference point                               |
| PCB     | SHIFT+G            | Display track length while routing                              |
| PCB     | SHIFT+W            | Show favorite track width while routing                         |
| PCB     | SHIFT+R            | Change routing conflict                                         |
| PCB     | SHIFT+S            | Toggle layers which is not active                               |
| PCB     | SHIFT+Double Click | Delete selected track segment                                   |
| PCB     | CTRL+SHIFT+V       | Paste object(s) and keep the prefix, and hide the ratline layer |
| PCB     | CTRL+SHIFT+SPACE   | Change routing angle, same as hotkey L                          |

## Basic Skills

---

To use EasyEDA, you need to be familiar with a few basic terms and concepts. The best way to learn them is to open up EasyEDA, open a new schematic:

**File > New > Schematic**, and play!

## Saving Your Work Locally

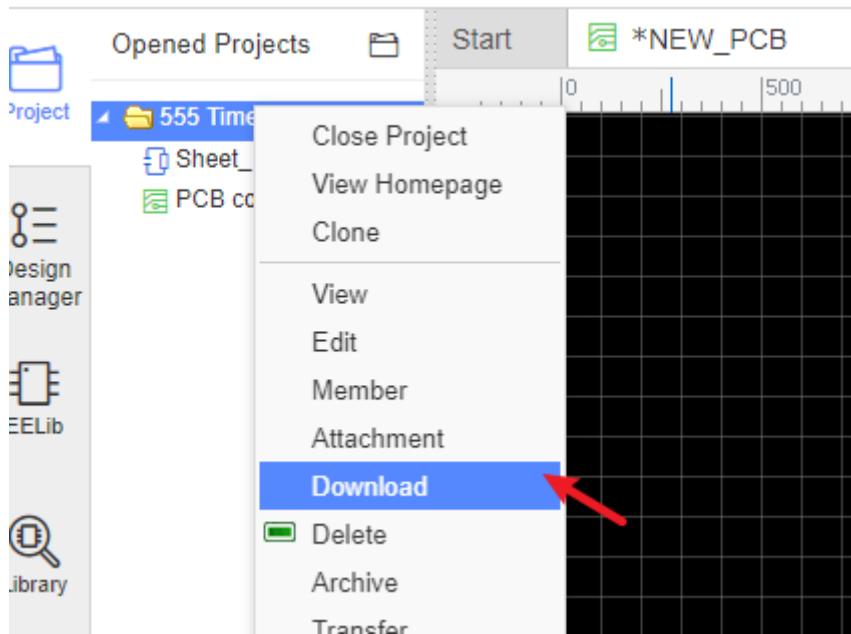
---

Although EasyEDA saves all your files on our Server, sometimes you may want to save your work locally and EasyEDA provides a hack way to do this.

You can right-click your project folder, and click "Download Project", or export your design as EasyEDA source file via "File > EasyEDA Source".

the more detail you can view at [Export EasyEDA Source](#) section.

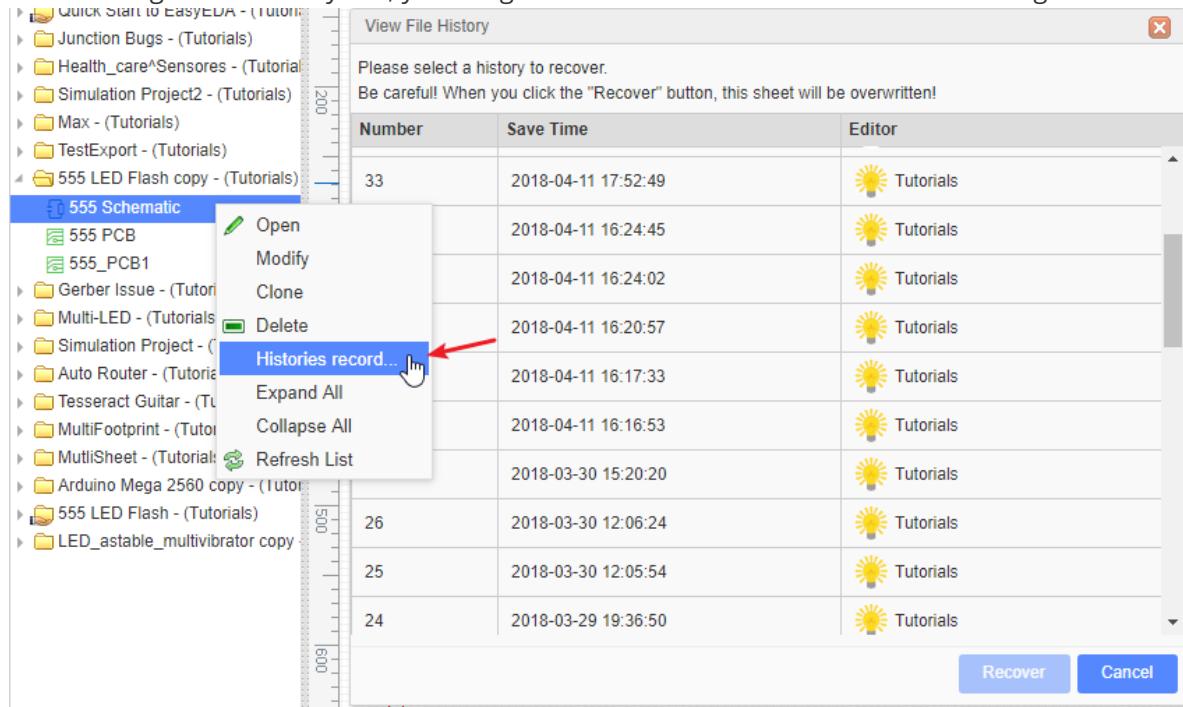
Or you can download your project.



## Histories Record

It is easy to use this function, right click on the document for which you need the history in like in the image below:

After clicking on the history link, you will get a list of all of the Histories like in the image below.



Click the History number, you can open the saved file in the editor, if this is what you need, you can save it to your project and delete your bad file.

### Note:

1. For now all of the Histories are marked as number, we will allow you to add a tag soon.
2. Don't save your files too frequently, or you will get lots of Histories and it will be hard to find the exact one you want.

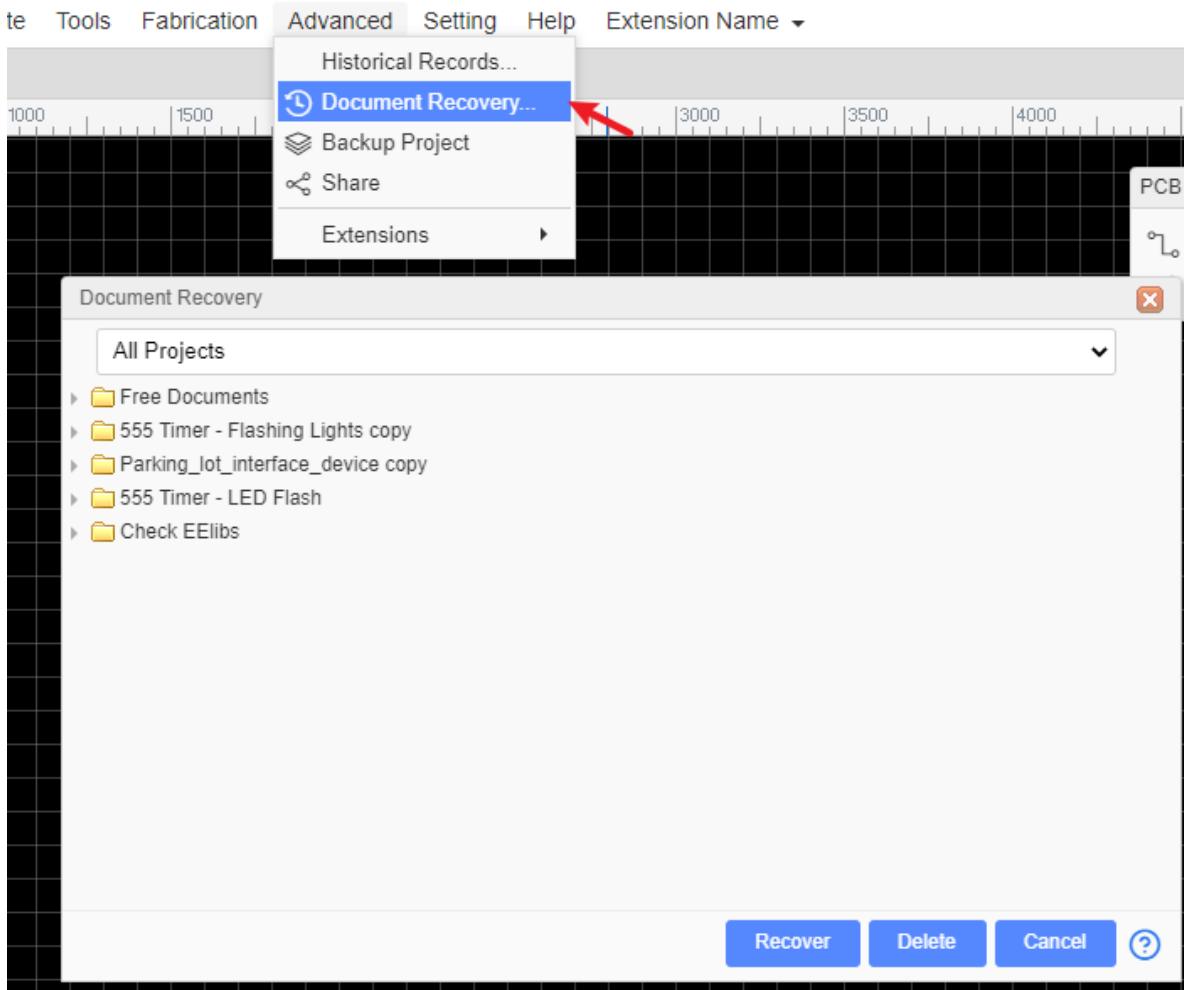
## Document Recovery

No operating system, software or network is perfect, so sometimes things can go wrong. Having your Desktop or web browser freeze or your broadband connection drop, two hours into laying out a PCB, could spoil your day.

However, with EasyEDA, your day will be just fine.

This is because EasyEDA auto saves and makes backups of all your open files to your computer so crash recovery is built into EasyEDA.

On the top menu, click **Menu - Advanced - Document Recovery** as below:



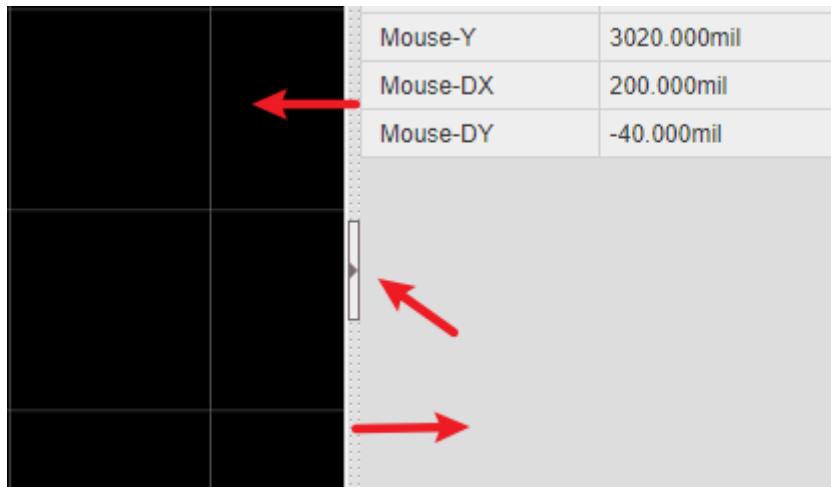
Expand the folder to the latest, Select the file which you would like to **recover**, then click the Recover button; your file will be opened in a new tab, then save the opened file.

**Please note:**

- *EasyEDA saves these crash recovery files on your computer and not on the EasyEDA server. Therefore you cannot recover files from a crash on one computer or browser by changing to a different computer or browser.*
- *And if you cleaned your browser's cache, the recovery files will disappear.*
- *If you make a mistake to delete a file and remove the cache already, maybe you can find your document back via recycle bin: <https://easyeda.com/account/user/recycles/personal>.*

## Resizing the canvas area

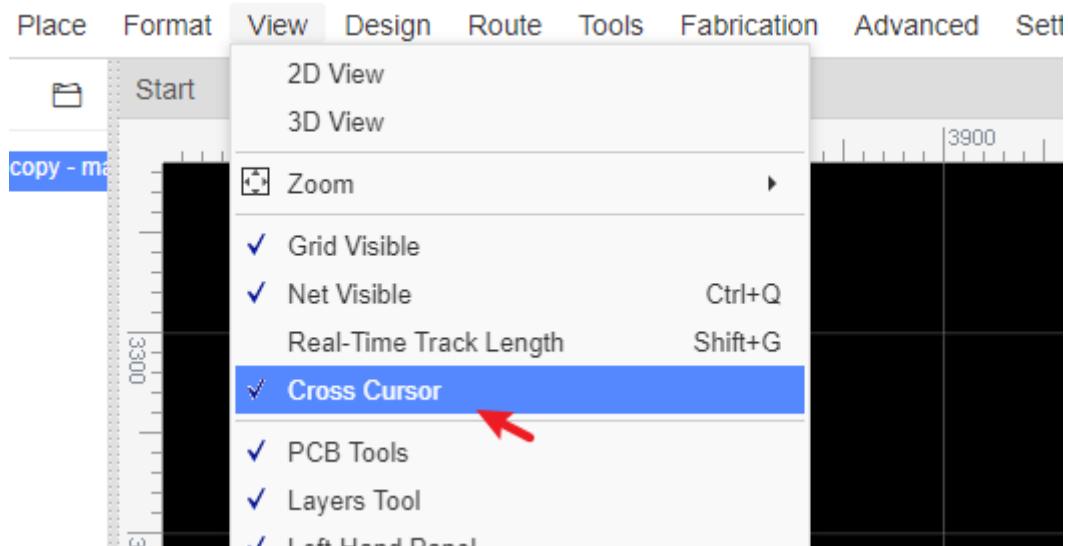
Hovering the mouse cursor over the areas indicated by the three green ellipses will bring up blue sidebar toggle lines. Clicking on them will toggle the visibility of their associated, right and left areas to expand the canvas area. The vertical lines can also be dragged horizontally to resize the panels.



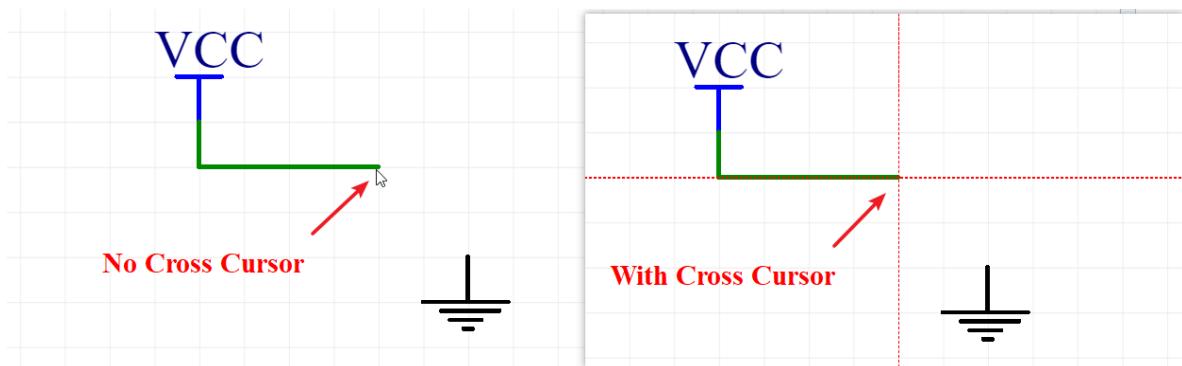
## Cursor Style

Some users don't like the cross cursor, so you can change it to arrow cursor like in the image below.

Via: Top Menu - View - Cross Cursor



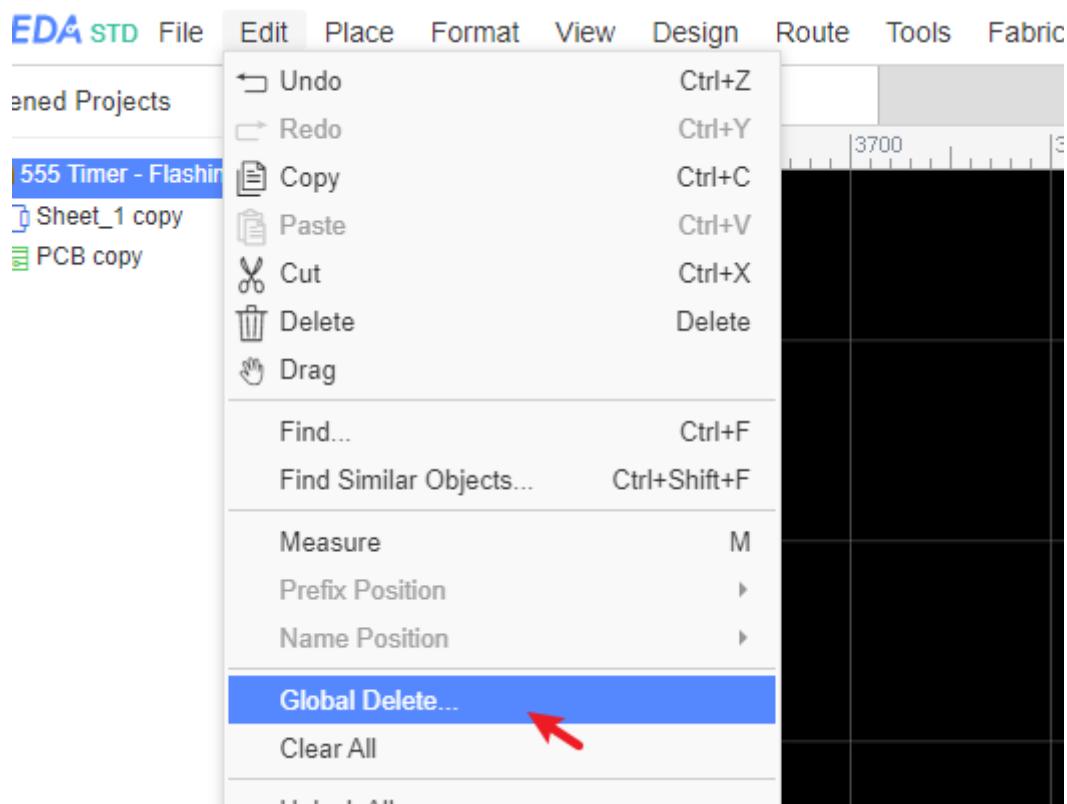
These difference between these options is as below:



## Clear and delete

If you think your schematic or PCB looks terrible, and you want to redraw all units, you can:

- Top Menu > Edit > Global Delete.



- Delete this schematic and create a new one.
- Click one object or CTRL+A, press delete key to remove all objects.

## Left clicking

---

Similar to other EDA software:

- Click on an item to select it;
- If over a selected item, click and hold to drag a selected item;
- If not over a selected item, clicking and holding while dragging creates a selection box;
- the selection box, using click and drag to the right, selects everything inside the box;
- the selection box, using click and drag to the left, selects everything inside and intersected by the box;
- Double click on a text area to edit it;
- The exact left click functionality depends on what item is being selected and in what Canvas the item exists (Schematic or PCB).

## Right clicking

---

EasyEDA does not support right click context menus in the Schematic or PCB Canvas. Instead, right clicking executes a context sensitive command:

- When you are placing a symbol, after a right click, the active symbol will be removed;
- When you are drawing a shape such as a polyline, after a right click, the polyline will be stopped at the place where you right click but the mouse will remain as a **cross**, so you can draw another shape;
- To get out of the current active context sensitive command such as placement or drawing mode and go back to **select mode**, just double right click.

**Ctrl+Right** clicking anywhere in the Schematic, waveForm or PCB Canvas drags the canvas around within the EasyEDA window.

## ESC key

Pressing the `ESC` key ends the current drawing action but does not exit the current active context sensitive command mode (i.e. it does not return the cursor to select mode).

## Select more shapes

- `CTRL+Left Click` on items adds those items to your selection;
- Clicking and holding creates a selection box;
- Creating a selection box, using click and drag to the right, selects everything inside the box;
- Creating a selection box, using click and drag to the left, selects everything inside and intersected by the box;

## Zoom in and Zoom out

- Using the middle mouse button:
- Roll forward to zoom in;
- Roll back to zoom out;
- Using hotkeys, the default hotkey `A` for zoom in, `Z` for zoom out.

### Please note:

*Do not scroll your mouse at the same time as pressing the `CTRL` key when your cursor on the top menu, the browsers will zoom the whole website, if you just want to zoom the canvas in the EasyEDA window, you need to make your cursor into the canvas. If zoom the whole website happens, just press `ctrl+0` to reset the browser view zoom.*

## Double clicks

Double clicking any text area opens a resizable text box to allow you edit the text inline.



Press enter to create new line. Click outside the text box to close it.

## Pan/Move Canvas

- Right click anywhere in the Schematic, WaveForm or PCB Canvas and Hold down right button to drags the canvas around within the EasyEDA window.
- If your canvas is bigger than the EasyEDA window and is showing scroll bars, you can use either the scroll bars or the Arrow keys to scroll the canvas to pan.
- When drawing a wire, a graphic line or shape that you wish to extend beyond the edge of the EasyEDA window holding down the left mouse button after starting the line will pan the canvas to keep the drawn item inside the window.

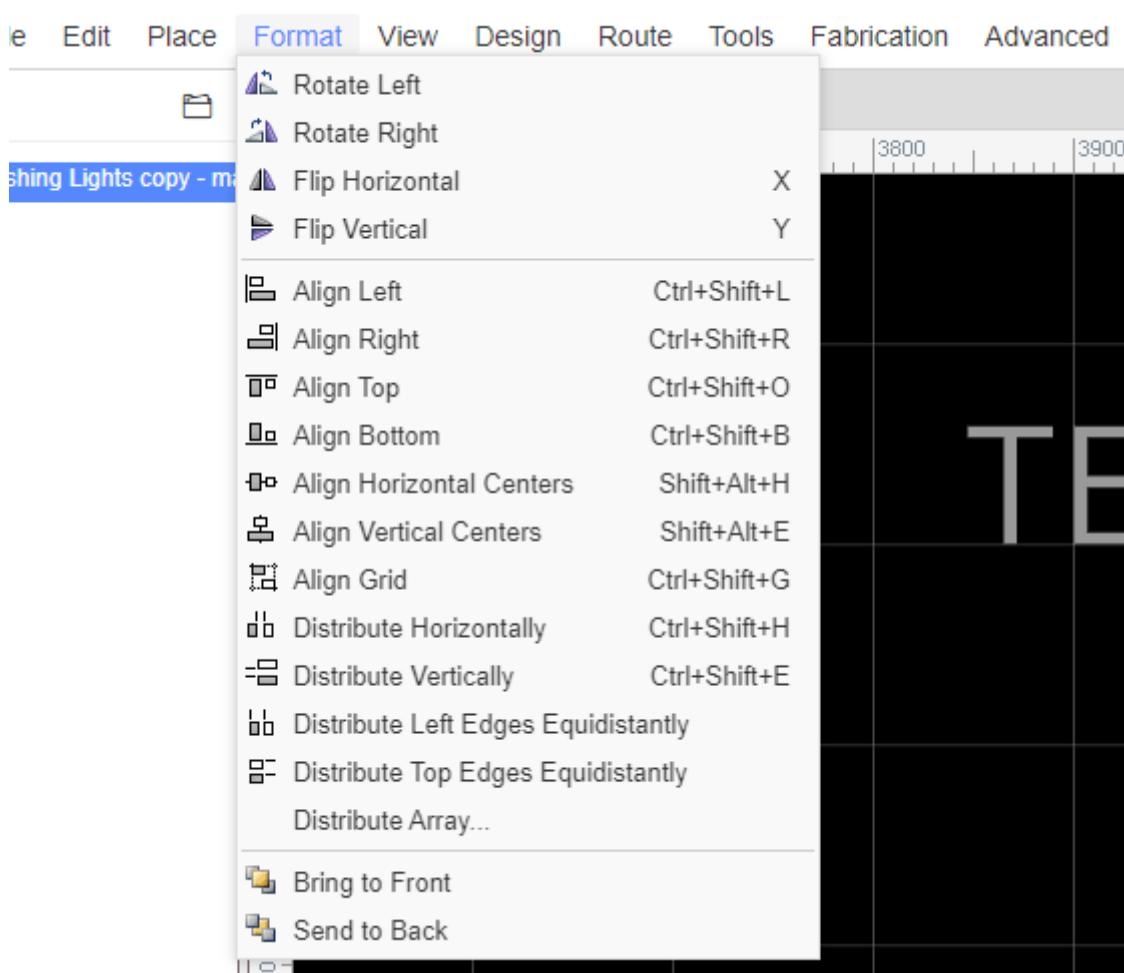
### Tip:

If you use Chrome, and cursor is in the canvas while pressing CTRL or ALT key and rolling your mouse, the canvas will move vertically, and when pressing SHIFT and rolling your mouse, the canvas will move horizontally.

## Rotate

After selecting one or more items, you can rotate the selected items using:

Top Menu > Format > Rotate or by pressing the default rotate hotkey: Space .



when in PCB, you can click the footprint and change it's rotation at the right property panel.

### Please note:

Rotating a multiple selection rotates each item about its own symbol origin. It does not rotate the items about the centroid of the group of items.

## Flip

To place a Q2 as shown in the schematic below you need to Flip the item. Via: Top menu - Format - Flip.



You can Flip one or more selected items using:

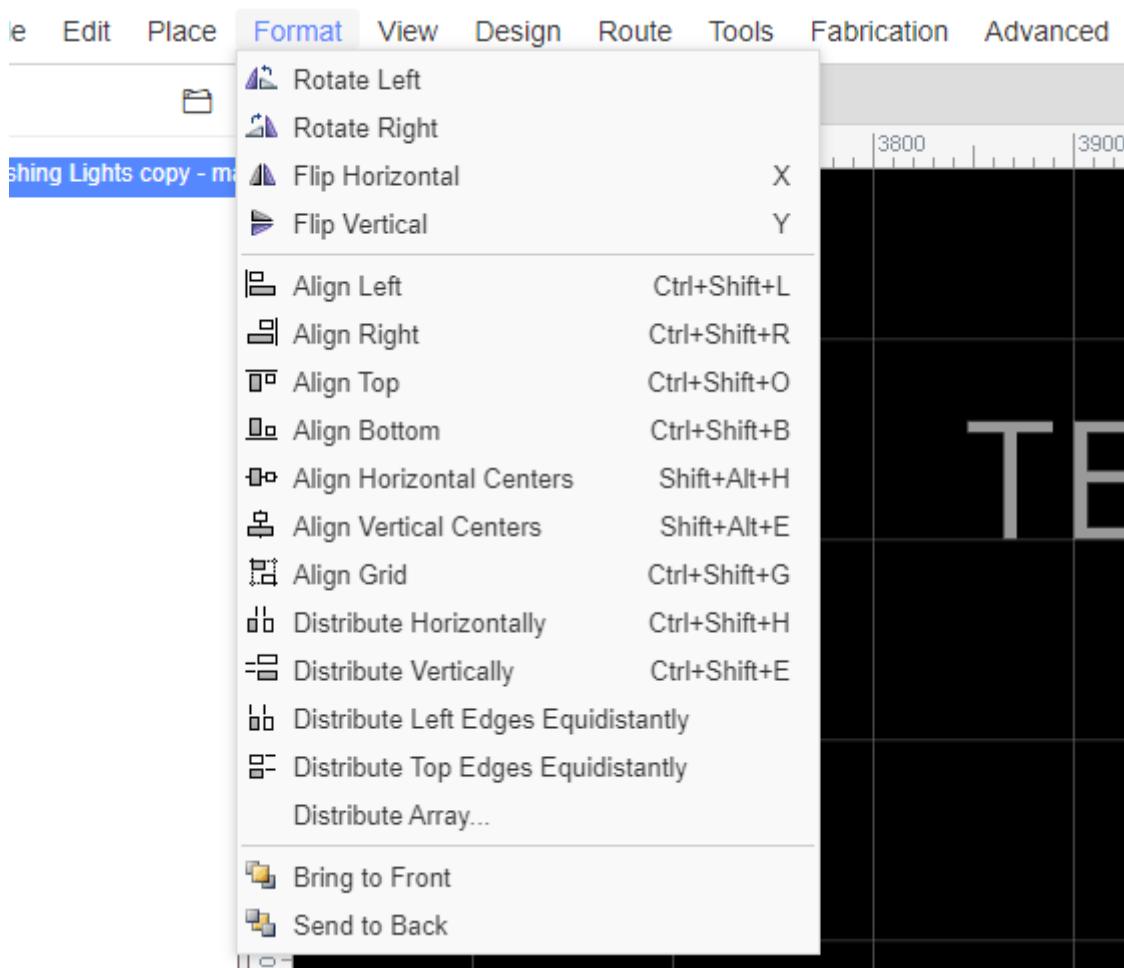
**Rotate and Flip > Flip Horizontal or Flip Vertical** from the toolbar,

or by pressing the default flip hotkeys: **X** to Flip Horizontal, **Y** to Flip Vertical.

Notice: Footprint doesn't support to flip.

## Align

EasyEDA provides many align option features, you can align your symbols or footprints very easily. Via: Top menu - Format - Align.

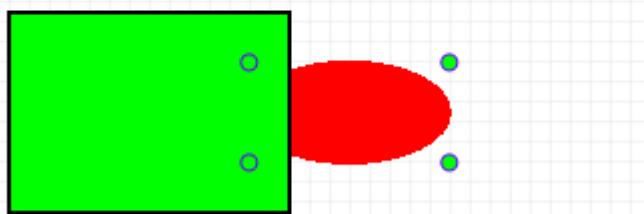
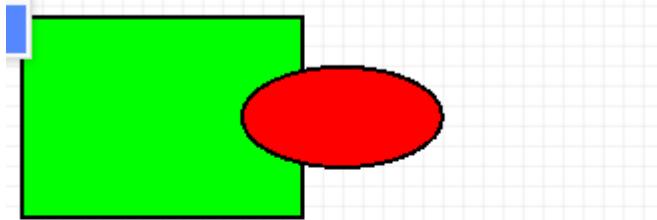


## Bring to Front and Send to Back

In the image below, both the rectangle and the ellipse are filled. Via: Top menu - Format - Bring/Send to Front/Back.

If you draw the ellipse before drawing the rectangle, the rectangle will overlap and therefore hide the ellipse. To reveal the ellipse, select the rectangle and then use Bring and Send function, you will see:

## Ellipse at the top



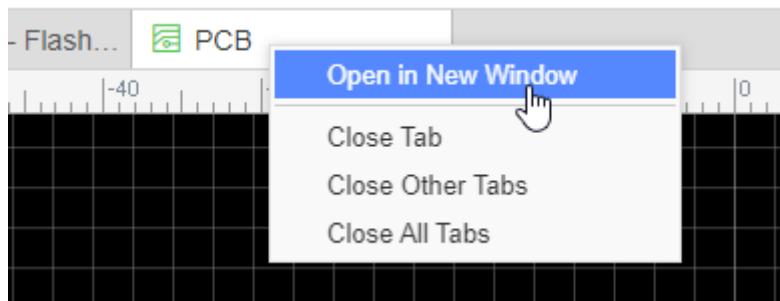
## Ellipse at the bottom

## Multiple Windows

Since v6.4.0, EasyEDA supports multiple windows design.

How do it works?

1. Open schematic and PCB
2. Right-click the schematic or PCB tab, click "Open in New Window"

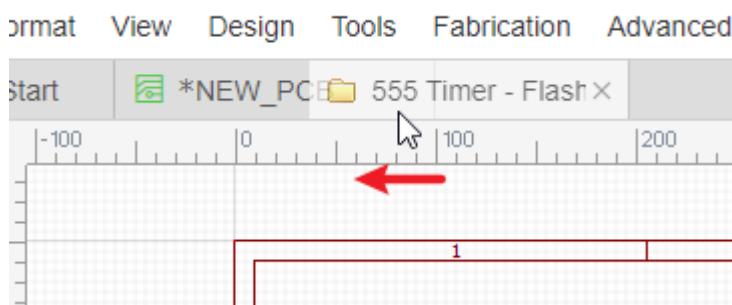


3. It will open this document in new window, then you can do the cross probe: Click the component, pads, click the Design Manager list, the "Cross Probe and Place" works too.

## Documents Tab Switch

It's easy to fit your documents tab location.

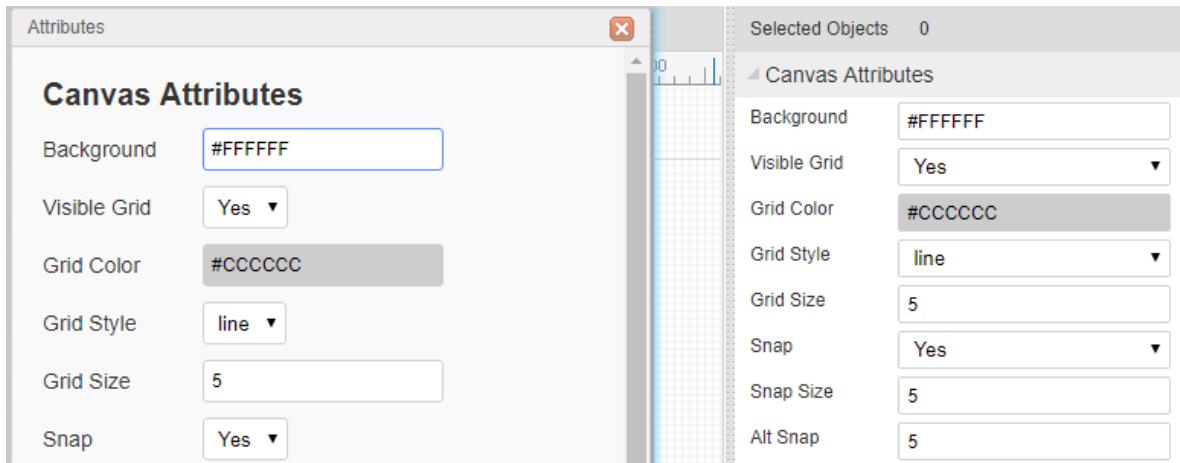
drag tab location, or use hotkey SHIFT+1, SHIFT+2



# Schematic Capture

During this tutorial we will create a simple Schematic design to guide you in using EasyEDA Schematic capture.

You can find the canvas Properties setting by clicking on any the blank space in the canvas.

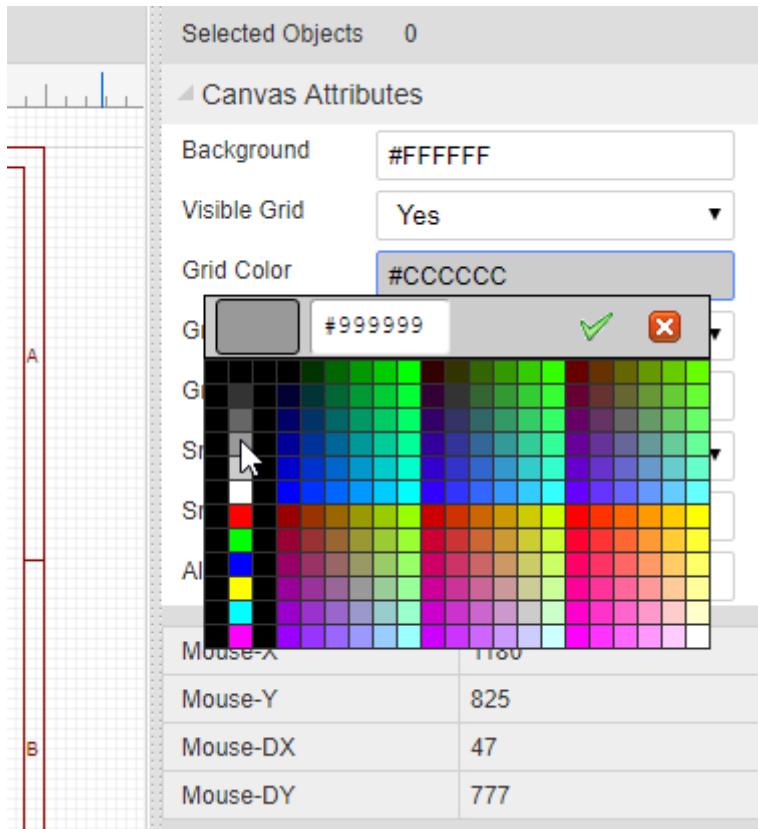


As described earlier, background and grid colours and the style, size, visibility and snap **attributes** of the grid can all be configured.

The canvas area can be set directly by the Width and Height or by using the available preset frame sizes.

## Grid:

- **Visible Grid** : Yes or No
- **Grid Color**: Any valid colour
- **Grid Style**: Line or Dot
- **Grid Size**: To ensure proper alignment of all EasyEDA parts, it is advisable to set in 10, 20, 100. the unit is pixel.
- **Grid** (and background) colour can be set directly by entering the hexadecimal value of the colour you want or by clicking on a colour in the palette that opens when you click on the colour value box:



### Snap:

- **Snap:** Yes or No. Pressing this key toggles switching snap to grid on and off.
- **Snap Size:** To ensure proper alignment of all EasyEDA parts, it is advisable to set in 10, 20, 100 but any valid number can work, such as 1, 5, 10.

It is strongly recommended that you keep **Snap = Yes** all the time. Once items are placed off-grid it can be very difficult to reset them back onto the grid. Off-grid placement can result in wires looking as though they are joined when in fact they are not and so causing netlisting errors that can be hard to track down.

If you need to draw detailed parts of new symbols or footprints that need to go between grid points, try to reduce the grid spacing to draw these elements and then reset the grid back to your chosen default value as soon as you have completed that part of the drawing. Setting Snap=No should only really be used as a last resort.

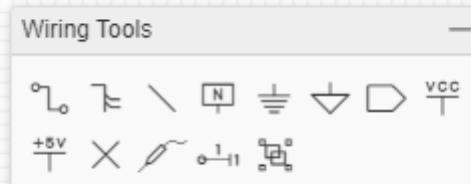
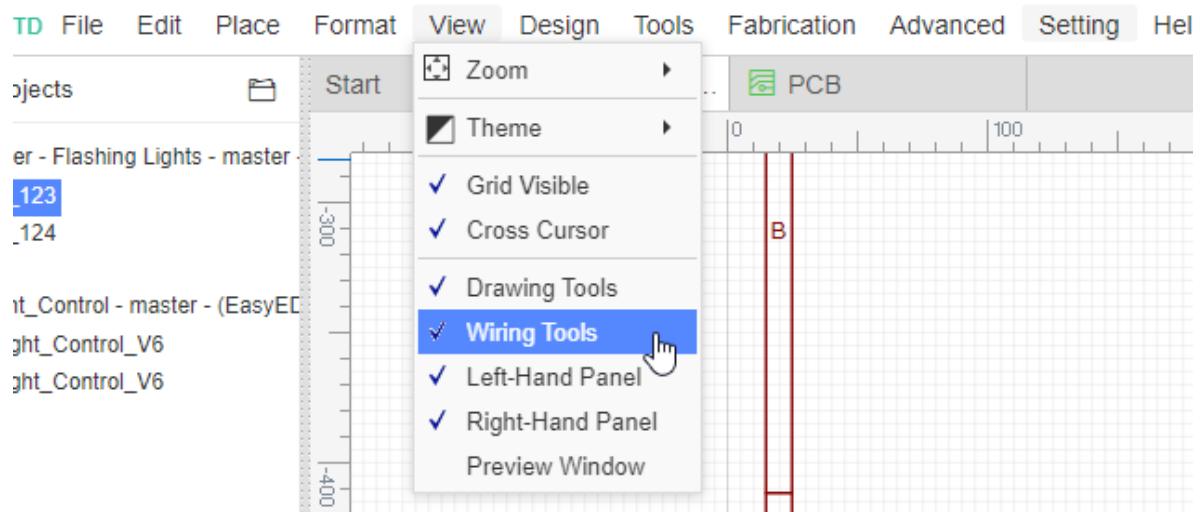
- **ALT Snap:** Snap size when pressing the **ALT** key.

## Wiring Tools

---

If you have hidden your tools , you can open them from here:

Top toolbar **Top Meun > View > Wiring Tools...**



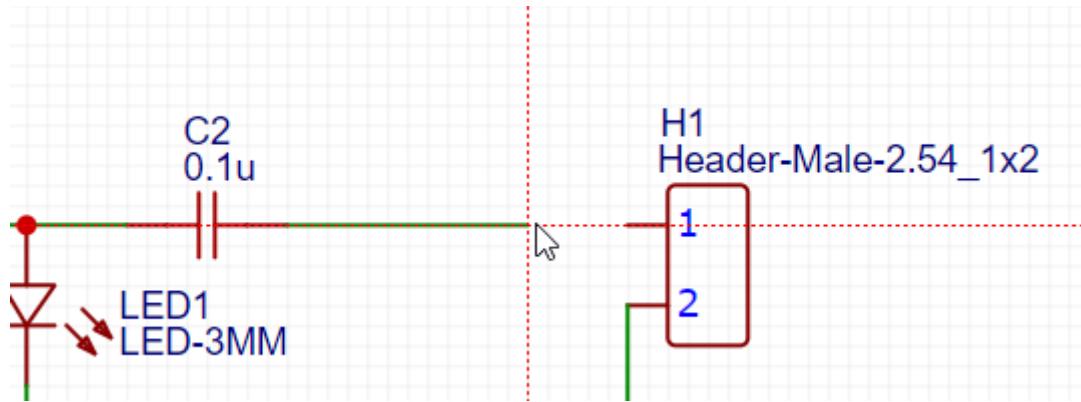
**Note:** All of the commands in Wiring Tools are electronics related. Don't use a wire when you just need to draw a line, shape or an arrow: use Drawing Tools instead.

## Wire

There are three ways to enter the wire mode in EasyEDA.

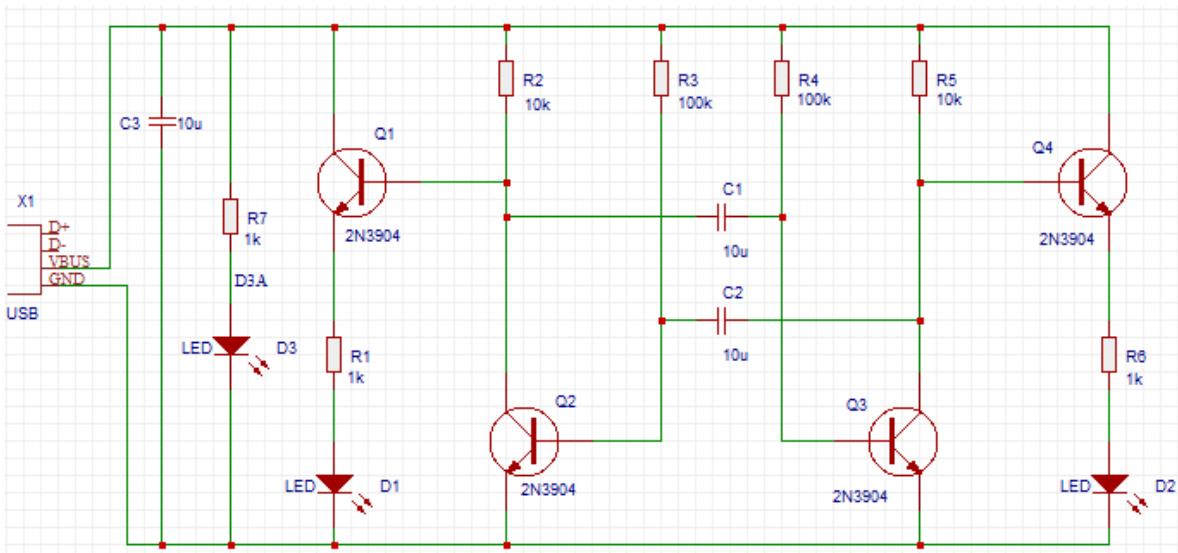


1. Click the **Wire** button from the **Wiring Tools** palette.
2. Press the **w** hotkey.
3. Click on the end of a component pin (where the grey pin dot appears if you select the component):



EasyEDA automatically enters **Wire mode**.

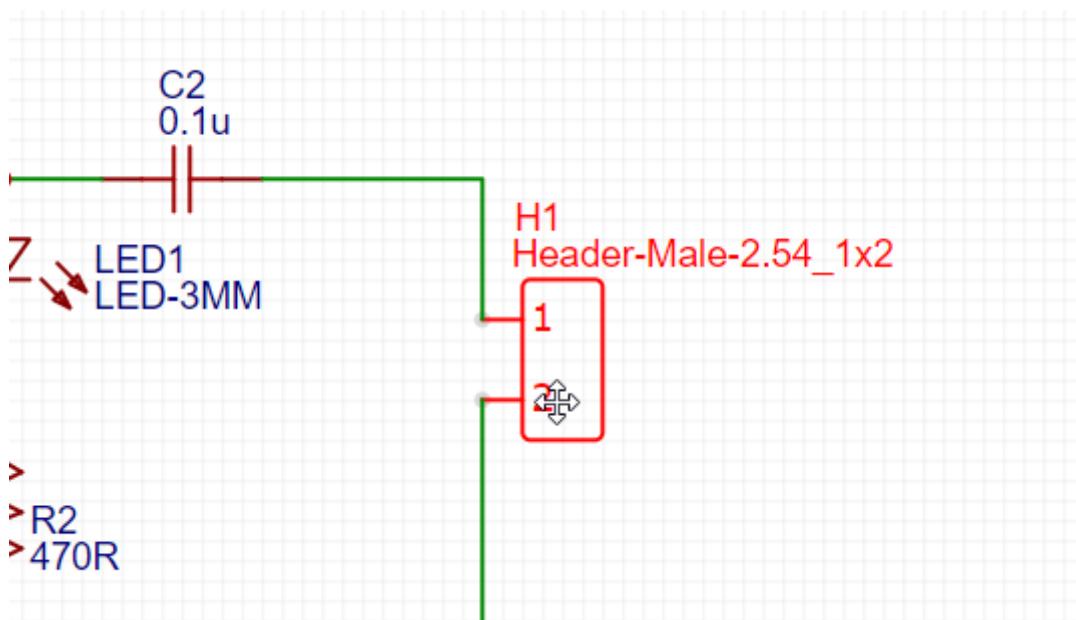
Here is a screenshot of the **Astable Multivibrator LED project schematic** after wiring:



### Moving Components and Wires:

If you place a component, such as a resistor, on top of a wire then the wire breaks and reconnects to the ends of the component.

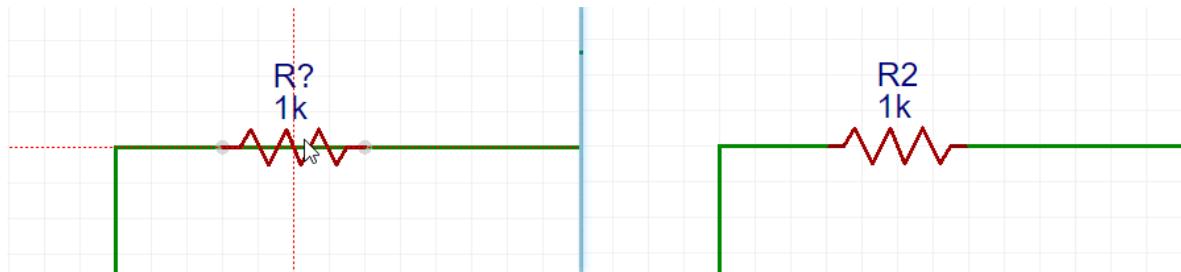
When moving selected components using the mouse, they will drag attached wires with them ("rubber band") to some extent but please be aware that the rubber banding feature has some limitations. When moving selected components most wire will move vertically and horizontally. Using the arrow keys will not rubber band. Selected wires do not rubber band.



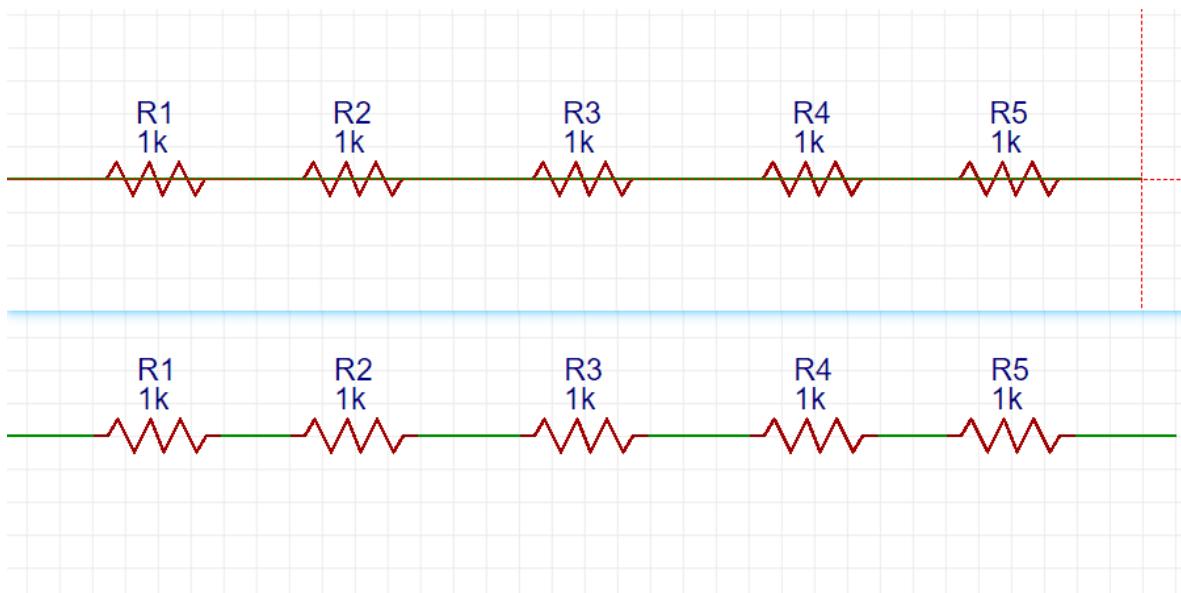
A selected wire can be moved directly by clicking on it using the mouse or by the arrow keys. If a wire is selected by clicking on it using the mouse then green grab handles will appear at the ends and vertices.

### Auto adjust connection

If you put a resistor or capacitor on a wire, the wire will auto connect the pins as below:

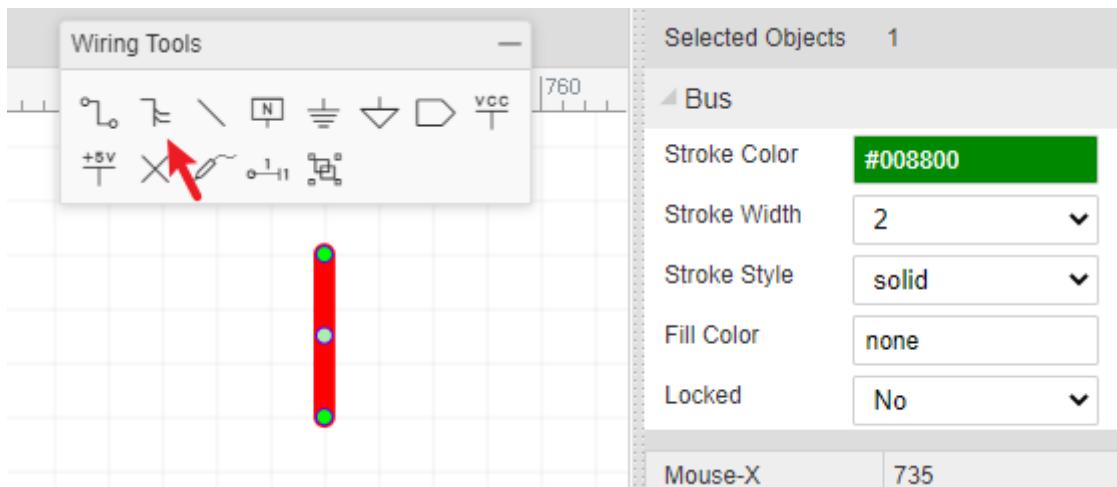


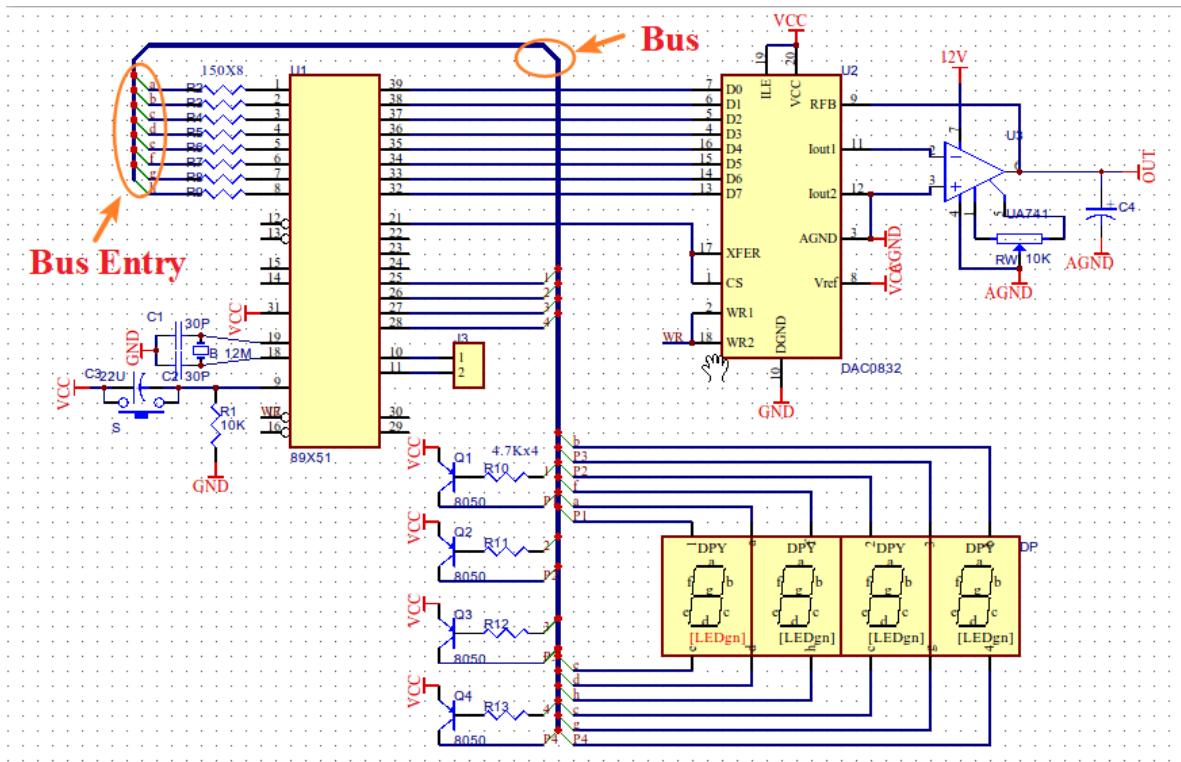
When you want to wiring a series of resistors which are in a row, you can just wire through them, and then you will find they all be connected.



## Bus

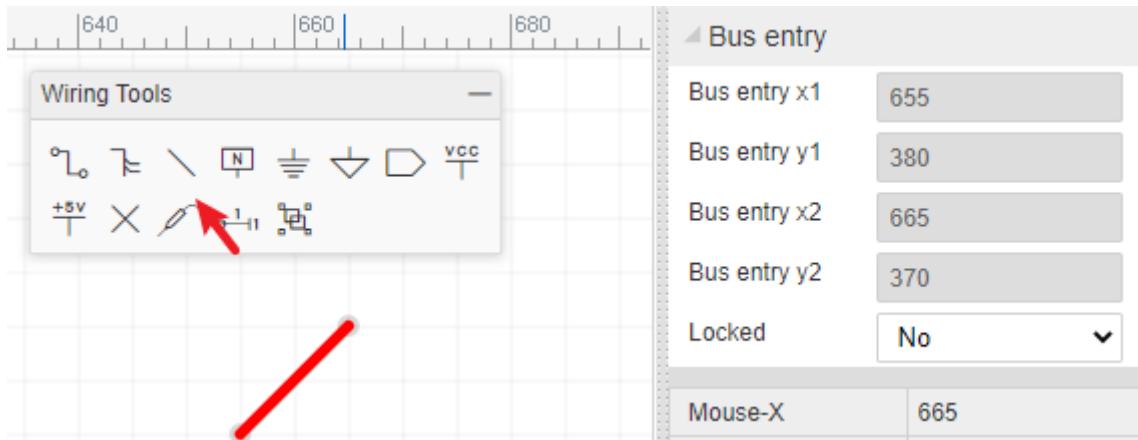
When you design a professional schematic, perhaps it will use a lot of wires. If you wiring one by one, much time would be wasted, and then you need to use **Bus**.





## Bus Entry

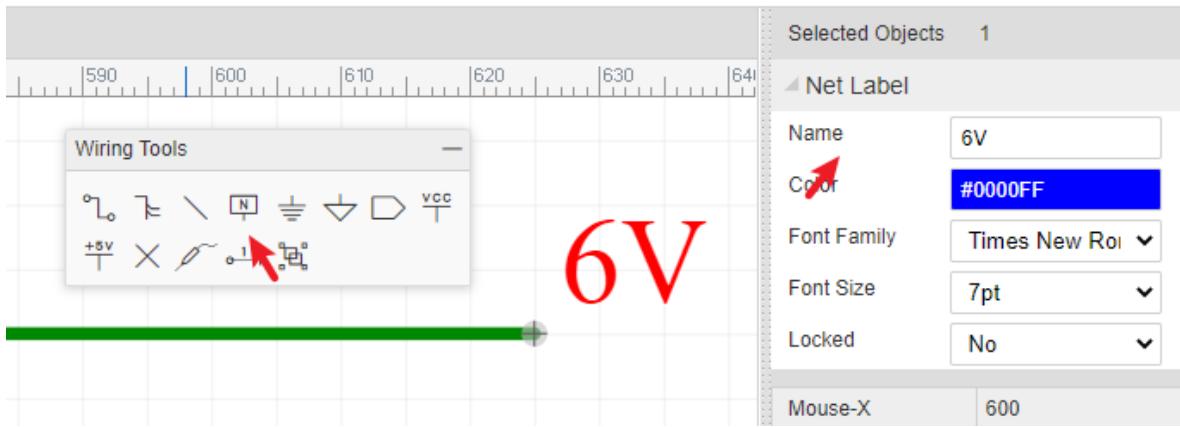
If you decide to wire with `Bus`, the `Bus Entry` must connect to Bus and other nets with wires, such as in the above image.



The "Bus" and "Bus Entry" just for the indication, because when you place Bus and Bus Entry, you have to place the netlabel on the Bus Entry dot point.

## Net Label

**NetLabel** can be used to give your wires names to help you find them and identify any misconnections. You can find the **NetLabel** from the Wiring Tools palette or by using the **N** hotkey. When selecting the netlabel, you will find its attributes in the right hand Properties panel:

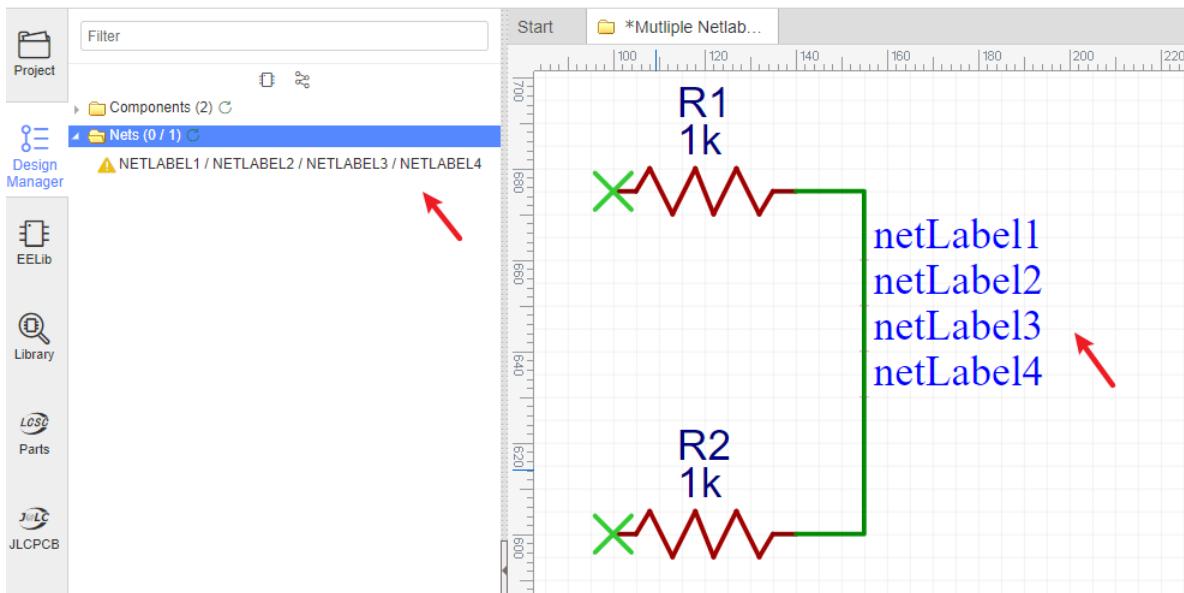


You can change its name and colour. If you only want to change its name, it may be easier to just double click the netlabel.

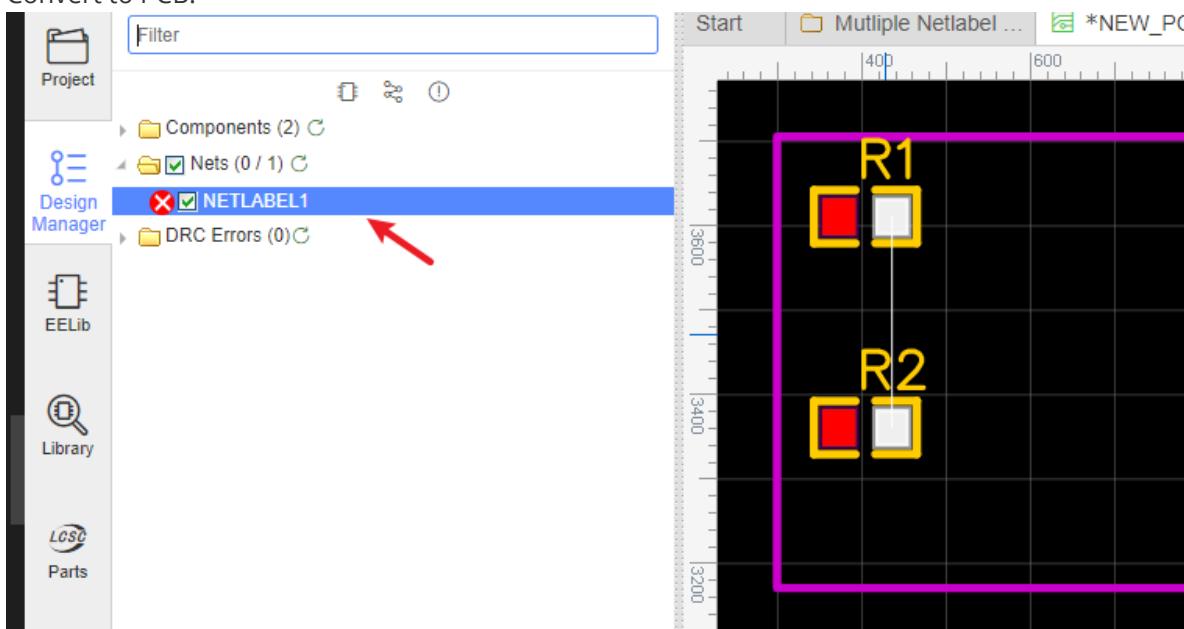
### Multi-NetLabels in One Wire

EasyEDA support multi-netlabel in one wire now.

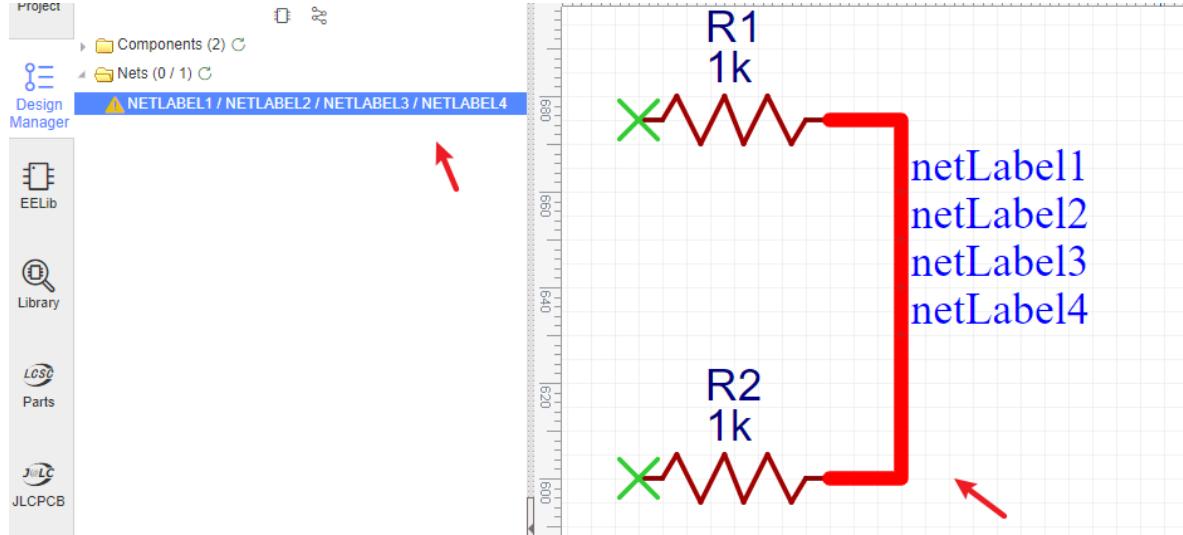
When you convert the schematic to the PCB, the editor will choose the first netlabel you placed as the net name for this wire, as below NETLABEL1.



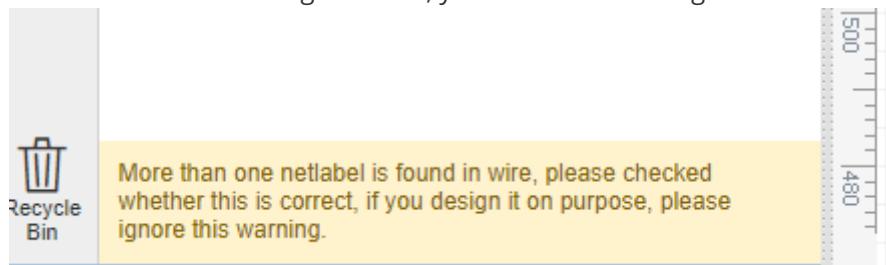
Convert to PCB:



As above image, when you click anyone netlabel's name in the design manager, the wire will be highlighted.



And check the bottom right corner, you will see a warning:

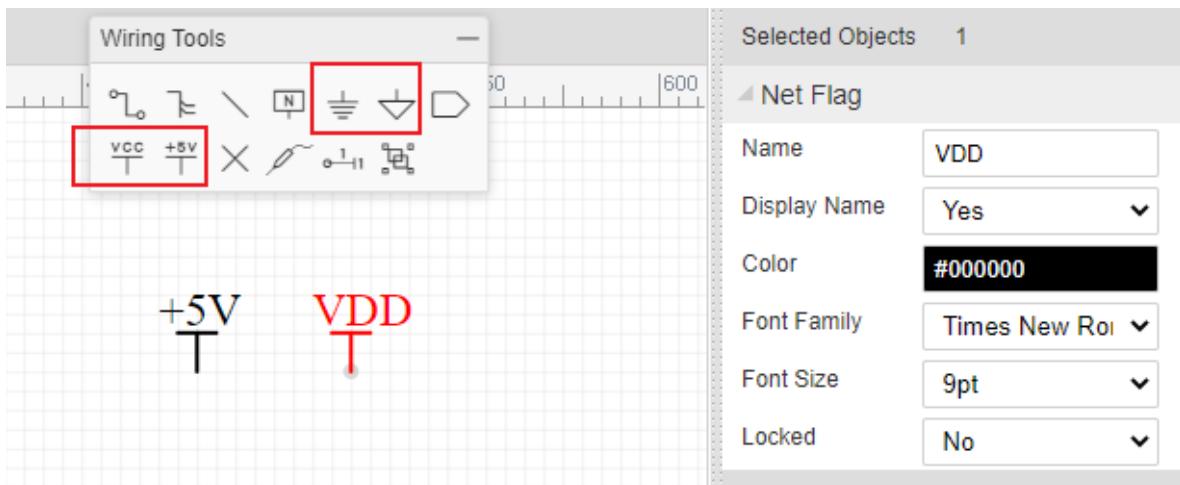


#### Notice:

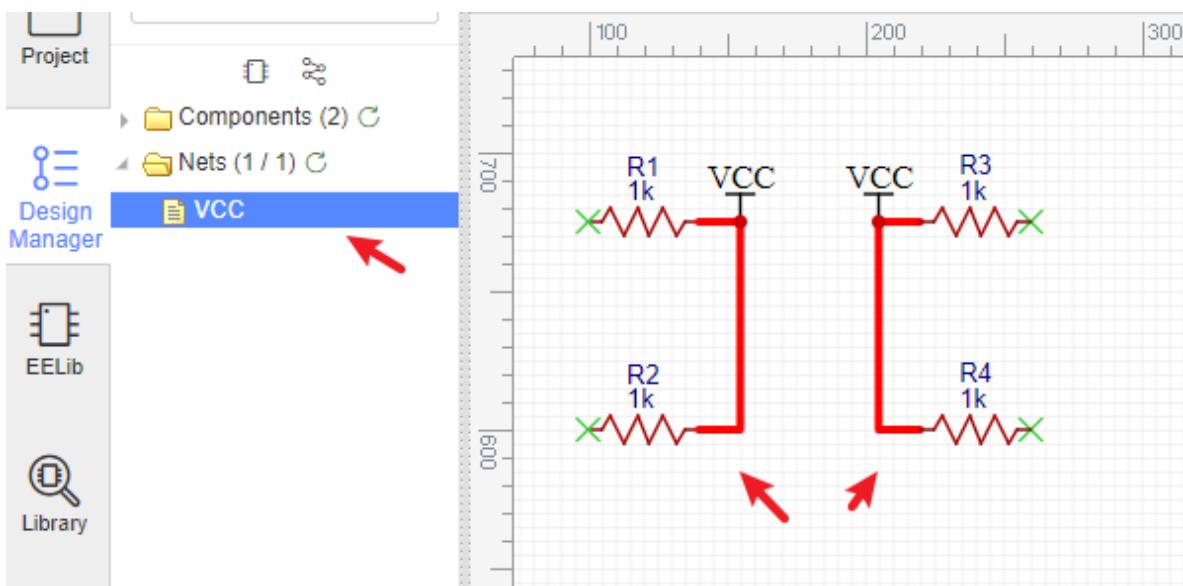
- If wire 1 has 3 netlabels A B and C, and wire 2 has netlabel A, then wire 1 and wire 2 are the same net.
- Netlabel/Netflag/Netport/volprobe only support English characters and letters, and Arabic numerals.
- If a part prefix is P1, which has two pins, it will have two nets "P1\_1" and "P1\_2" by default, if you place a netlabel named P1\_1 at other wire which is not connect with P1 pin1, the default "P1\_1" will change to "P1\_1(1)" for avoid the wrong connection with netlabel "P1\_1".

## Net Flag

**NetFlag** is the same as NetLabel, you can find the NetFlag from the Wiring Tools palette or using the **ctrl+G** hotkeys for **GND** or **ctrl+Q** for **VCC**. You can also change its name, for example from **+5V** to **VDD**:



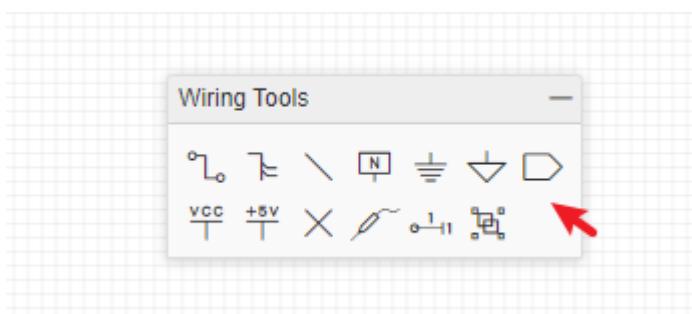
When appear two and more Netflag or Netlabels which are the same name, they will connected with each other.

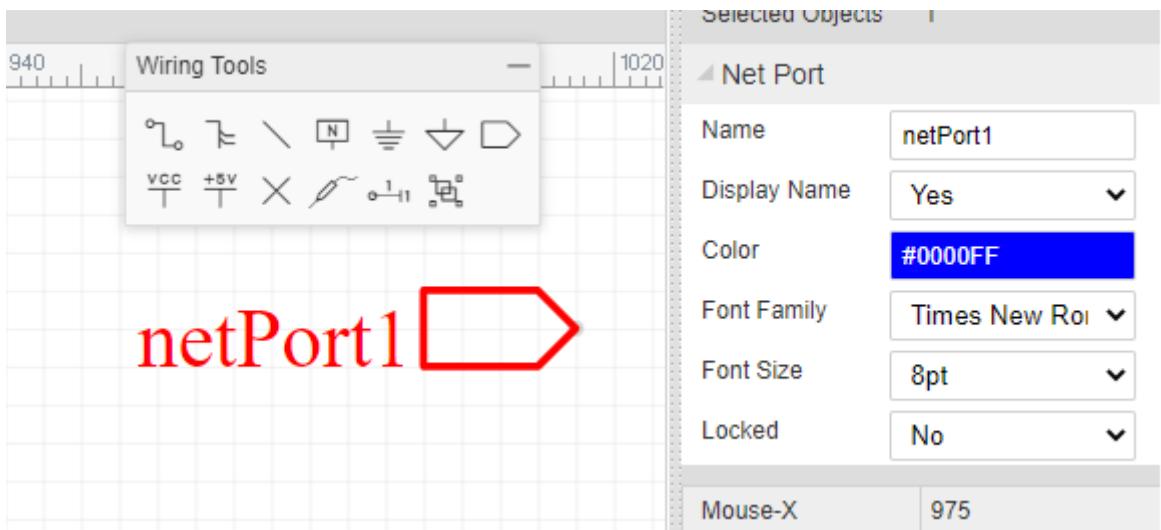


Wiring Tools palette provides NetFlag: Digital GND, Analog GND, VCC and +5V for your convenience.

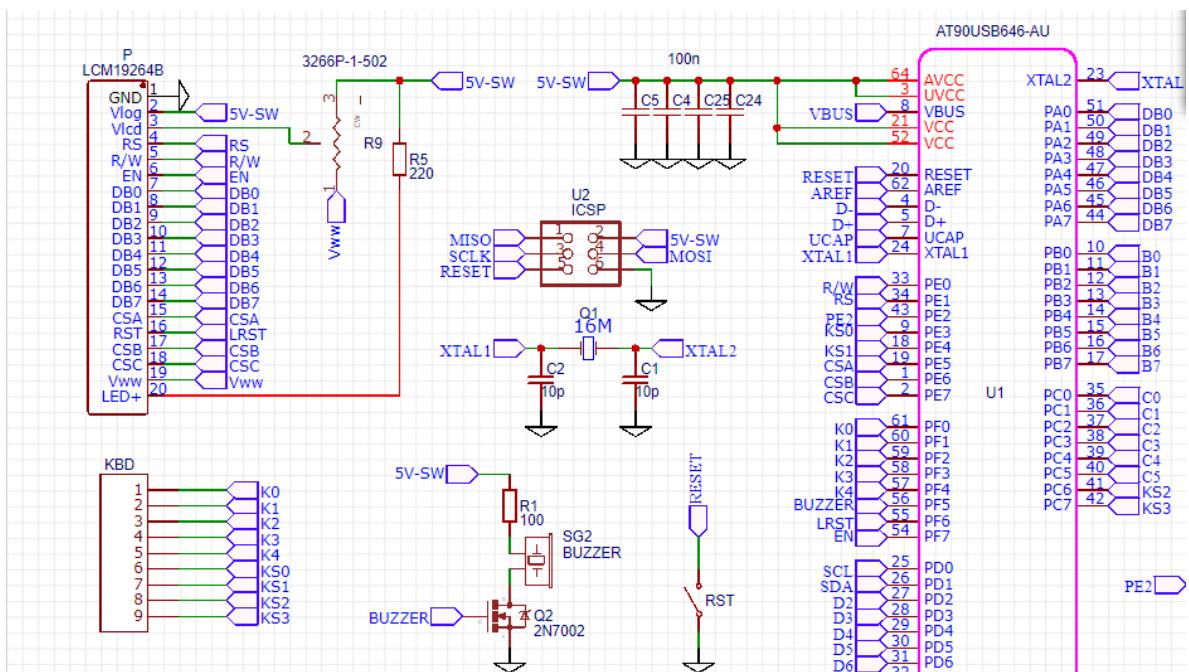
## Net Port

At EasyEDA, Net Port works like Net Label, it doesn't differentiate the input and output net port. When you don't want to route too many wires, how about trying [Net Port](#):



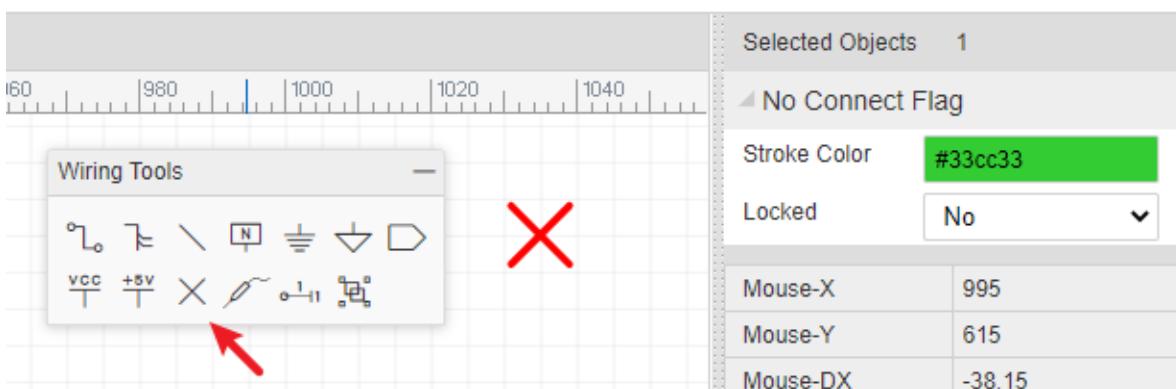


It will make your schematic look more clean, and you just need to set each Net Port a net name.

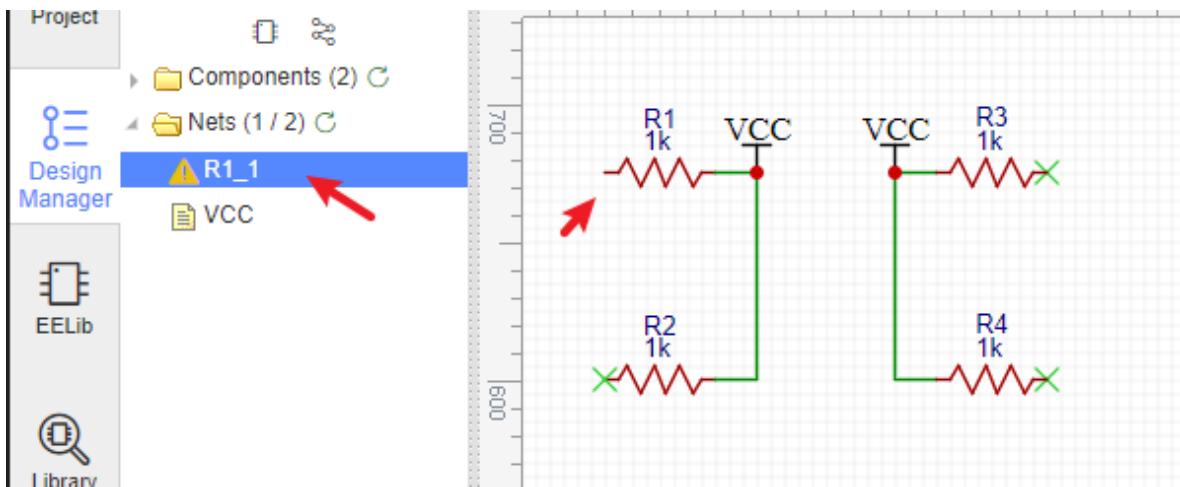


## No Connect Flag

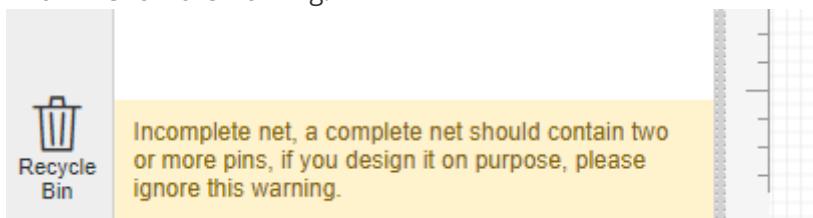
You can find the `NO Connect Flag` via wiring tool,



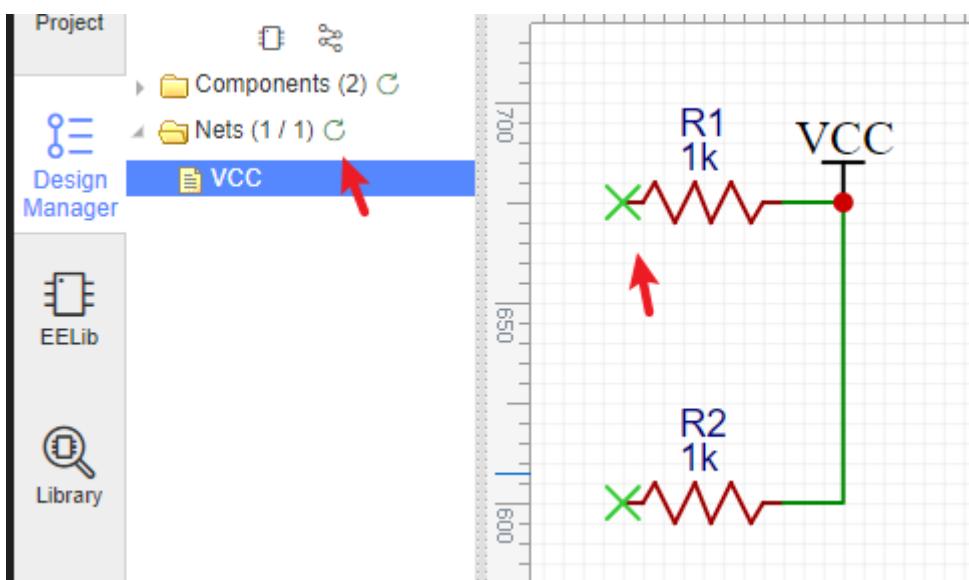
In the below schematic, if you don't add a `NO Connect Flag`, there is an error flag in the nets collection of the design manager.



And will show the warning:

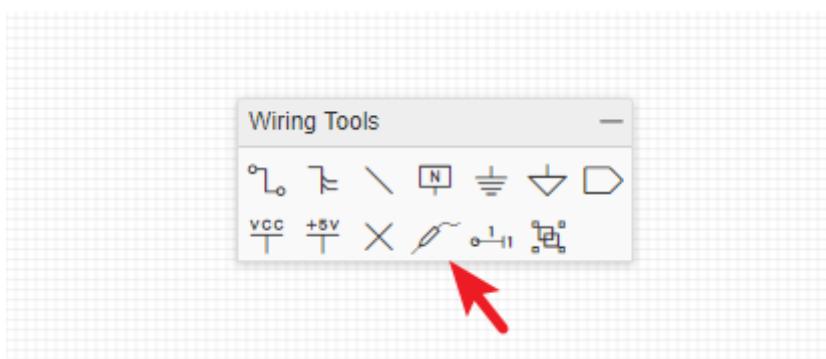


After adding a `NO Connect Flag`, and then refresh the Nets folder, the error disappears.

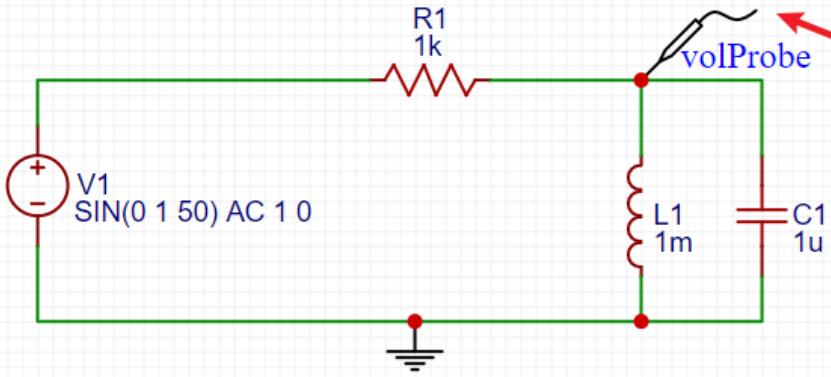


**Note:** `NO Connect Flag` only works on the symbol's pin directly.

## Voltage Probe



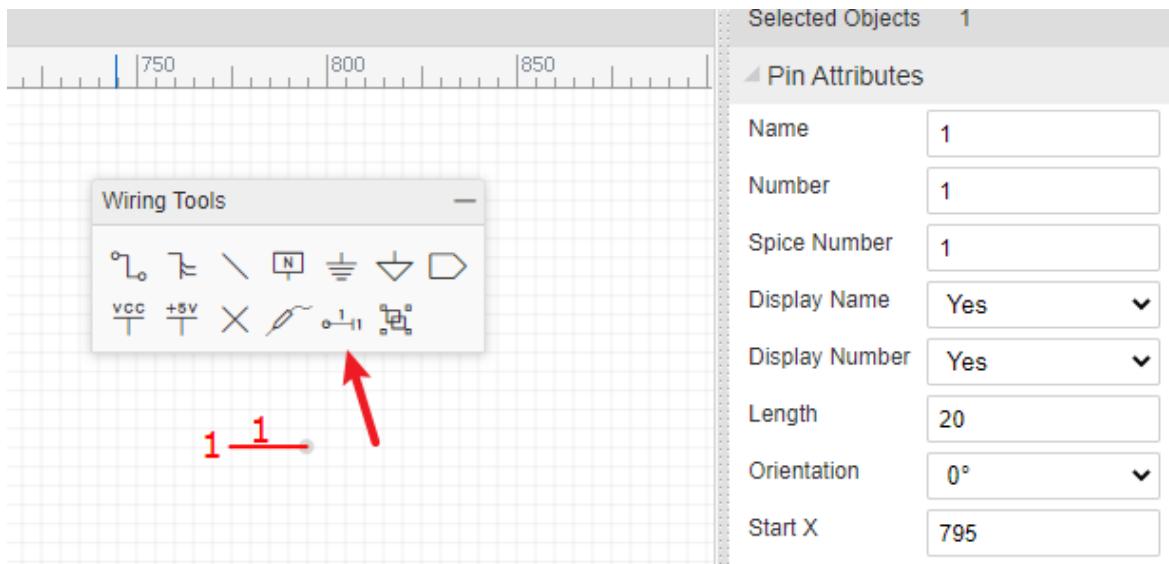
EasyEDA provides a simulation feature for the schematic. After the simulation is running, you will see the waveform where you placed the voltage probes in the circuit.



For more detail about the simulation, please check the [Simulation](#) section.

## Pin

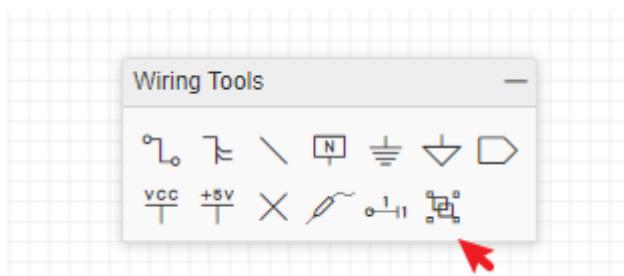
When you create a new symbol in schematic and schematic lib, you must use **Pin** to create pins for the new symbol, otherwise your symbol can't be wired with wires.



For more information please refer to the [Symbol Library - Create Symbol](#) section.

## Group/Ungroup Symbol

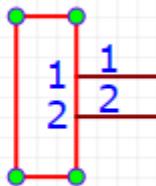
On the **Wiring Tools** palette there is the **Group/Ungroup Symbol...** button.



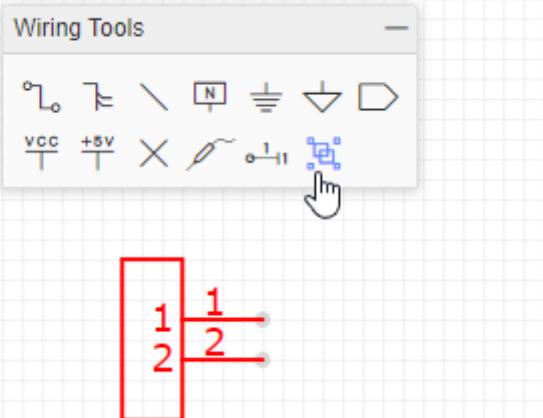
Just like the **Symbol Wizard**, this tool is also for you to quickly create schematic library symbols.

Here's how.

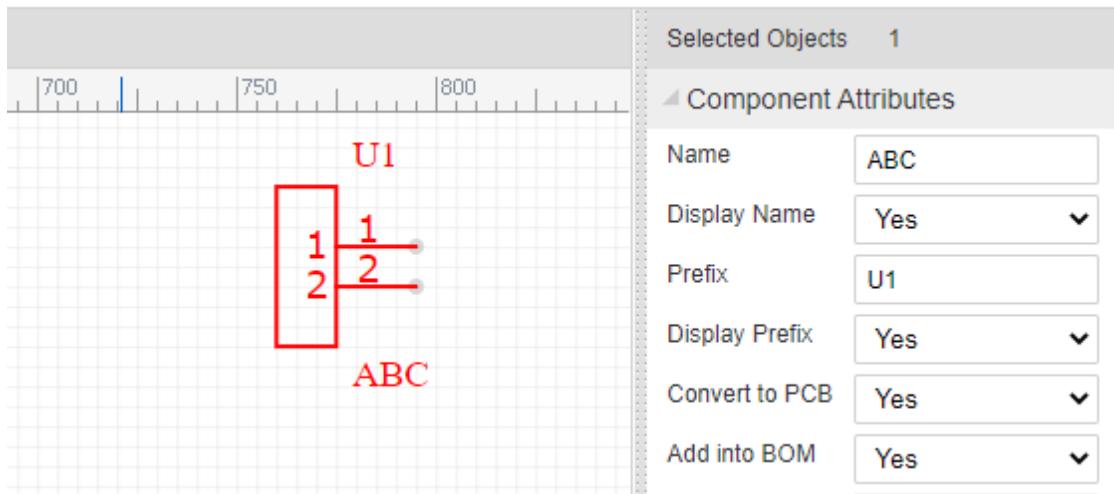
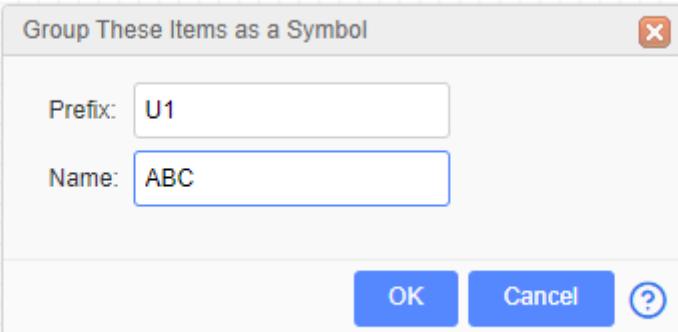
- Place Pins and other objects such as rectangle



- Select them, and click the "Group/Ungroup Symbol" icon



- Type the prefix and name, press OK, done. A part is created.



So what does Ungroup do? Try selecting a symbol and then click the Group/ungroup command to see what happens!

**Note:**

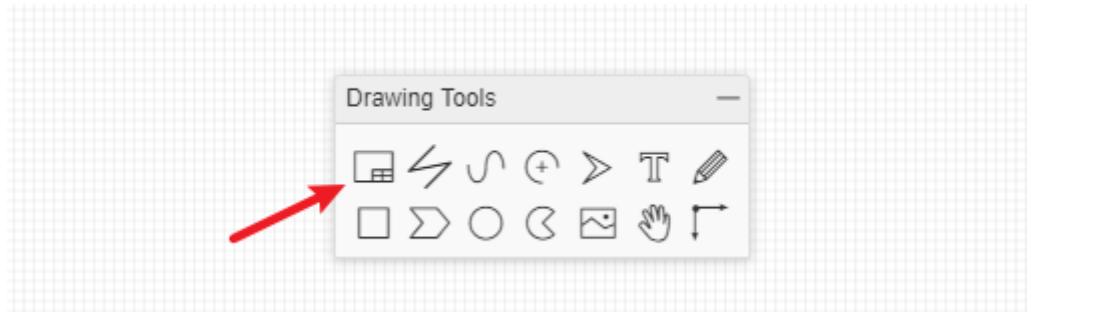
- The symbol you created in the schematic will not be saved in the personal libraries, if you want to use it repeatedly, please create a Symbol via: Top Menu - File - New - Symbol.

# Drawing Tools

## Sheet Setting

It is now possible to add design notes to the frame and the frame selection, for example A4, which can assist in aligning and improve the look of printed schematics and PCB designs.

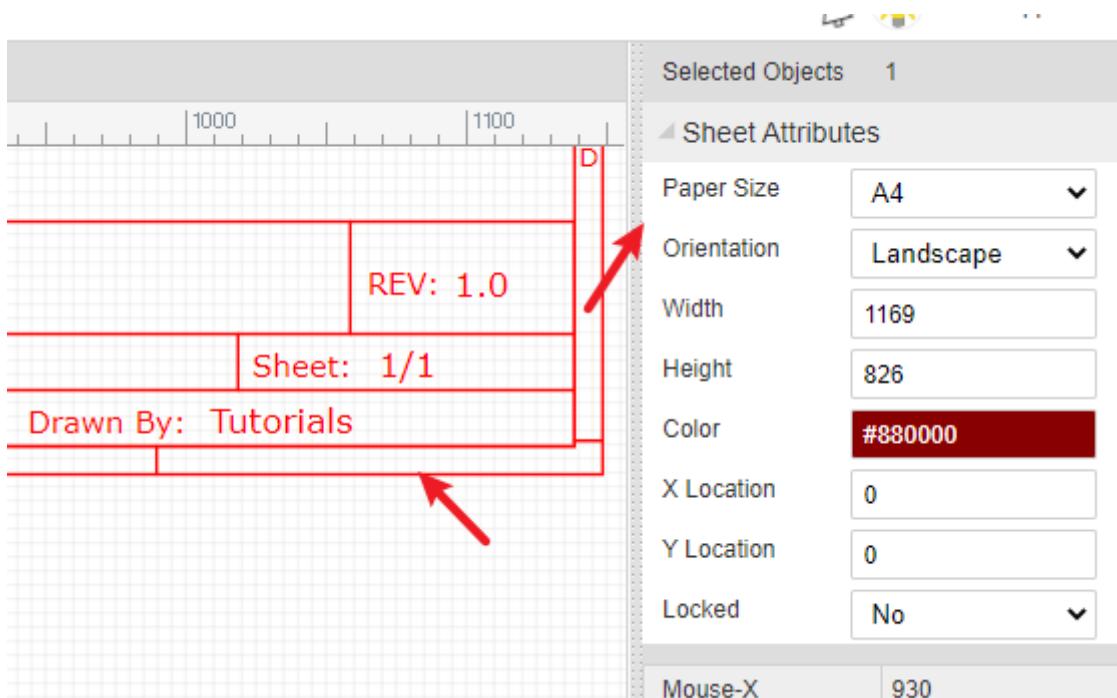
Click the frame/drawing/document button like in the image below:



And you can edit the blue text when you've selected the text attributes or double clicked it.

The bottom right zone can be selected and dragged or the frame can be dragged and deleted.

When you've selected the bottom right zone, you can edit the sheet attributes:

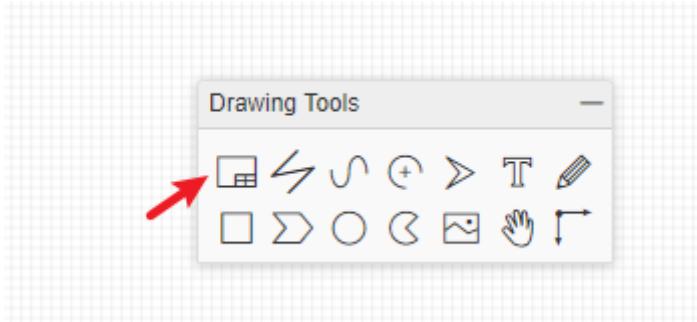


## Custom Sheet

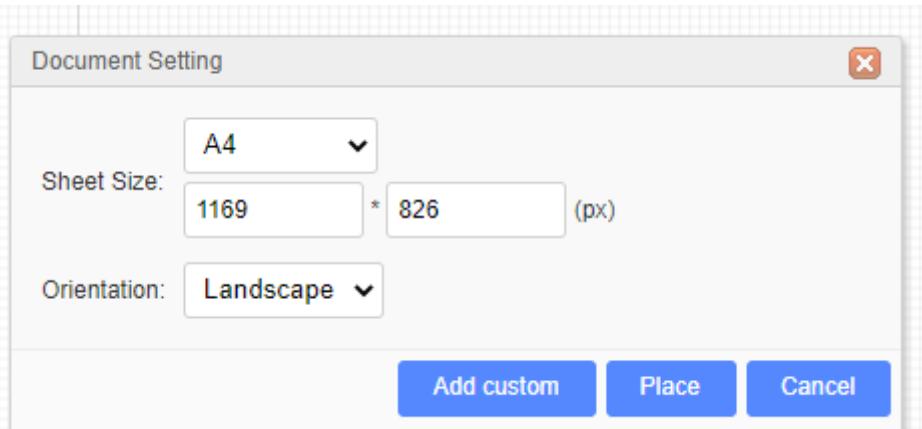
EasyEDA supports the schematic diagram drawing frame required by custom. At present, custom drawings need to be placed manually, and automatic reference of custom drawings is not supported when creating new schematic diagram.

How to create:

1. Click the "Sheet Setting" button at "Drawing Tool".



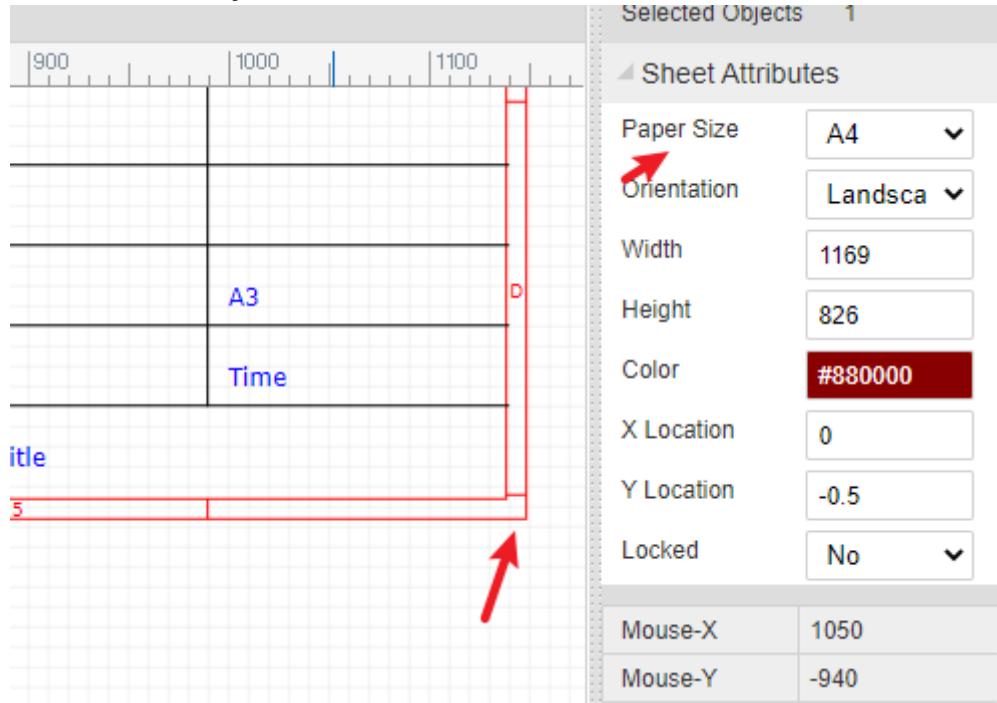
2. Click "Add Custom" button.



3. It will create a new symbol editor, you can edit the table by line as you want, as below:

|          |            |               |      |   |
|----------|------------|---------------|------|---|
|          | MPN        |               |      |   |
| Verifier | Type       |               |      |   |
| Draw by  | BoardType  |               | A3   | D |
| Revision | Department |               | Time |   |
| Date     | Company    | Project title |      |   |

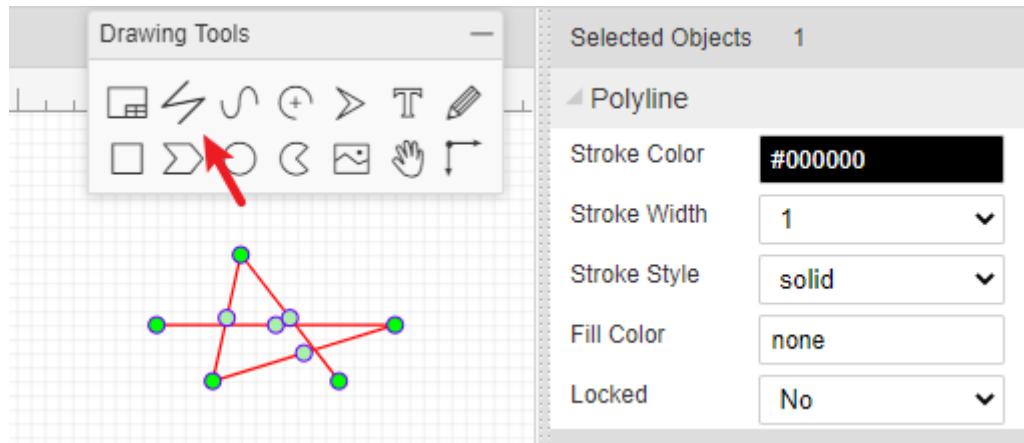
4. Select the outline, you can edit its size.



5. Save it. You can place it in schematic such as a part at "Library".

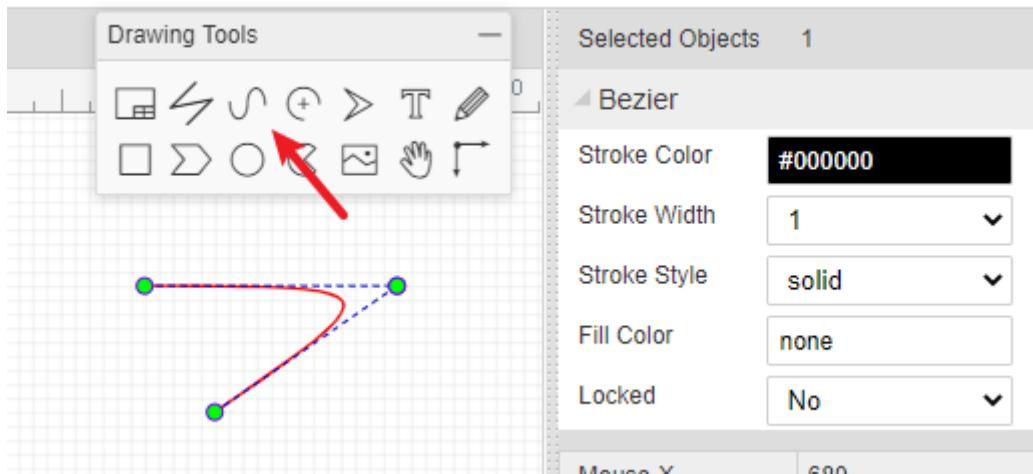
## Line

In the Schematic editor, you can draw a line with any direction. You can change its attribute as in the image below:



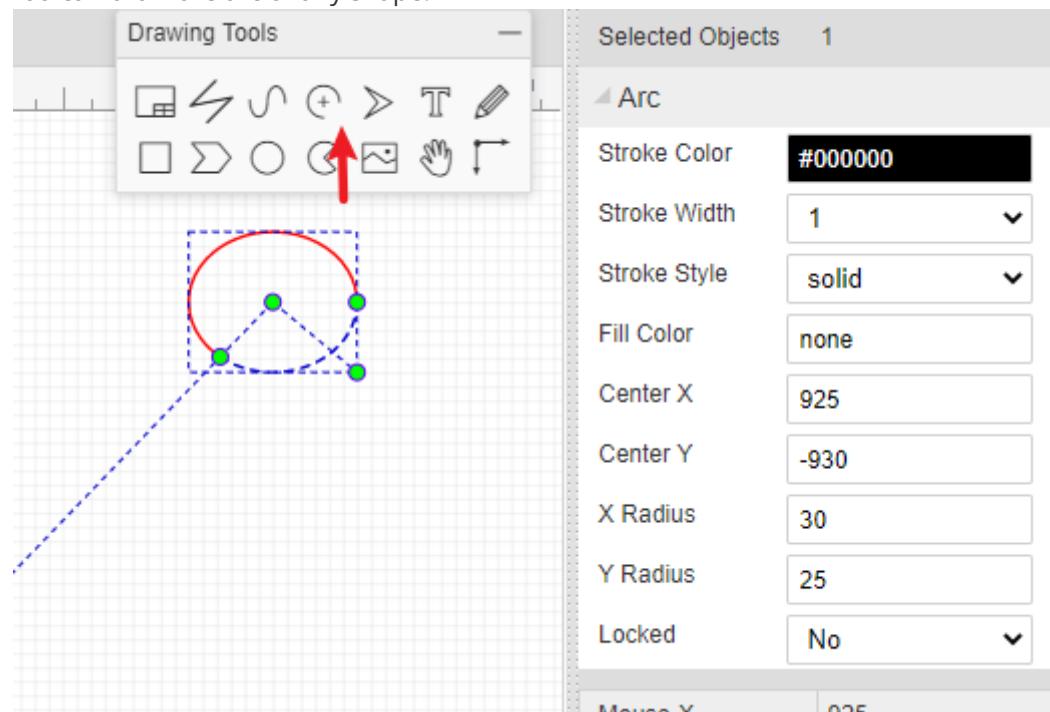
## Bezier

With this tool, you can draw a pretty cool pattern.



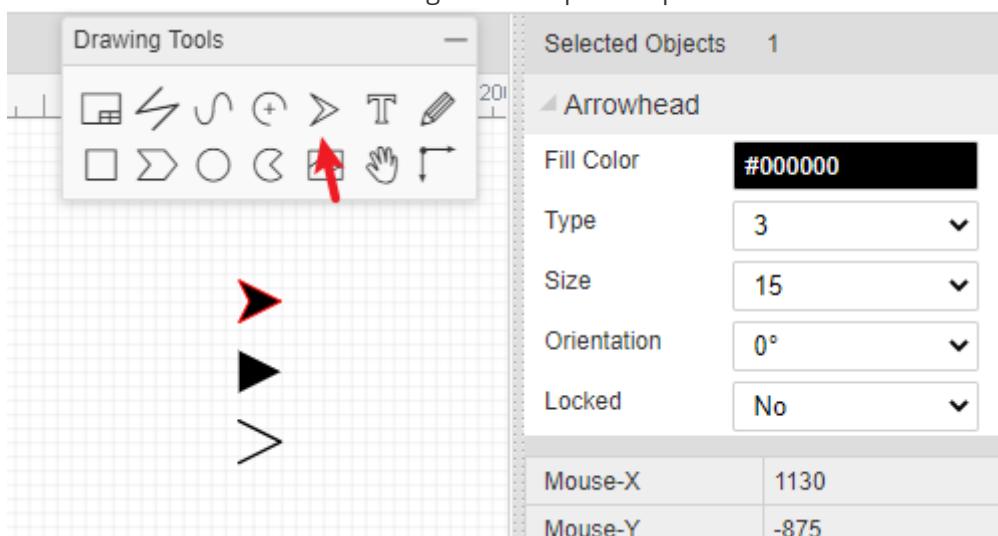
## Arc

You can draw the arc of any shape.



## Arrow Head

You can add arrow head to marking text or important part.

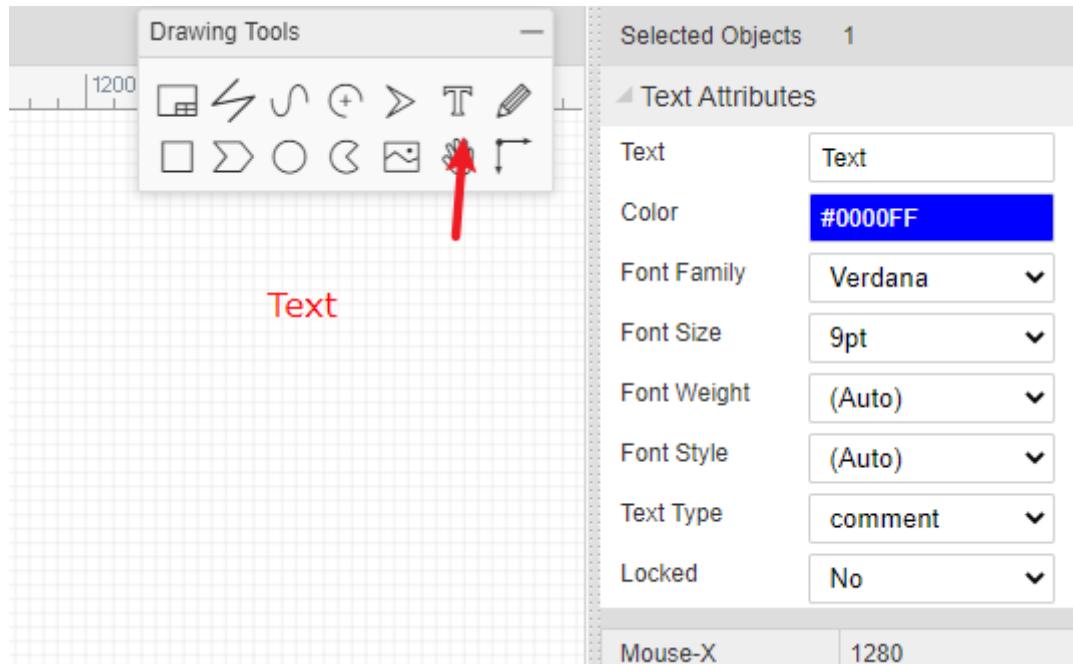


# Text

Text attributes provide many parameters for setting:

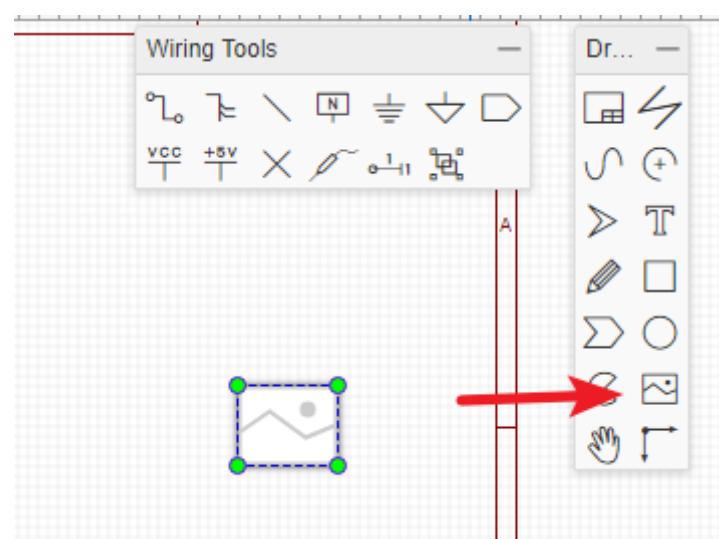
- **Text:** You can change text in inner box or double click the text. For every new text, the default text is `Text`.
- Color:** Defines text color.
- Font-family:** It provides 12 fonts for choosing.
- Font-Size:** Defines Text size.
- Font-weight:** Defines Text weight.
- Font-Style:** It contains (auto), normal, italic.
- Text type:** types include comment and spice.

The editor will remember your last text parameters.

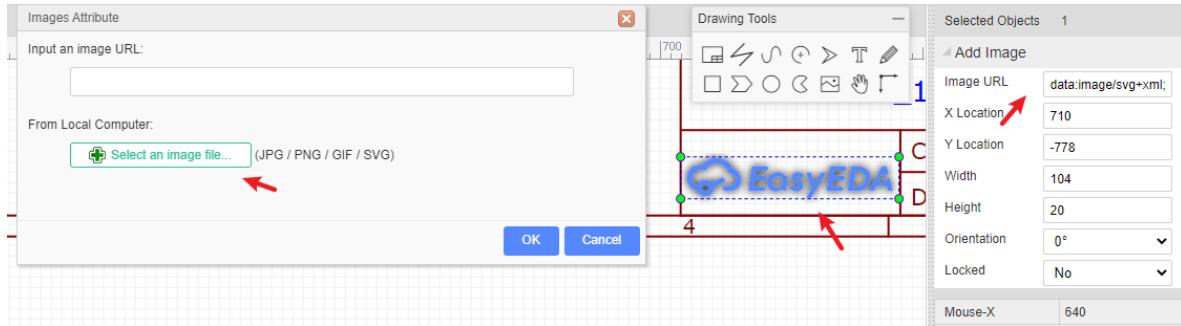


# Image

When you select Image from the Drawing Tools palette, an image place holder will be inserted into the canvas:



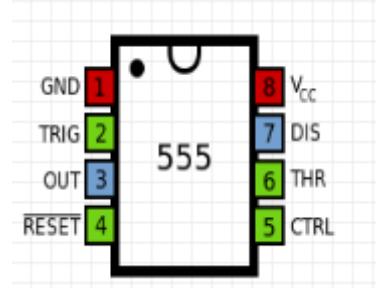
Select the place holder, so you can see the image's attributes in the right hand Properties panel:



Set the URL of your image. For example, setting the URL to:

[http://upload.wikimedia.org/wikipedia/commons/thumb/c/c7/555\\_Pinout.svg/220px-555\\_Pinout.svg.png](http://upload.wikimedia.org/wikipedia/commons/thumb/c/c7/555_Pinout.svg/220px-555_Pinout.svg.png)

will make your image look like this:

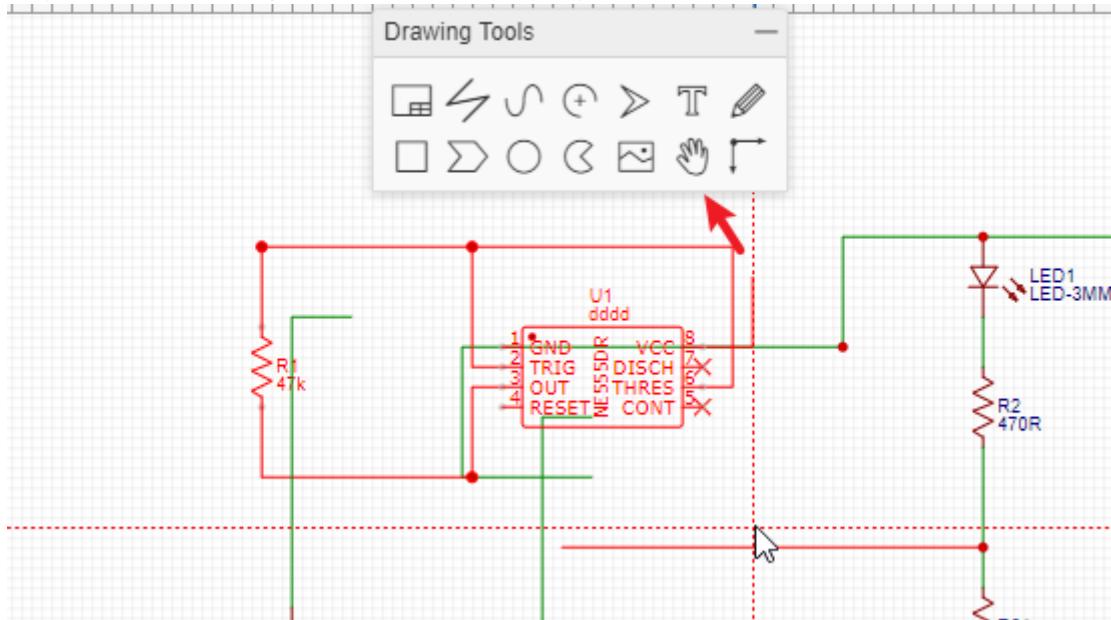


Please note: at present, EasyEDA cannot host images, so you need to upload your images to an image sharing site.

## Drag

If you want to move some kind of parts and wires, you can use drag, hotkey D.

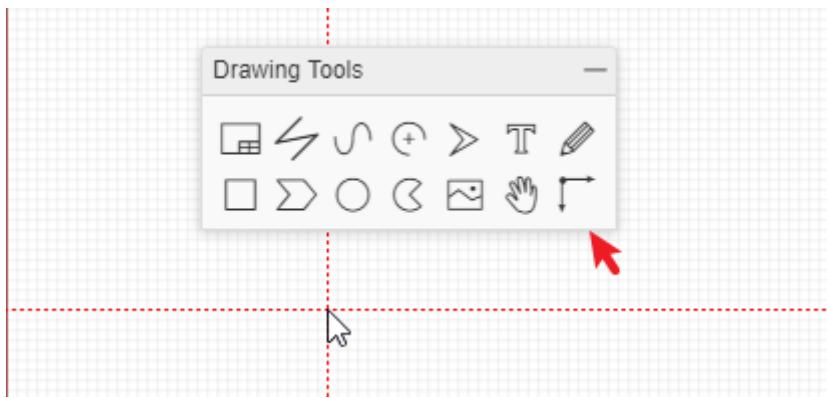
Or you can select the parts and wires area firstly and move them.



## Canvas Origin

Canvas origin default is set at left top corner of the schematic sheet, but you can set it where you want via Canvas Origin.

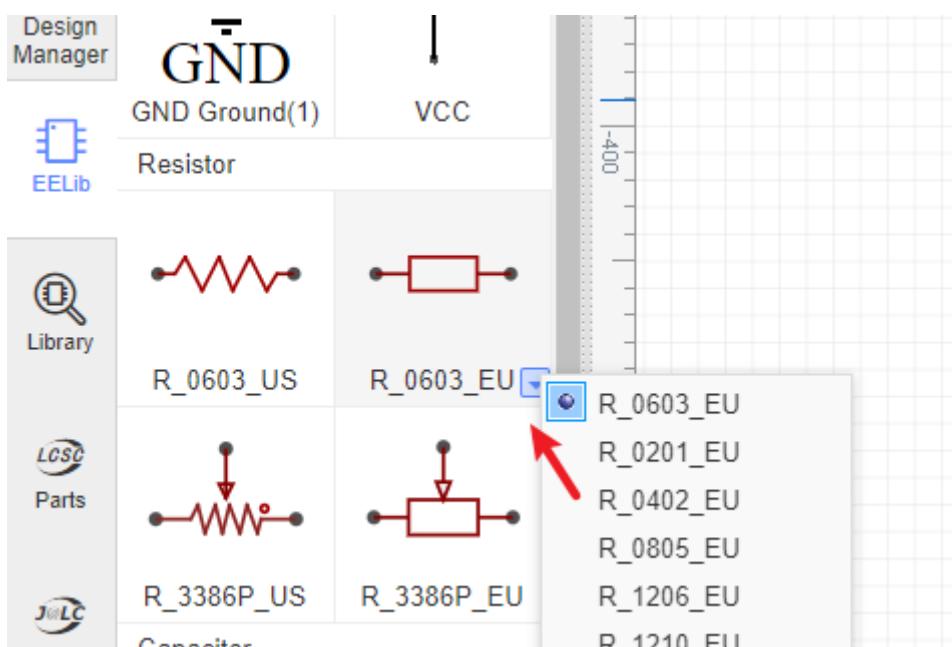
For another way to set canvas origin, you can try **Top Menu> Place > Canvas Origin**.



# Libraries

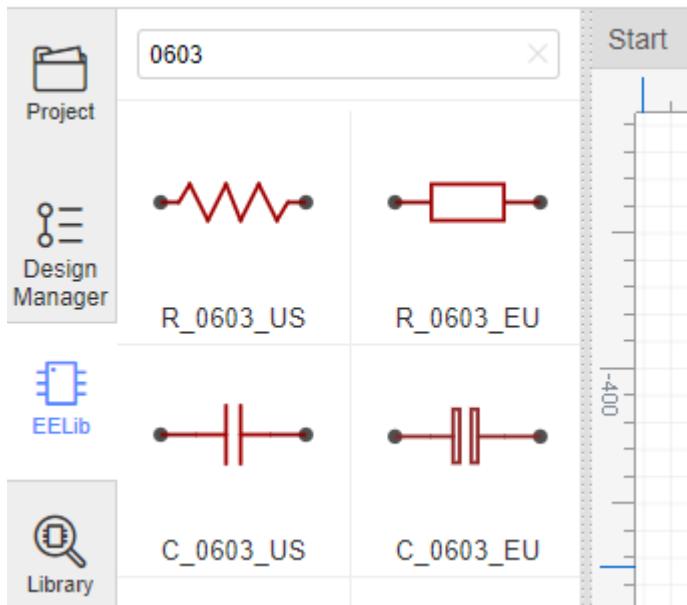
## EELib

That contains ready made symbols for a wide range of components and which can be simulated.



Many of these components have optional US and EU style symbols, we split them, so you can select those you like. Click on the drop down list or right click to popup the context menu, it contains many footprints or parameters. EasyEDA will remember your choices for the next time.

Don't forget to use Filter to locate a component fastly. For example, you just need to type `0603` to find all of resistors:



## Library

EasyEDA provide a lot of libraries, you can find them at "Left-hand Panel - Library", hotkey "SHIFT+F", at here you can search library from LCSC, system, user contributed etc.

| Title(PartNO)   | Footprint | Capacitance | Inductar |
|-----------------|-----------|-------------|----------|
| NTCG164BH103JT1 | R0603     |             |          |
| ERTJ0EV104GM    | R0402     |             |          |
| ERTJ1VV154J     | R0603     |             |          |
| ERTJ1VR223G     | R0603     |             |          |
| ERTJ0EP333H     | R0402     |             |          |
| ERTJ1VA220H     | R0603     |             |          |
| ERTJ1VG103HA    | R0603     |             |          |
| ERTJZER104H     | R0201     |             |          |
| ERTJZEP473G     | R0201     |             |          |
| ERTJ1VT202H     | R0603     |             |          |
| ERTJ0EA680H     | R0402     |             |          |

## Type

- Symbol: Schematic symbols
- Spice Symbol: Symbols for spice simulation
- Footprint: PCB footprints, PCB pattern.
- SCH Modules: Schematic modules, a part of the circuit design. It can not assign the PCB module, doesn't like the schematic Symbol can assign the footprint . when it be placed on the schematic, it will be separated.
- PCB Modules: As like as Schematic modules.
- 3D Model: It is bind with footprint via "3D Model Manager".

## Classes

- Work Space: It include your personal parts and your teams' parts.

- LCSC: EasyEDA online part store [LCSC.com](http://LCSC.com) parts(Offical Parts). It will add new libraries everyday
- LCSC Assembled: JLPCB Assembled parts. All JLPCB assembly parts will contain a SMT icon, that means this part can be JLPCB assemble.
- System: EasyEDA system parts, it comes from open source libraries, such as Kicad libraries, company public libraries, user contributions.
- Follow: If you follow a user at EasyEDA(You can follow a user at him/her user page), you can view and use his/her libraries.
- User Contributed: When you searching a part, maybe you can find it at this class. At EasyEDA, all libraries are public. the detail you can refer at: [Contribute](#)

We add an "JLPCB Assembled" Components option of the Parts, It's easy to choose which component can be assembled by JLPCB. Yes, JLPCB will provide the assembly service. the more information please refer at: [How to order a SMT order](#)

## Search Engine - EasyEDA

Simply type your part number or symbol's name to Search. before searching, you must choose the "Type" first.

and then click the "Table of contents" to open the categories list to choose your components.

From there you can scroll up and down to browse parts from each category.

- If you know the component's name

Suppose you want to find the **MAX232** (which converts signals from an [RS-232](#) serial port to signals suitable for use in [TTL](#) compatible digital logic circuits). Simply type `Max232` into the Search box and press Enter:

| Title(PartNO)                | Footprint                          | SMT Type | Manufacturer           |
|------------------------------|------------------------------------|----------|------------------------|
| <a href="#">MAX232AEPE</a>   | DIP-16_L20.0-W6.4-P2.54-LS7.6-BL   |          | MAXIM                  |
| <a href="#">MAX232N</a>      | PDIP-16_L19.7-W6.6-P2.54-LS10.9-BL |          | Texas Instruments      |
| <a href="#">MAX232IDR</a>    | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | TI                     |
| <a href="#">MAX232ACSE+</a>  | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | Maxim Integrated       |
| <a href="#">MAX232ESE+T</a>  | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | MAXIM                  |
| <a href="#">MAX232DWR</a>    | SOIC-16_L10.3-W7.5-P1.27-LS10.3-BL | Extend   | TI(Tex as Instruments) |
| <a href="#">MAX232ESE+</a>   | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | Maxim Integrated       |
| <a href="#">MAX232DR</a>     | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | Texas Instruments      |
| <a href="#">MAX232ECSE+T</a> | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | Maxim Integrated       |
| <a href="#">MAX232ID</a>     | SOIC-16_L9.9-W3.9-P1.27-LS6.0-BL   | Extend   | Texas Instruments      |
| <a href="#">MAX232DWRC4</a>  | SOIC-16_L10.3-W7.5-P1.27-LS10.3-BL | Extend   | TI                     |

EasyEDA > Symbol > LCSC > Keyword:max232  
**\$0.0769** LCSC Part#: C524451 Stock: 3195 Minimum: 5 Distributor: LCSC

- If you don't know the component's name

For example, you want to find a resistor which value is 1kohm, footprint is 0603, at Libraries you can follow below steps:

- 1.Choose the library type
- 2.Typing the keyword such as `1k 0603`
- 3.Click the search button
- 4.Select the class you which is wanted of the result

- o 5.If you don't need the search you need to remove all the search keywords

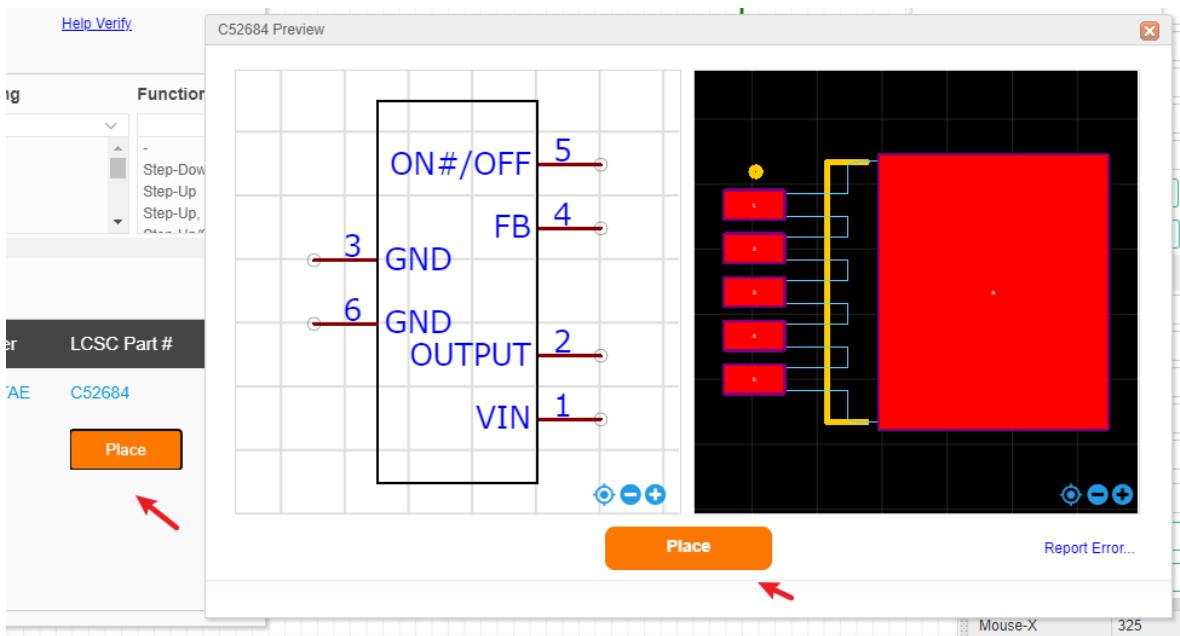
The screenshot shows the EasyEDA Library search interface. The search bar at the top contains '1k 0603'. Below it, there are tabs for 'Symbol', 'Footprint', 'Spice Symbol', 'SCH Module', 'PCB Module', and '3D Model'. The 'Footprint' tab is selected. A dropdown menu labeled 'Classes' shows 'Work Space(0)', 'LCSC(999+)', 'JLCPCB Assembled(907)', 'System(999+)', 'Follow(0)', and 'User Contributed(999+)'. The main area displays a table of components matching the search criteria. The columns are 'Title(PartNO)', 'Footprint', 'SMT Type', and 'Resistance'. The table includes rows for various RES-ARRAY-SMD parts with 1K resistance. On the right side, there are preview images of the component symbols and footprints.

## Search Engine - LCSC Electronics

When you want to find some parts by clearly parameter, you should try "Search Engine - LCSC Electronics", it all most same as LCSC.com.

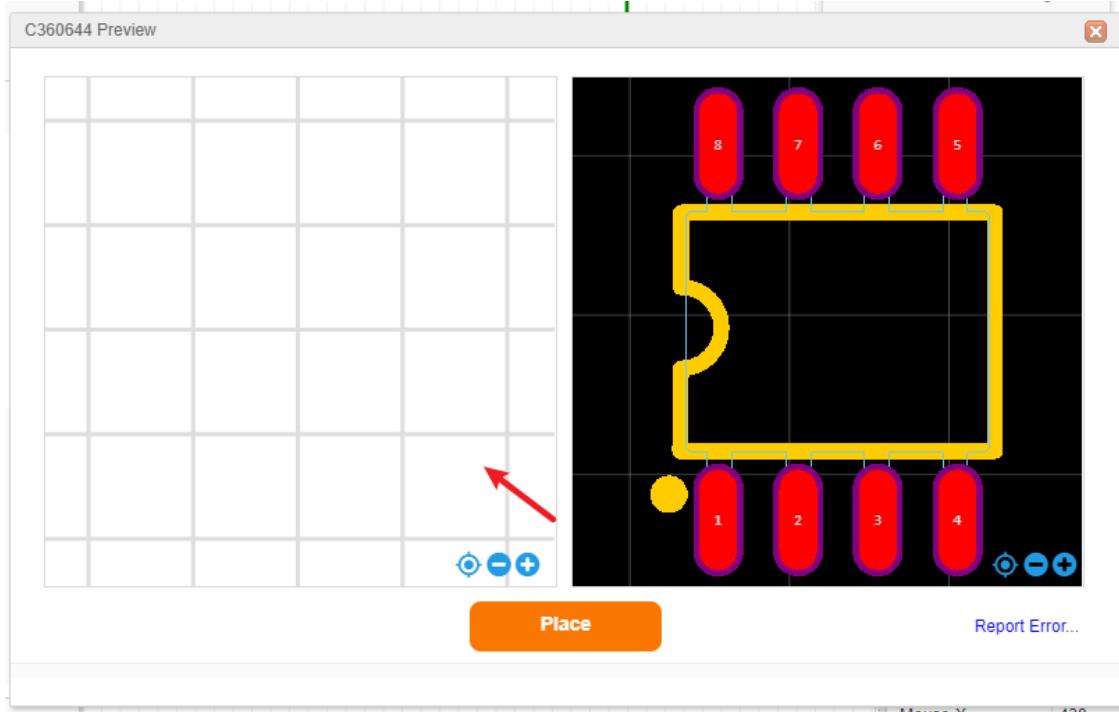
The screenshot shows the LCSC Electronics search interface. The search bar at the top contains '1k 0603'. Below it, there are tabs for 'EasyEDA' and 'LCSC Electronics'. The 'LCSC Electronics' tab is selected. A red arrow points to the 'LCSC Electronics > Power Management ICs > DC-DC Converters' breadcrumb trail. The left sidebar has a dropdown menu 'amp' with 'Capacitors' selected, and another dropdown 'Analog ICs' with 'Analog Comparators' selected. The main area shows filters for 'package' (SOT-23-5, SOT-23-6, TO-263-5, SOIC-8\_150mil), 'Current - Output' (3A, 2A, 1A), 'Frequency - Switching' (500kHz, 150kHz, 1MHz), and 'Function' (Step-Down, Step-Up, Step-Up, Step). Below the filters, a table lists products. One product is highlighted: LM2596R-ADJ by HTC Korea TAE, JIN Tech, with a price of \$0.708333. A red arrow points to the 'Place' button in the bottom right corner of the product card.

When you find out part, and you can place into the schematic:



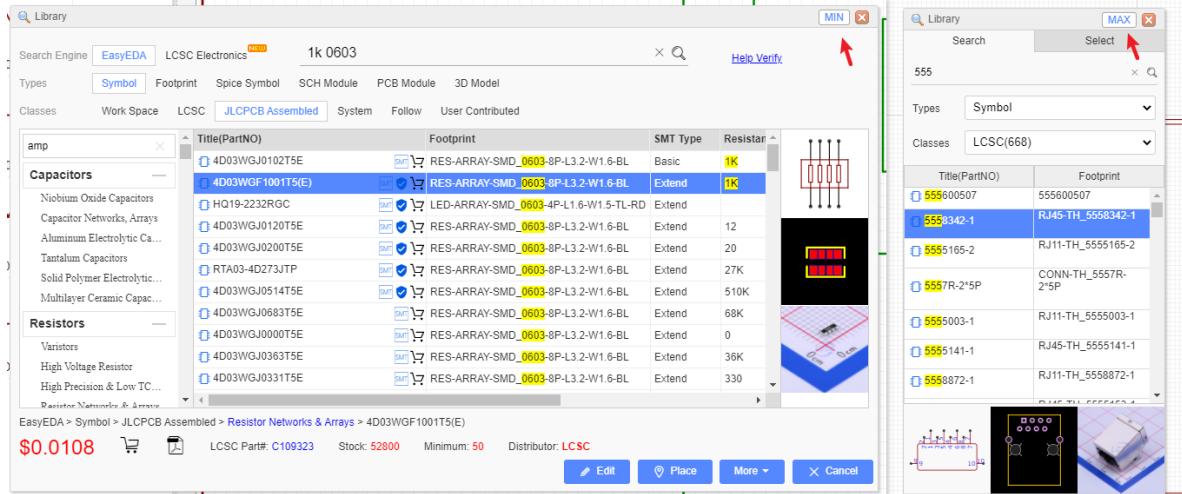
Notice:

- The subpart can not be preview at Preview dialog window, if you find out this, you need to change to "Search Engine - EasyEDA" to place this part.



## Max and Min mode

If you want to place without close the "Library" dialog, you can change dialog mode to Min mode, just click the Min button at the top-right corner.



## Operations

When you hover the mouse over the picture of the Schematic symbol or PCB footprint, you will find a toolbar with "Edit", "Place", "More" buttons.

### Place:

For parts you use infrequently, you don't need to Favorite them; just Place it into your canvas directly. Or you can double click the library to place.

Note:

- *EasyEDA supports multi-documents so please make sure that you are placing the part into the right (active) document. The active document is the one with the highlighted tab.*
- *You can't place a Schematic symbol into a PCB file, or a PCB Footprint into a schematic.*
- *EasyEDA will try the best to make sure the library is correct, but it still has incorrect parts, if you find any incorrect parts please let us known. suggested order a sample first before ordering a big order.*

### Edit:

If you want to create your own version of a symbol or footprint then you can open an existing part from the library to use as a template, edit it and then save it to your local **Work Space** library in **Library** of the Navigation Panel.

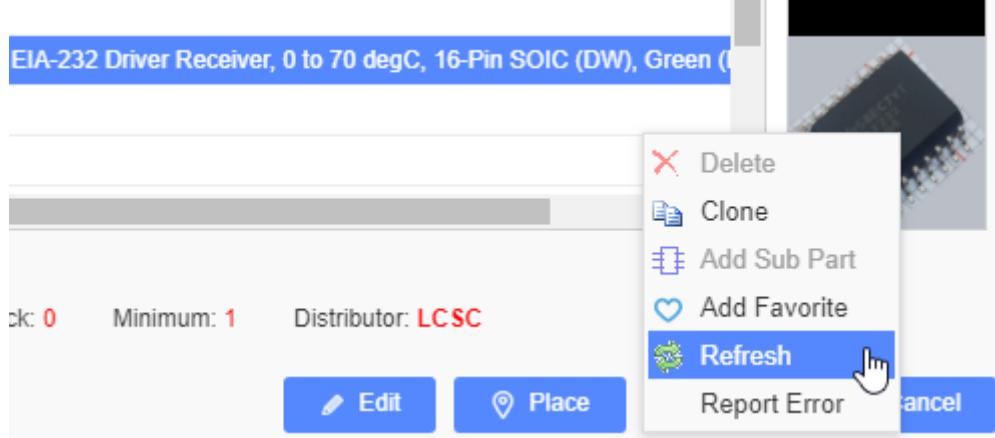
### More:

We can't promise that every component in the library is free of errors so please check all symbols and footprints carefully before you commit to a PCB order.

If you do find a mistake in a component, please use the **Report Error**, so that we can fix it.

Components with sub parts (multi-device footprints).

When you find a component with sub-parts, you can't Place or Edit it, but you can Favorite and Clone it as your own part, which you can then edit.



## Right-Click

When you right-click the part list, you can edit its tags, add favorite etc.

| Title(PartNO)  | Footprint                         |
|----------------|-----------------------------------|
| 4D03WGJ0102T5E | SMT RES-ARRAY-SMD                 |
| 4D03WGF        | Edit   RES-ARRAY-SMD              |
| HQ19-223       | Modify   LED-ARRAY-SMD            |
| 4D03WGJ        | Delete   RES-ARRAY-SMD            |
| 4D03WGJ        | Clone   RES-ARRAY-SMD             |
| RTA03-4D       | Add Sub Part   RES-ARRAY-SMD      |
| 4D03WGJ        | Add Favorite   RES-ARRAY-SMD      |
| 4D03WGJ        | Refresh   RES-ARRAY-SMD           |
| 4D03WGJ        | View Datasheet...   RES-ARRAY-SMD |
| 4D03WGJ        | Report Error...   RES-ARRAY-SMD   |

## Preview Image

Every library when you click, you can check its preview image, such as symbol, footprint, production picture. Click the the image you can open it quickly.

The screenshot shows the EasyEDA library search interface. The search term is '1k 0603'. The results table includes columns for Title(PartNO), Footprint, SMT Type, and Resistor. One row is selected, showing a resistor symbol and a footprint image. The footprint image is a 0603 package with two pads. The right side of the interface shows a detailed view of the footprint with dimensions and pad information.

| Title(PartNO)    | Footprint                             | SMT Type | Resistor |
|------------------|---------------------------------------|----------|----------|
| 4D03WGJ0102T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Basic    | 1K       |
| 4D03WGF1001T5(E) | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 1K       |
| HQ19-2232RGC     | LED-ARRAY-SMD_0603-4P-L1.6-W1.5-TL-RD | Extend   |          |
| 4D03WGJ0120T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 12       |
| 4D03WGJ0200T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 20       |
| RTA03-4D273JTP   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 27K      |
| 4D03WGJ0514T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 510K     |
| 4D03WGJ0683T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 68K      |
| 4D03WGJ0000T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 0        |
| 4D03WGJ0363T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 36K      |
| 4D03WGJ0331T5E   | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL    | Extend   | 330      |

# Placing Components

Find the component which you plan to place to your schematic at "Libraries", then move your mouse to the canvas and left click. If you want to add more, just left click again. To end the current sequence of placements, right click once or press **ESC**.

The screenshot shows the EasyEDA Library search interface. The search bar at the top has '1k 0603' entered. Below it, there are tabs for 'Symbol', 'Footprint', 'Spice Symbol', 'SCH Module', 'PCB Module', and '3D Model'. The 'Symbol' tab is selected. A search filter 'JLCPCB Assembled' is applied. On the left, there are two dropdown menus: 'Capacitors' and 'Resistors'. The 'Capacitors' menu lists options like Niobium Oxide Capacitors, Capacitor Networks, Arrays, Aluminum Electrolytic Ca..., Tantalum Capacitors, Solid Polymer Electrolytic..., and Multilayer Ceramic Capac...'. The 'Resistors' menu lists Varistors, High Voltage Resistor, High Precision & Low TC..., and Resistor Networks & Arrays. The main area displays a table of search results:

| Title(PartNO)         | Footprint                                 | SMT Type      | Resistance  |
|-----------------------|-------------------------------------------|---------------|-------------|
| 4D03WGJ0200T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 20          |
| RTA03-4D273JTP        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 27K         |
| <b>4D03WGJ0514T5E</b> | <b>RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL</b> | <b>Extend</b> | <b>510Ω</b> |
| 4D03WGJ0683T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 68K         |
| 4D03WGJ0000T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 0           |
| 4D03WGJ0363T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 36K         |
| 4D03WGJ0331T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 330         |
| 4D03WGJ0122T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 1.2K        |
| 4D03WGJ0101T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 100         |
| 4D03WGF499JT5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 49.9        |
| 4D03WGJ0105T5E        | RES-ARRAY-SMD_0603-8P-L3.2-W1.6-BL        | Extend        | 1M          |

On the right side of the table, there are two small diagrams showing the physical pin layout of the component. Below the table, the search path is shown as: EasyEDA > Symbol > JLCPCB Assembled > Resistor Networks & Arrays > 4D03WGJ0514T5E. At the bottom of the interface, there are buttons for '\$0.0108', a shopping cart icon, 'Edit', 'Place', 'More', and 'Cancel'.

Don't try to Drag and Drop a component to the canvas EasyEDA team thinks that Click-Click to place components will be easier to use than a Click-Drag mode.

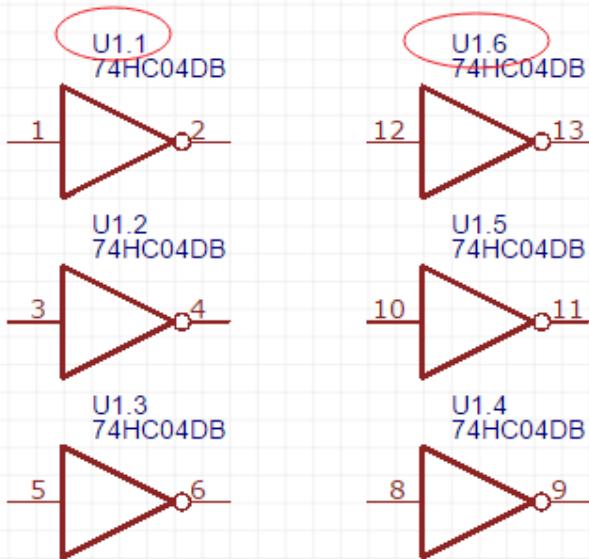
## Multi-part Components

The number of pins on some components can be quite large. That's why it's easier to divide such a component into several parts or functional blocks.

As a simple example, there are six gates in the 74HC04 Hex Inverter component. To avoid clutter in the schematic, GND and VCC pins of such components are usually served by a separate part of the component. This is really convenient as it doesn't interfere the working process with logical parts. The NetLabel names of VCC and GND Pin are usually hidden.

When placing the 74HC04 on a schematic, it will look like the screenshot below.

**Note:** The component Prefix will be in form of: U?.1, U?.2 etc.



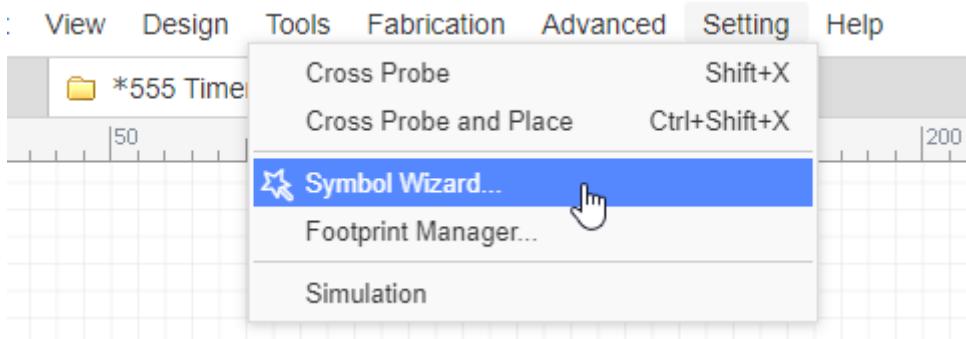
How to create multi-part(subpart) please refer [Create Symbol](#)

## Schematic Symbol Wizard

How many times have you hit a schematic capture roadblock because you couldn't find a component symbol?

Well, in EasyEDA that would be never because the Schematic **Symbol Wizard** provides a quick and easy way to create a general schematic library symbol.

Via: **Top Menu > Tools > Symbol Wizard** in a new schematic symbol or sheet document.

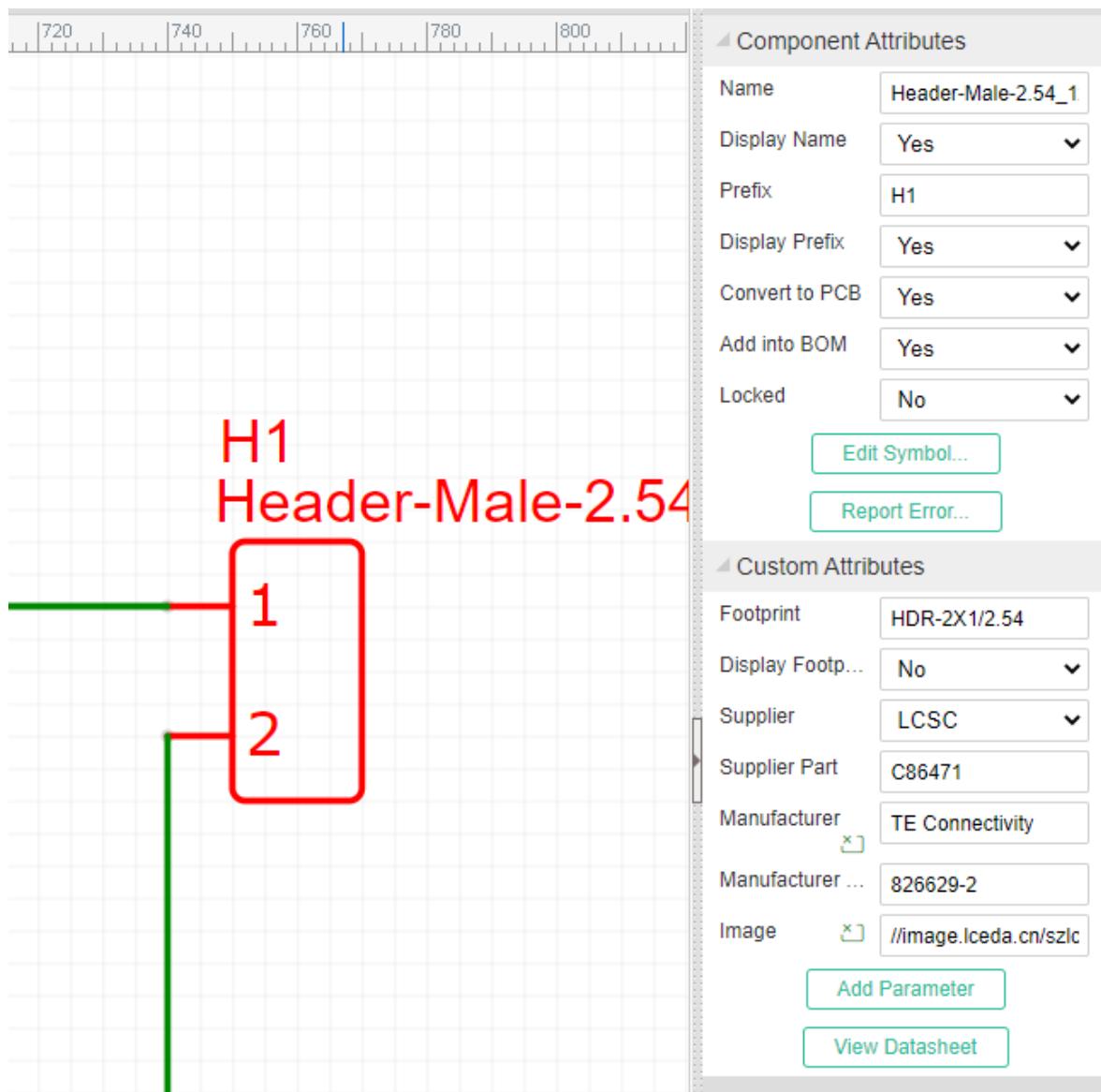


The professional function please refer at [Schematic Symbol Wizard](#)

## Component Attributes

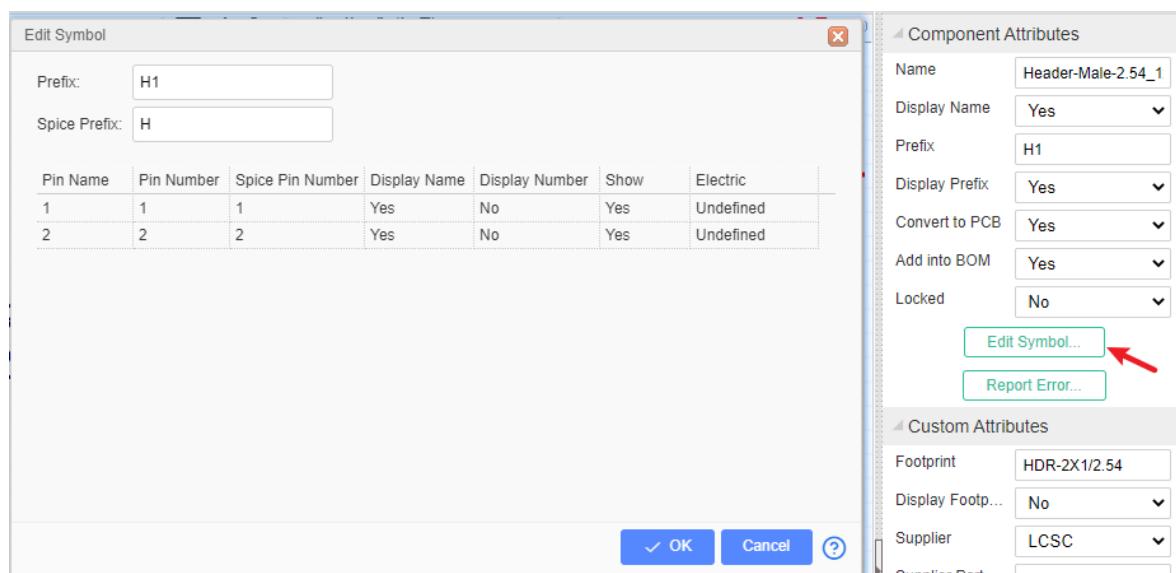
### Component Attributes

After selecting a component, you can find the component's attributes in the right hand Properties panel.



### 1. Component Attributes:

You can change the **Prefix** and **Name** here, And make them **visible** or **invisible**. If you want edit this component, you can click **Edit Symbol**.



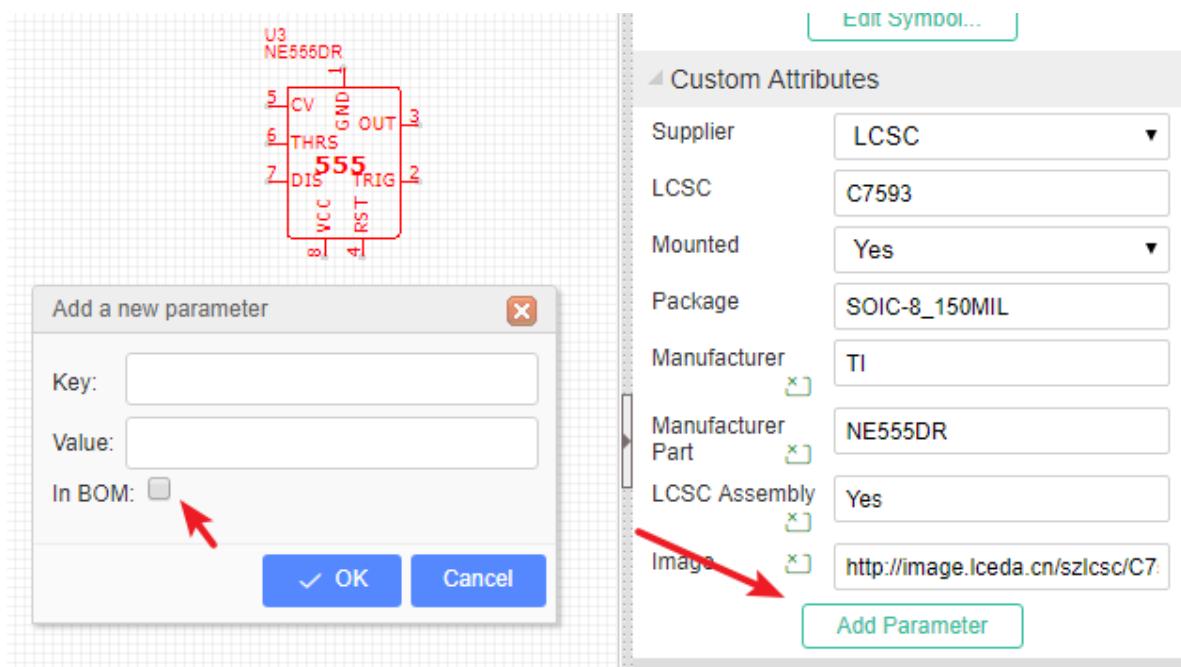
If the component's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

## 2. Custom Attributes:

You can change *component's supplier*, *change footprint*, and *add new parameter*.

## Define BOM Parameters

After selected a schematic symbol, you can add a parameter, and you can mark it as `In BOM`, when you export a BOM file, you can find this parameter in CSV file.

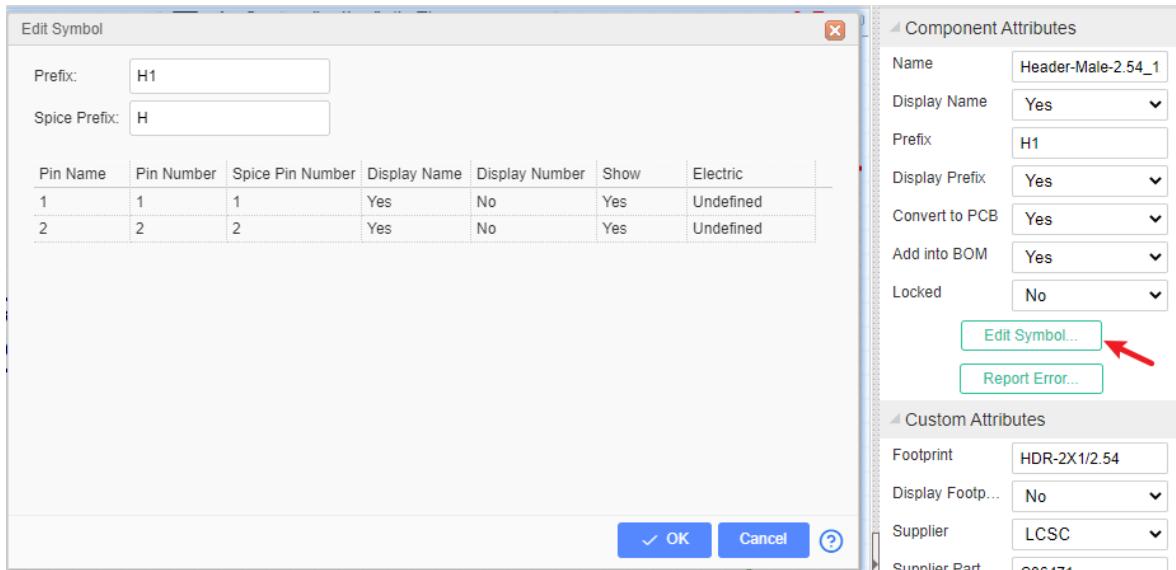


## Modify Symbol Pinmap Information

When you select a component, for opening the Modify symbol information dialog, you can do:

- Or press the `I` hotkey;
- Or click the Edit Symbol on the Parts Attributes on the left panel.
- Or click the Symbol and right-click, choose the "Edit Symbol" menu.

Using this dialog you can edit the pin names and numbers, for example, to suit a different footprint or device variant. You can also enter a Spice Prefix and swap the spice Pin order to make your symbol usable in simulation.



More detailed description of PCB and Spice Prefixes and pin numbers at next section.

## Prefixes and Pin Numbers

Device and subcircuit (or hierarchical block) symbols created for use in schematics that are intended to be run as spice simulations, in addition to having a PCB Prefix that is used for the reference designator in the schematic, also have a **Spice Prefix**. They also have two sets of pin numbers: PCB pins and Spice pins.

## PCB Prefix and Spice Prefix

For more information please refer at [Simulation: Schematic symbols: prefixes and pin numbers](#)

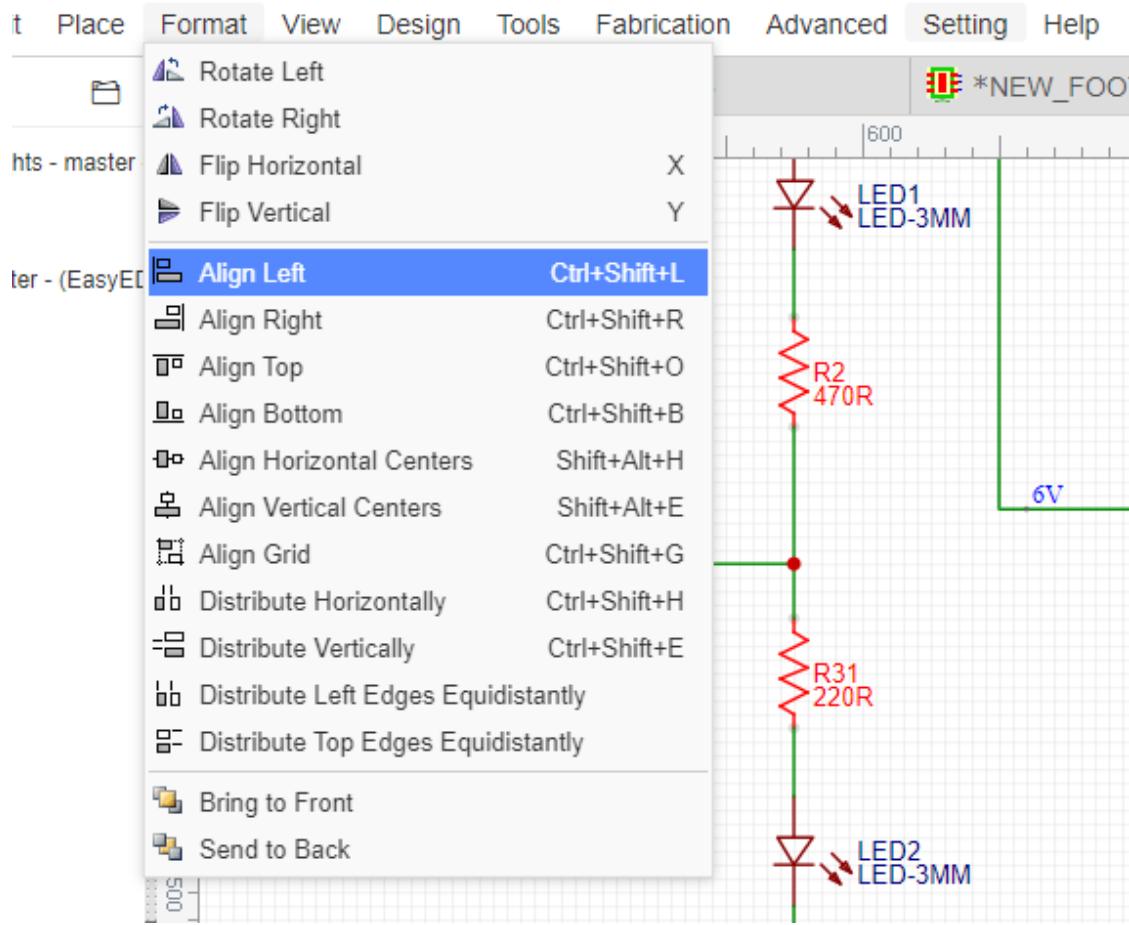
## Component Adjust

### Adjusting Components

About adjusting components you can:

1. Move components with your mouse
2. Move components with the arrow keys.
3. Find components with the Design Manager via the **CTRL+D** hotkey: select the component in the Design Manager to pan it to the centre of the canvas and then move it with your mouse.

4. Align the components:



### Rotating the Prefix and Value (Name) of components

The default Prefix and Value (or name) of EasyEDA components are horizontal. To change them to vertical, Left click the prefix or value and when it is highlighted in **red** color, then press the **rotation** hotkey **Space** and you're done.

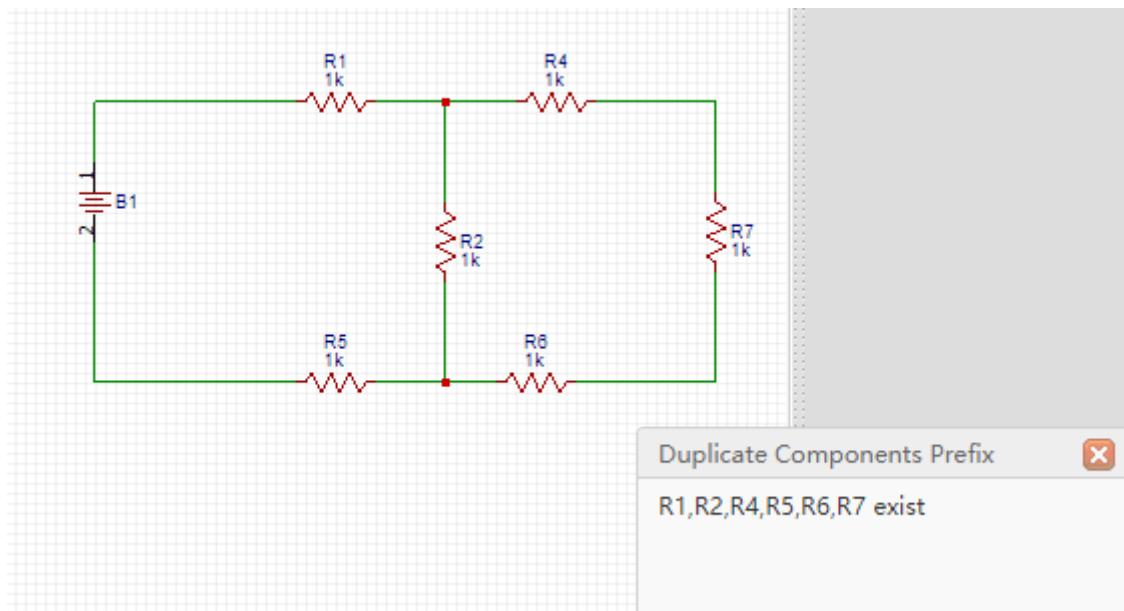
## Components Prefixes

### Prefix Start

In EasyEDA, at the first new schematic the prefix will start as U1/R1..etc, and EasyEDA support global unique prefix at multi-sheet now.

### Prefix Conflict Error

Sometimes, if you save a sheet to another project, when you convert a project to PCB, open the Design manager or run a simulation, you will get a Prefix Conflict error message.



In this schematic, you will find two components with the R4 reference designator, so you just need to change one to Rx where x is a unique number in that schematic.

It may be tempting to backup a schematic into the same project as the original, however, if an attempt is then made to do Convert Project to PCB, you will get the Prefix Conflict error for every component.



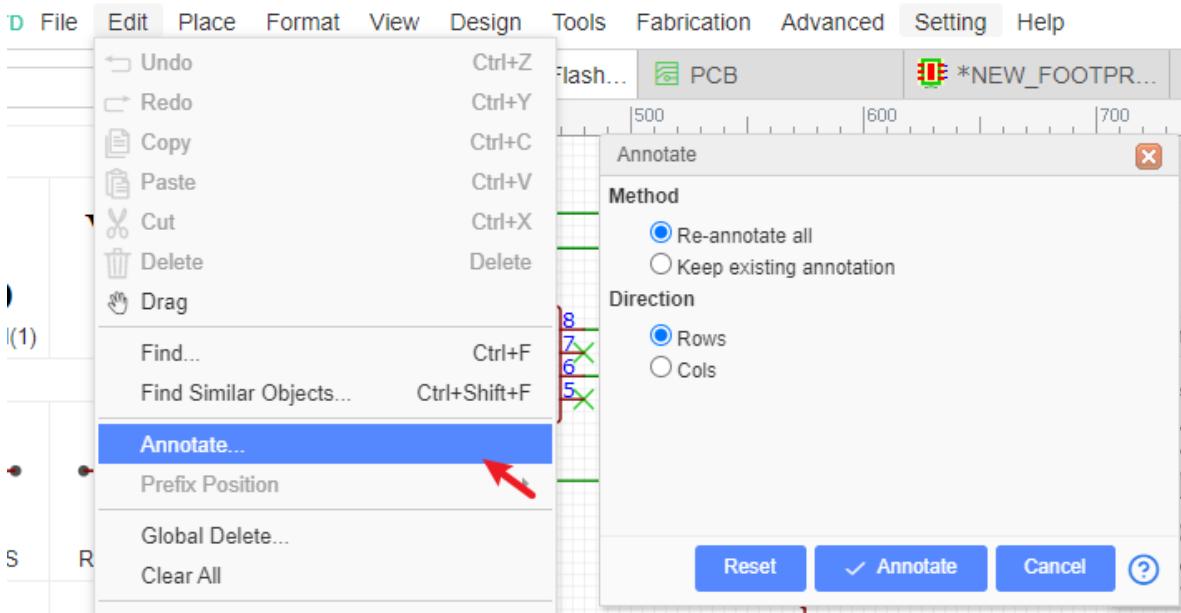
In the above image, you can find the two identical copies of the same schematic, which when you Convert Project to PCB, EasyEDA will try to merge into a single schematic, so every item will have 2 copies.

To fix this, you just have to create a backup project and remove or better still save backup copies of your schematics to that project.

## Annotate

After creating a schematic, it is quite likely that you have component Prefixes (reference designators) that are in no particular order on the canvas. You may also have duplicates. You can automatically renumber/reset all the components' prefix by using the **Annotate** function.

Via: Top Menu > Edit > Annotate



Various Annotate possibilities are available:

- **Re-annotate all:** resets all existing annotation and then annotates all components again from scratch;
- **Keep existing annotation:** annotates new components only (i.e. those whose reference designator finishes with ? like R? or U?).
- **Direction:** Rows annotates across the schematic in a raster pattern from top left to bottom right;  
Cols annotates down the schematic in a raster pattern from top left to bottom right.
- **Annotate:** applies the selected annotation actions.
- **Reset:** if you want to reset all the reference designators to end with '?', just click the Reset button. After that, R1 will be R?, U1 will be U? etc.

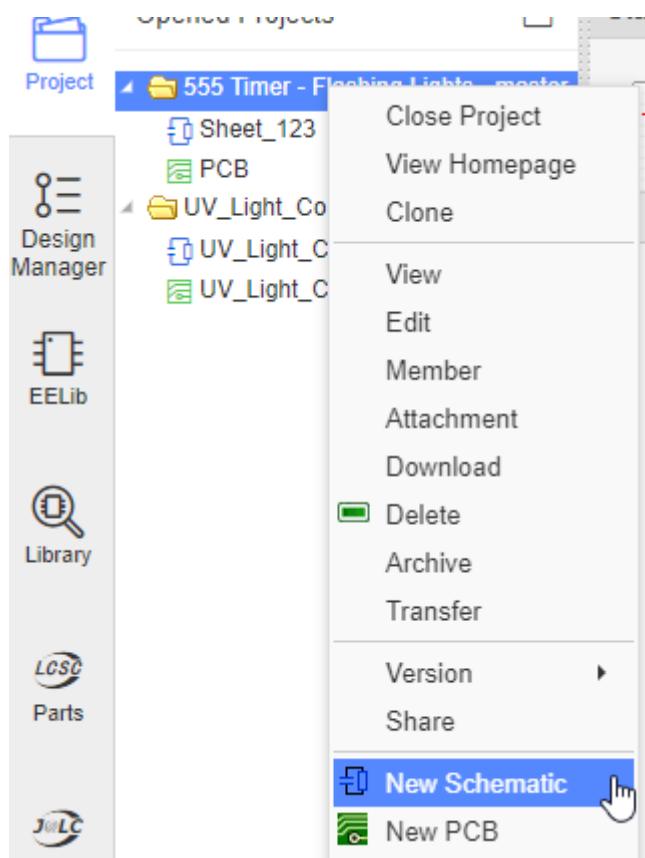
#### Note:

- *Reset does not reset annotation back to where it was before pressing the Annotate button.*
- *Annotation cannot be undone! if you do not accept the result: close all of the affected schematics without saving. If you do accept the result: make sure you save all of the affected schematics.*

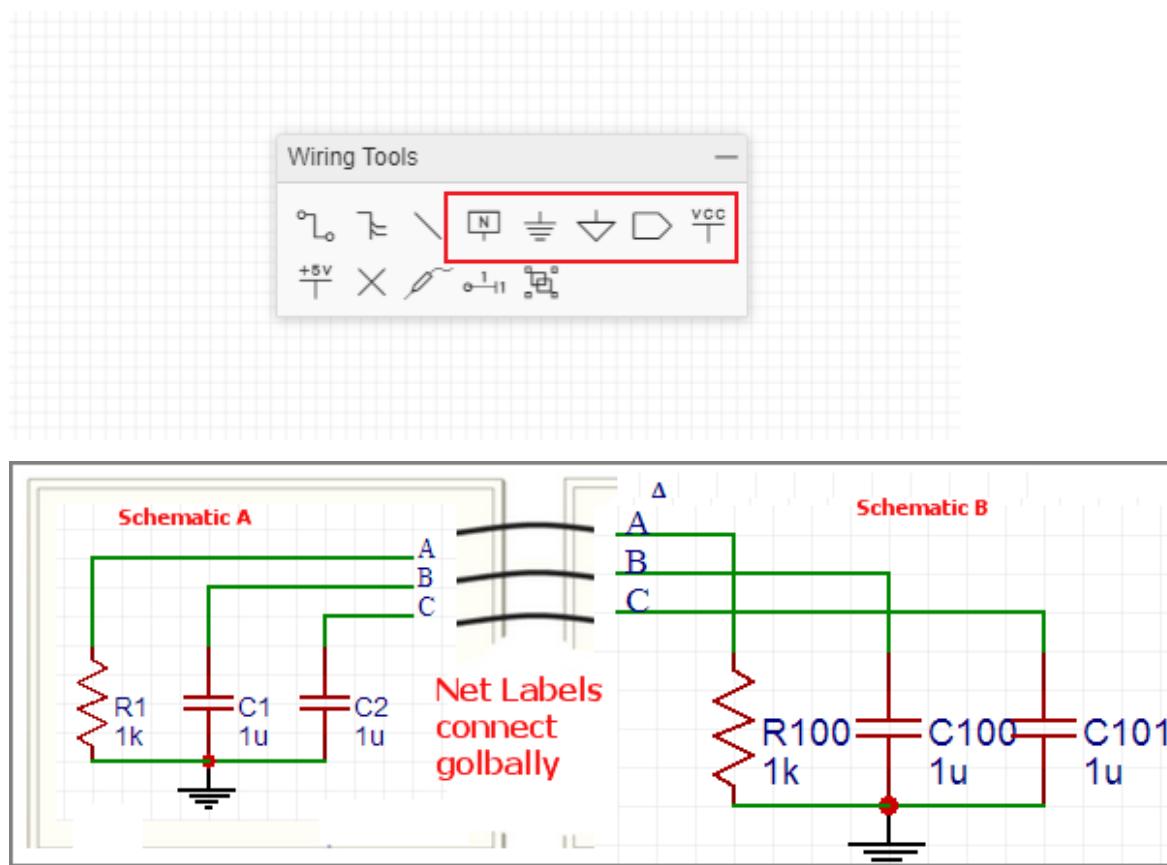
## Multi-Sheet

EasyEDA does not support true hierarchical designs but it does support **multi-sheet designs**.

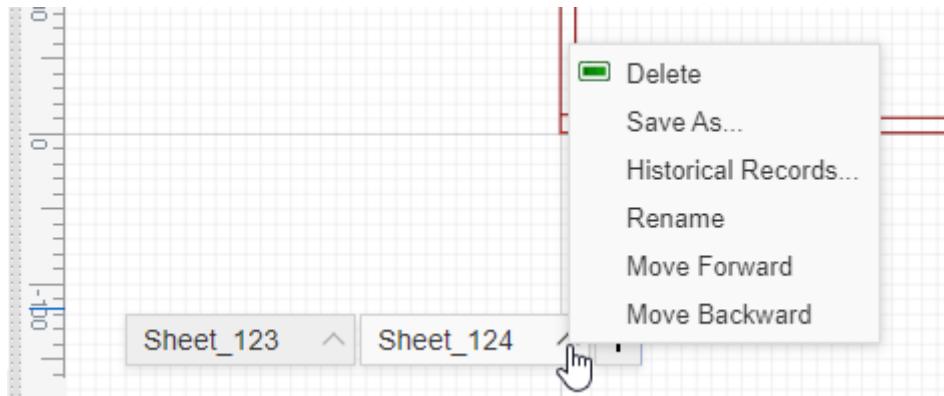
You can put several schematics in one project with connections between made by NetLabels/netPorts. All nets in EasyEDA are global so if you create a netlabel `DATA0` in sheet A and then create a netlabel `DATA0` in sheet B, when sheet A and sheet B are in the same project, they will be connected.



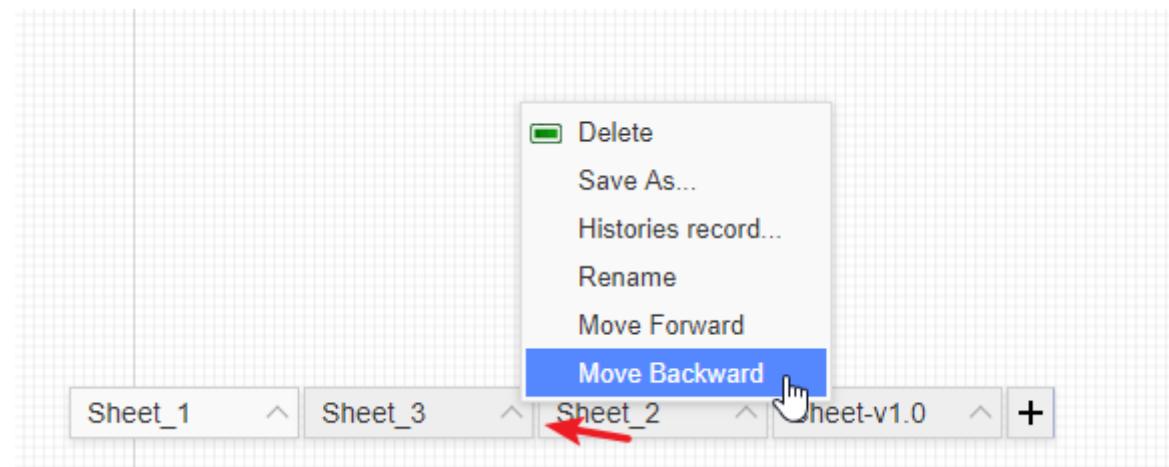
**Multi-sheet designs**(equivalent to a circuit spread over several pieces of paper), all schematics under the same project will be merged into one when be converted to PCB connecting in **Netlabel, Netflag, Netport**.



You can click the Sheet tabs on the left-down corner to switch the Sheets, and right-click the sheet tab you can "Save as", check "Histories record", "Move Forward/Backward", "Rename" and "Delete" the sheet.



If you want to arrangement the sheets order, you click the menu of the sheet icon: Move Forward/Move Backward.



**Note:**

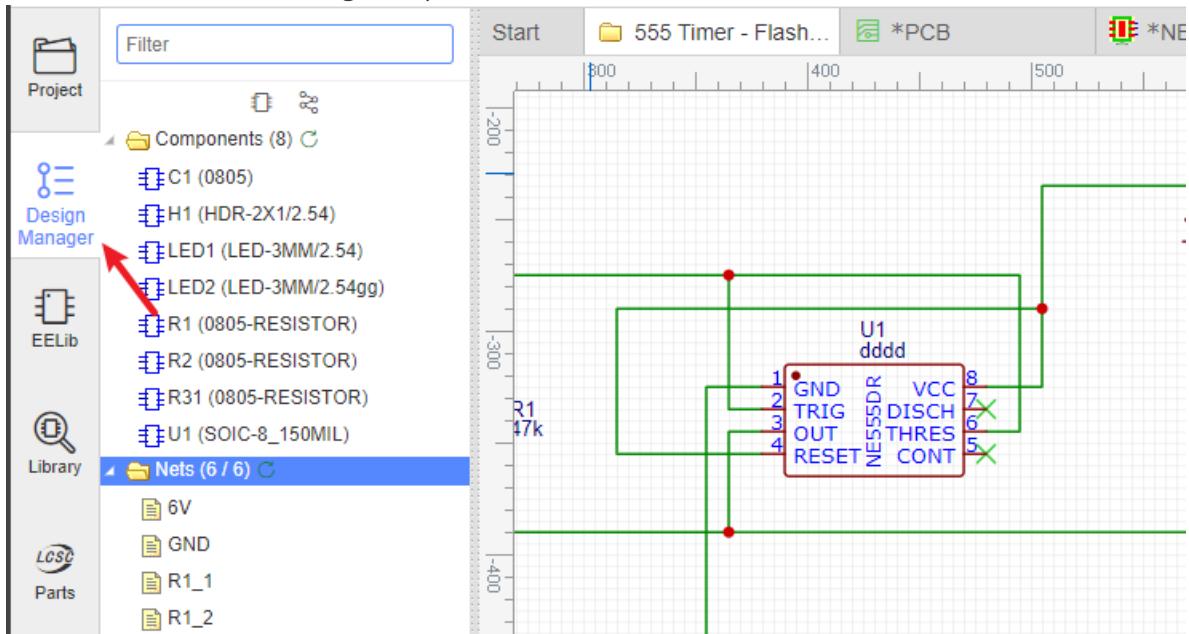
EasyEDA support global unique prefixes, when you place components in different sheet, the editor will auto annotate the prefix. If you save as a sheet to another project, please make all of the prefixes unique, if the Sheet A has a R1, and the Sheet B has a R1, then you will get a Prefix Conflict Error.

## Design Manager

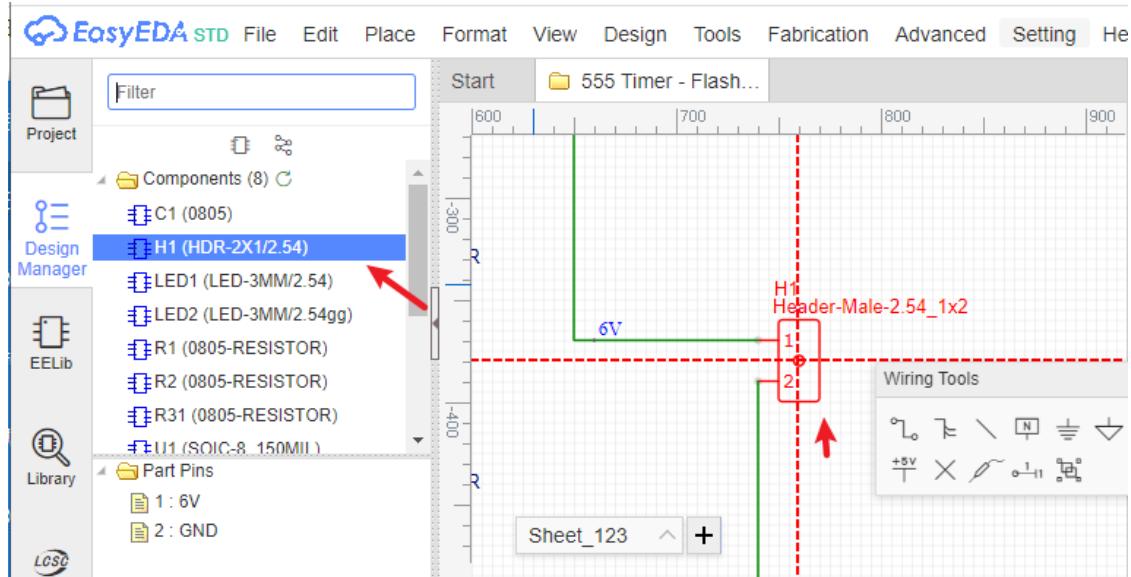
With large schematics it can be hard to find the components quickly. Sometimes, you may make a mistake such as wiring to a wrong component pin. So you need a tool to help you out. **Design Manager** is just the tool.

Just press the **CTRL+D** hotkey to open the Design Manager.

or click it via on the left navigation panel:

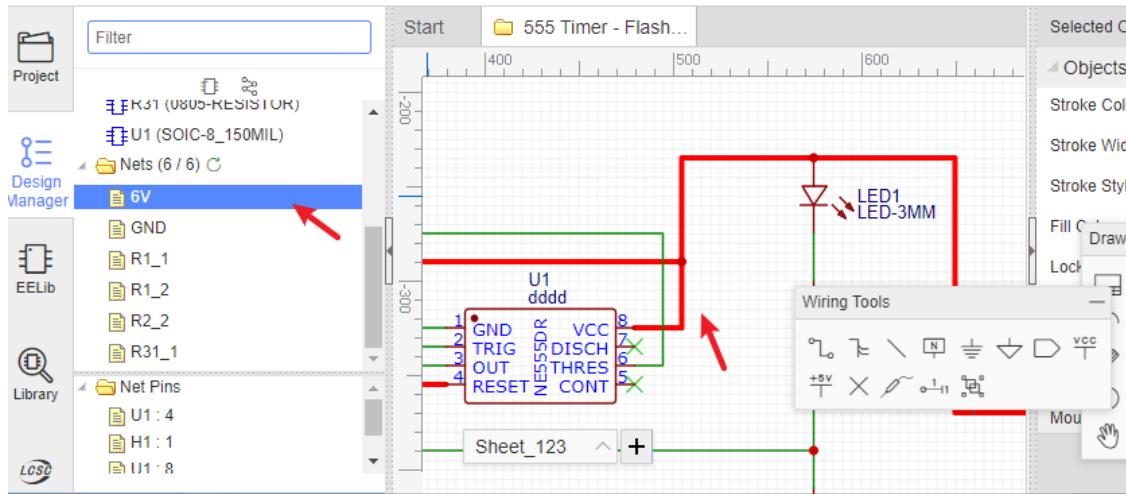


- Filter:** You can find your components or net name easily: for example, if you want to find all capacitances, you just need to type **C**;
- Components:** Lists all the components in this schematic. Clicking on a Component item highlights that component and pans it to the center of the window.

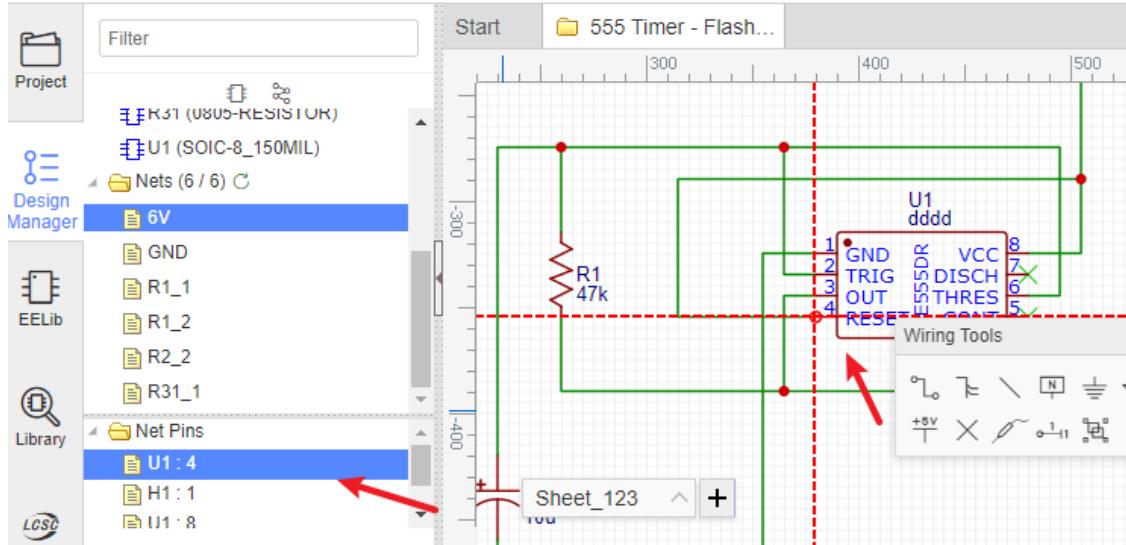


- Nets:** Lists all the nets in this schematic. A net must connect at least two Pins, or the net name will be marked as a red error. When click the net name, the canvas wire will highlight

and being large, when you click the empty space to unhighlight:



4. **Net Pins/Parts Pins:** Lists all the pins of the selected net name or components.



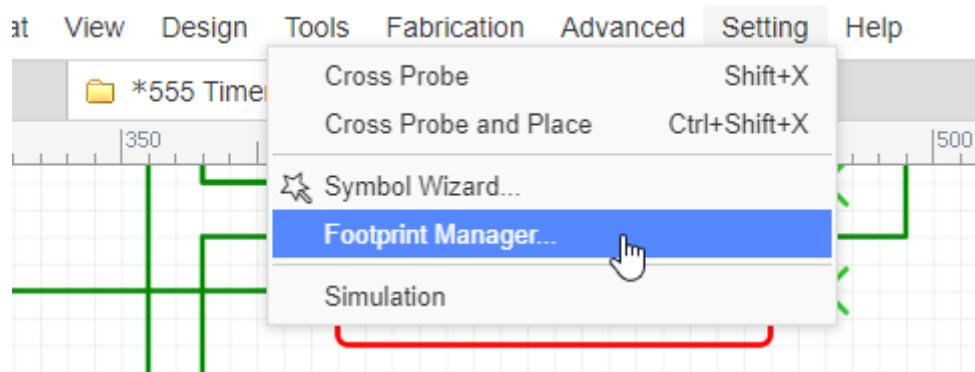
# Footprint Manager

## Introduction

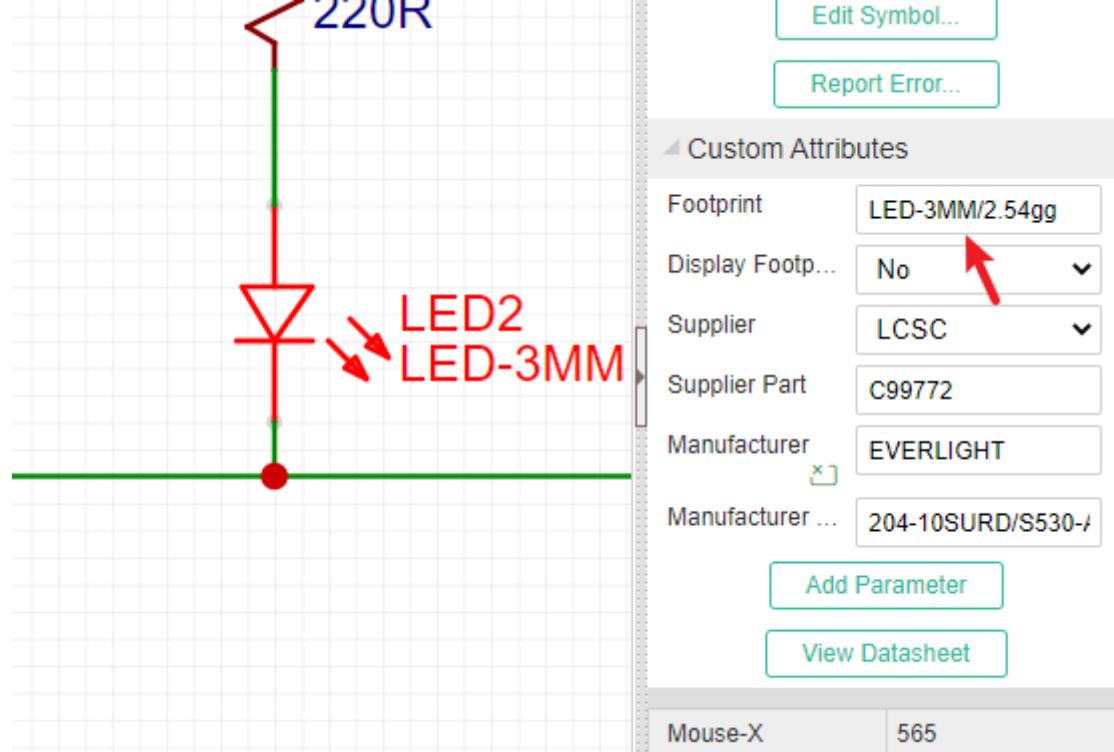
Want to batch modify components? Can't identify the corresponding relationship between component pins and footprint pins? Don't worry, EasyEDA can do this.

There are two ways to open the footprint manager:

- Click top menu, via: Top Menu - Tools - Footprint Manager



- Click the footprint input box of custom attributes when you've selected a component:



**1.** Footprint manager will check your parts footprint correct or not automatically when open it.

If the part without the footprint or this footprint doesn't exist in EasyEDA Libraries, or if the part's Pins doesn't correspond the footprint's Pads correctly, the footprint manager will show the red background alert.

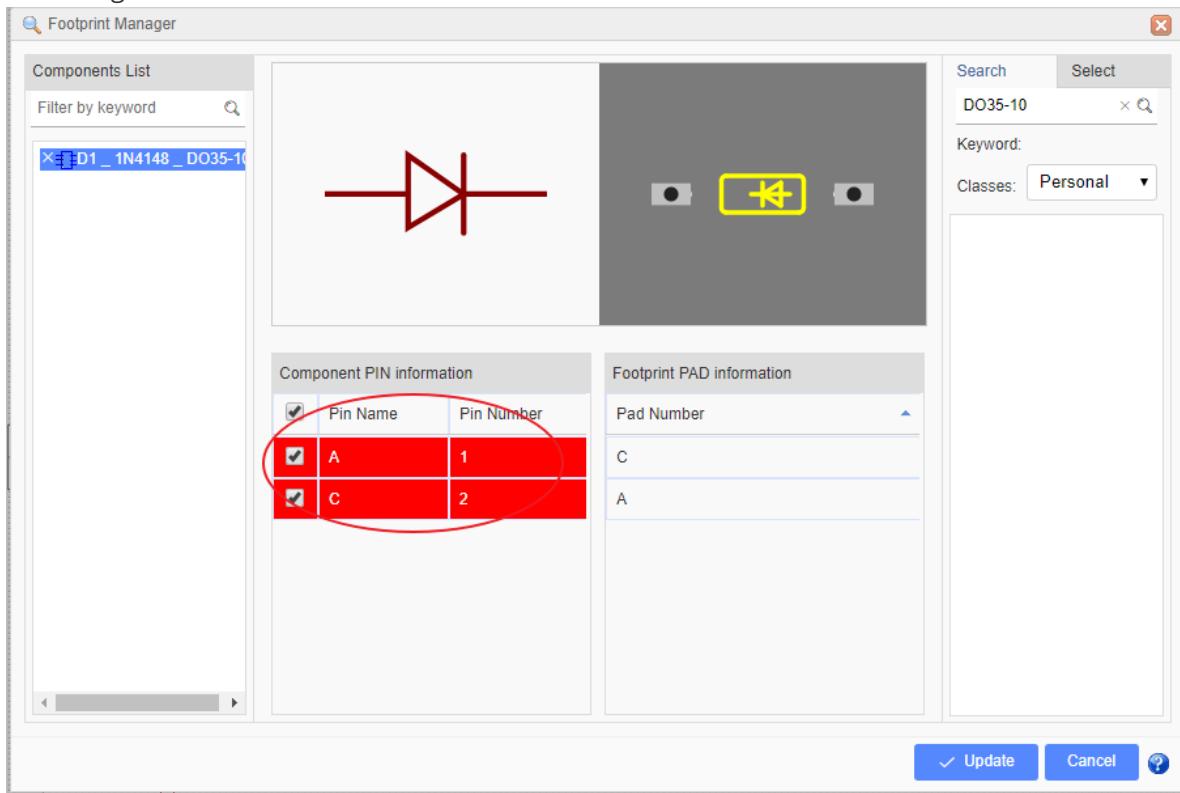
For example, If your part D1 has 2 pins,

- pin numbers are 1 and 2,
- pin names are A and C,

but you assigned a footprint has 2 pads,

- pad number are A and C,

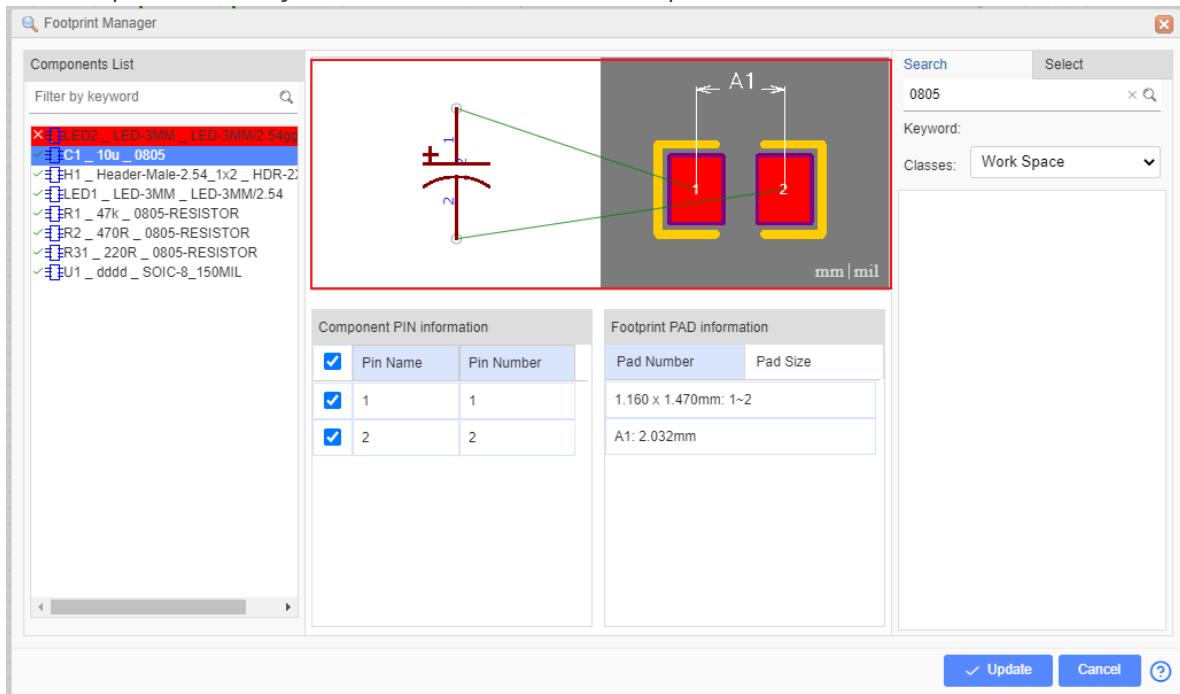
but the part's pin number doesn't match the pad number, so the footprint manager will alert red background:



In order to solve this:

- method 1: change part's [pin number](#) from 1 and 2 to A and C.
- method 2: change footprint's pad number as 1 and 2. That needs the footprint is created by you. And you can't change the Pad number in footprint manager, you need to find out the footprint at "Library > Footprints > Work Space", and then edit it.
- method 3: find an other footprint and update.

## 2. In the preview area, you can zoom in, zoom out and pan with mouse scroll button.



- Component PIN Information:** And you can modify component's pin map information in here.
- PCB PAD Information:**

- **Pad Number:** You can check the footprint's pad number, but you can't modify it. When you select the component on the left side, it shows component's footprint pad number, if you selected a footprint which is searched or selected from the classes, it will show the selected footprint's pad number.
- **Pad Size:** You can check the footprint's pads size and distance, it same as "Check Dimension" tool of footprint editor. Click the preview area unit text to change size unit.

## Update footprint

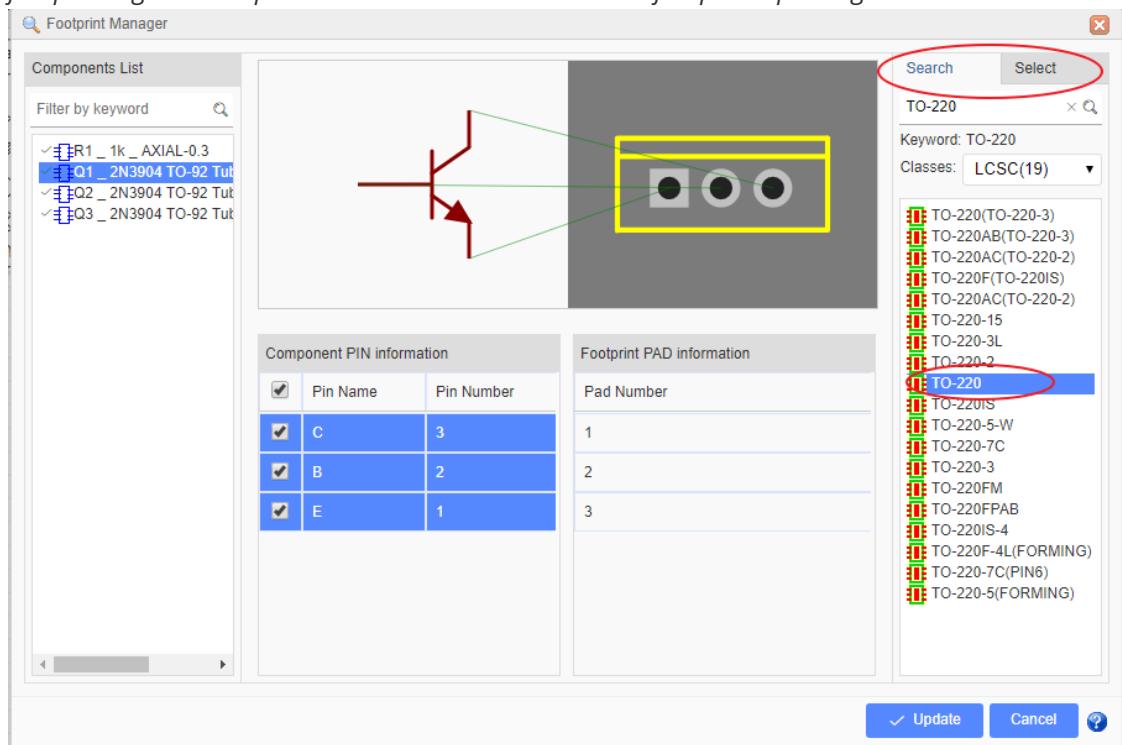
If you want to change the footprint, for example, select a component such as Q1, from **TO-92 TO TO-220**, you just need to click in the footprint input box. EasyEDA will popup the footprint manager dialog. You can follow the instructions.

- Type **TO-220** into the search box and search, Or change to Select tab,
- Select the classes you want and select **TO-220** footprint,
- Verify it in the preview box,
- then press the **Update** button.

After that you will find you have changed the footprint to **TO-220**.

### Note:

- To ensure that you use a footprint type that is already in the EasyEDA library, it is recommended that you use this technique to change component footprints rather than just typing a footprint name directly into the footprint text input box.because of the footprint manager will add the footprint's global unique ID into the schematic when the footprint updating.



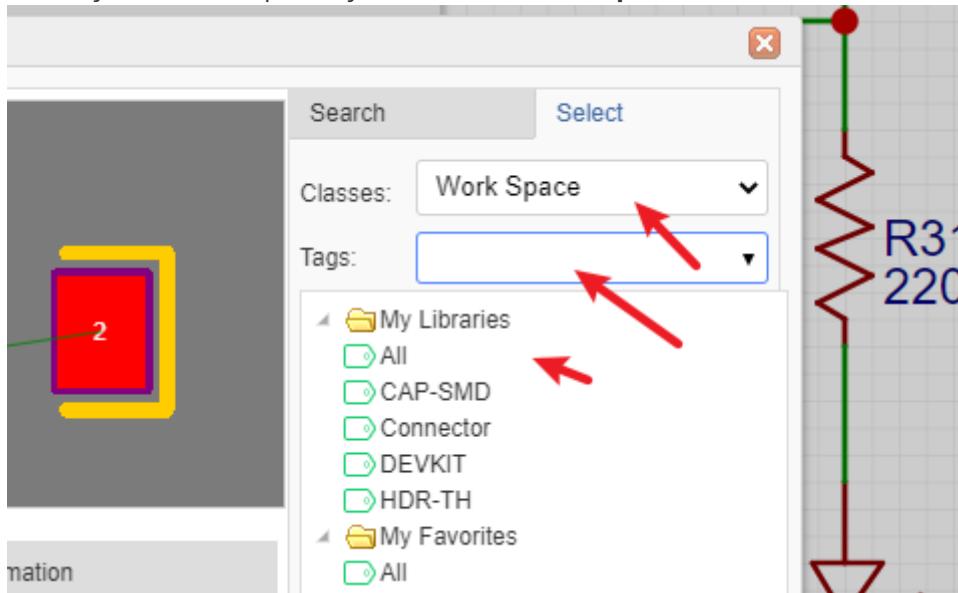
- When you select a subpart, the others subparts will be selected too, so they will update the footprint together.
- If the part's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

## Update in Batch

If you want to batch modify components' footprints,

- In the footprint manager dialog, you can press **CTRL + click** or **SHIFT + select** to select the components, and then select the footprint to update.
- In schematic canvas, you can frame select the components as you want, and then click the "footprint" attribute input box at the right-hand property panel.

To use your own footprints, you can select **Work Space** under the Select tab.



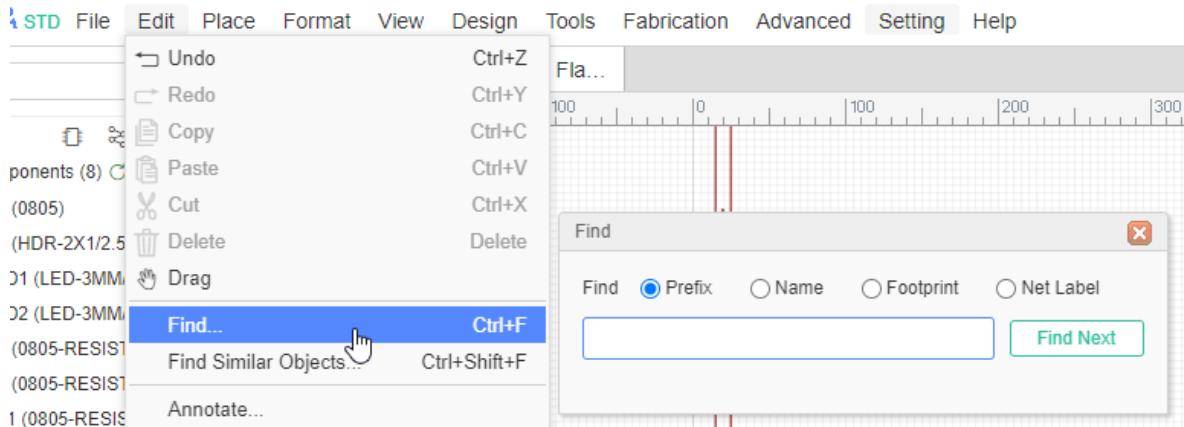
## Find Similar Objects

### Find Components in the Schematic

Finding individual **components** in a dense schematic can be very time consuming. EasyEDA has an easy way to find and jump to components:

**Top Menu> Edit > Find...**

(or **Ctrl+F**)



**Note:** You have to click **OK?** in this dialog or use the **Enter** key.

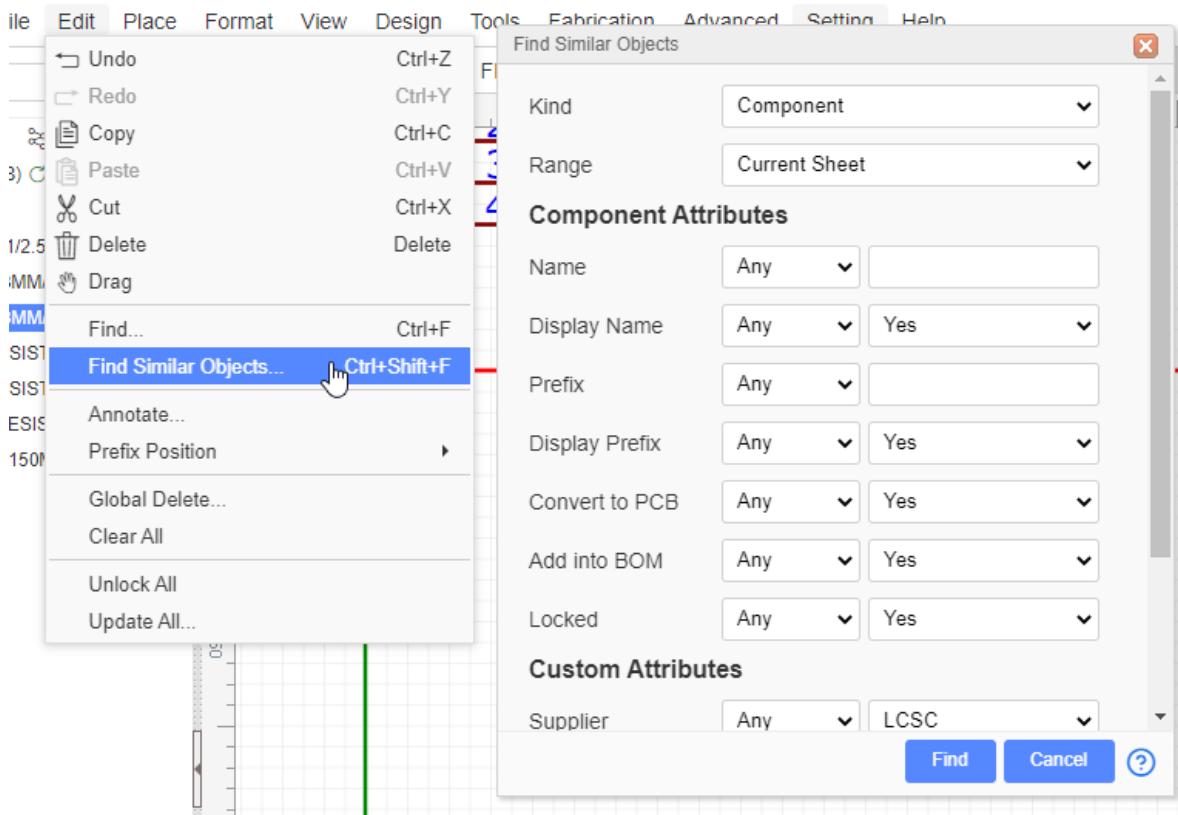
This feature will find, highlight and center in the window, parts by their Prefix (or reference designator). However, it cannot be used to find net names or other text in a schematic.

This is where the Design Manager comes in. For more information please refer Design Manager chapter.

## Find Similar Objects

EasyEDA provides a powerful find similar tool, you can find what you want very easily.

Via Top Menu > Edit > Find Similar Objects...

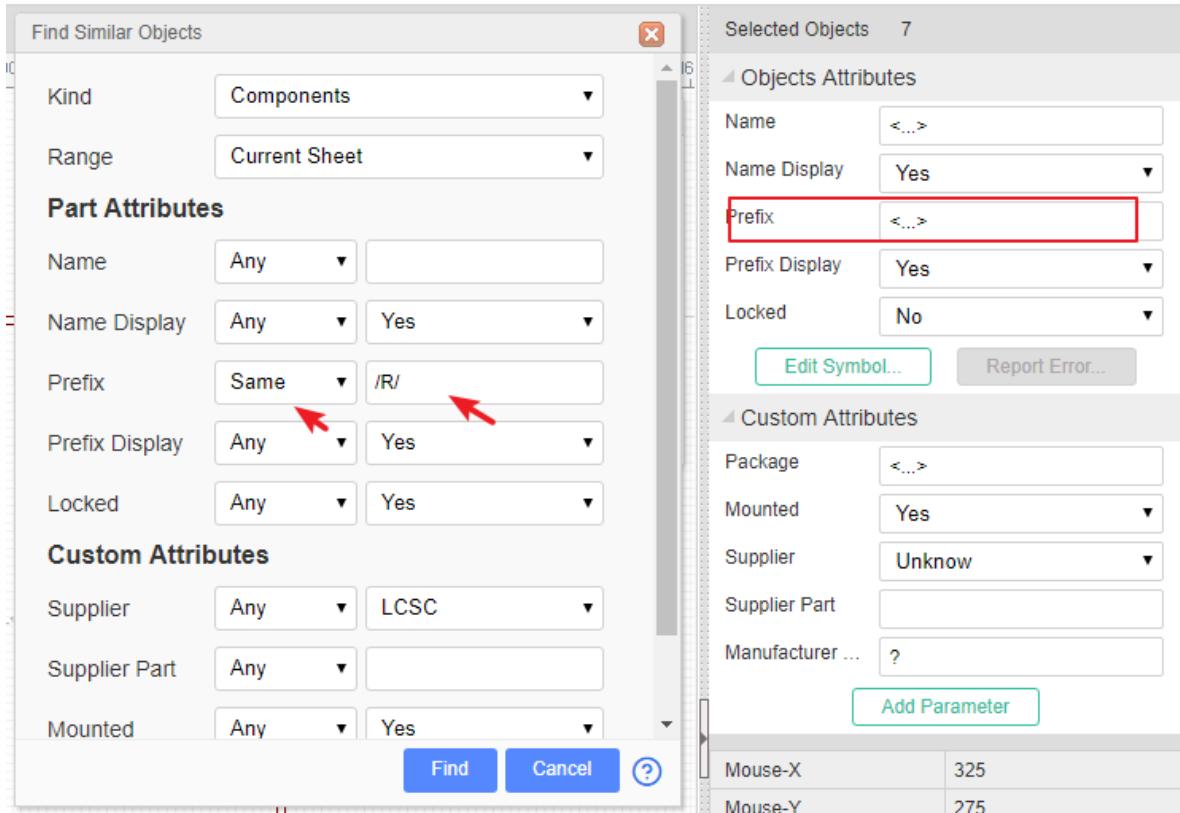


**Kind:** Select the object you want to find.

**Range:** This option is only for the schematic, you can find the object for current sheet or all sheets.

**Find Parameters:** Any: Find any objects; Same: Only find the object which attribute is the same as this attribute. Different: Find the object which attribute is different than this attribute.

The input box support the Js Regular Expression, you can type `/keyword/` to find what you want, such as find all prefix which are including "R":



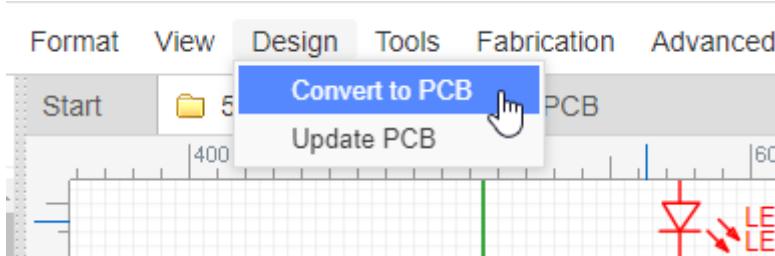
After click the "Find" button, all the suitable objects will be selected, and the right-hand panel will show all the attributes, the different attributes will show as the `<...>`, you can change the attributes directly, and they will apply to all selected objects.

The find similar objects only support to find a part of custom attributes. Such as footprint, supplier etc.

## Convert Schematics to PCB

### Convert to PCB

Most of the time, schematics are created with the aim of producing a PCB. So how do you convert your schematic to a PCB in EasyEDA? You just need to click the PCB icon on the toolbar with the title **Convert to PCB**.



#### Note:

- Before converting, you need to use the Design Manager and Footprint Manager to check all the components, nets(connection) and footprints to ensure no errors exist.

# Footprints Verification

After clicking the **Convert to PCB** button, if the project has errors the following dialog will open:

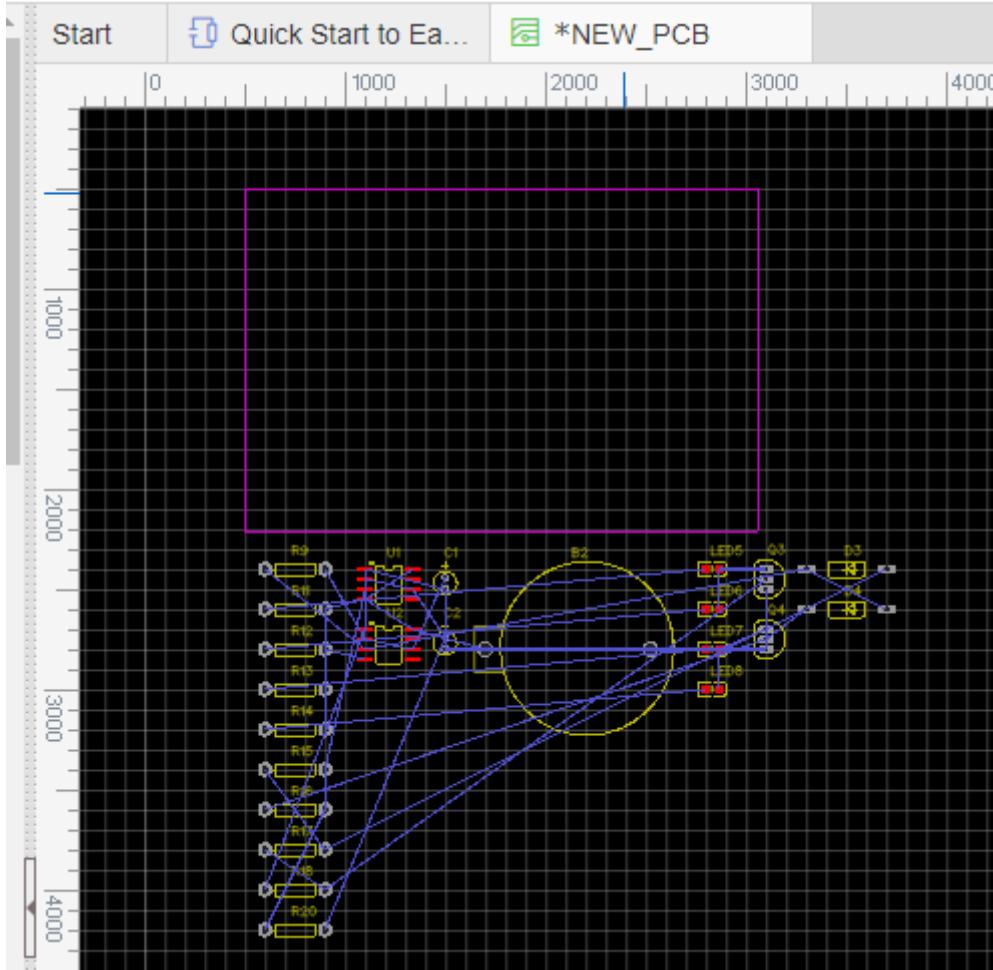
| Footprints Verification          |        |                        |                |                                                                                                                                 |
|----------------------------------|--------|------------------------|----------------|---------------------------------------------------------------------------------------------------------------------------------|
|                                  | Prefix | Name                   | Footprint      | Content                                                                                                                         |
| 1                                | LED2   | LED-3MM                | LED-3MM/2.54gg | The footprint name associated with the component and the footprint name on the server are inconsistent, please re-associate it. |
| <br><br><br><br><br><br><br><br> |        |                        |                |                                                                                                                                 |
| <a href="#">Check Footprints</a> |        | <a href="#">Cancel</a> |                | <a href="#">?</a>                                                                                                               |

The row in red indicates that EasyEDA can't find a PCB footprint matching the footprint that the schematic symbol is calling for.

This could be because you have made an error entering the footprint attribute in the symbol's Properties or maybe you haven't yet created a PCB footprint for the footprint that your symbol is calling for.

In this case the footprint should have been **AXIAL-0.3** but instead it is empty. To correct it you can click on the row and update the footprint **AXIAL-0.3** for it at the footprint manager.

After making any necessary corrections, click the **Convert to PCB** button and EasyEDA will automatically load all the PCB footprints into the PCB editor as shown in the image below.



This shows the footprints placed in arbitrary positions with the connections between them shown as blue Rat lines.

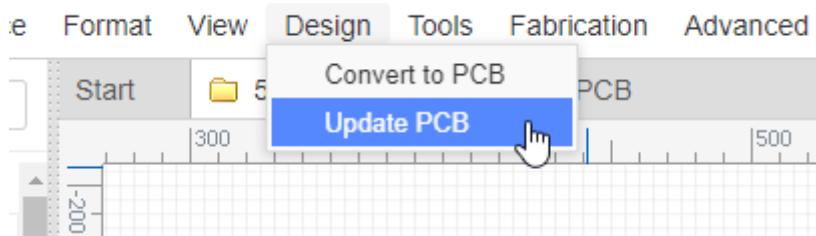
## Invalid footprint

The footprint's PAD number is different from the symbol's PIN number, e.g. the diode footprint's PAD numbers are A,C but the symbol's PIN numbers are 1,2. You just need to change one to fit the other. It is case sensitive!

For changing method please refer the **Schematic - Footprint Manager** section.

## Update PCB

Converting a schematic to PCB can be done using the `Convert to PCB...`, but if you do modifications to the schematic, by using the `Update PCB` button you can immediately be passed forward to update the selected PCB without having the PCB editor window already open or without creating a new PCB file.

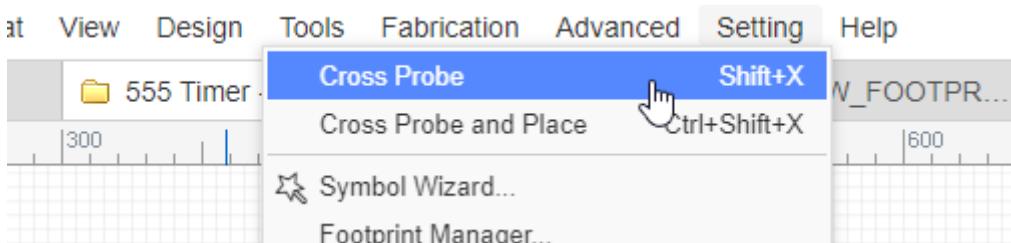


or you can use "Top Menu - Design - Import Changes" at PCB editor.

# Cross Probe

---

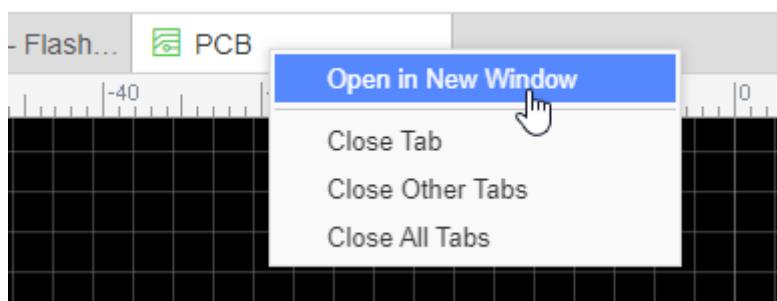
This tool is used to cross probe from chosen objects on the current schematic to its corresponding counterparts in the PCB, or from PCB Footprints to corresponding counterparts in the schematic.



Since v6.4.0, EasyEDA supports multiple windows design to cross probe.

How do it works?

1. Open schematic and PCB
2. Right-click the schematic or PCB tab, click "Open in New Window"



3. It will open this document in new window, then you can do the cross probe: Click the component, click the Design Manager list, the "Cross Probe and Place" works too.

## Note:

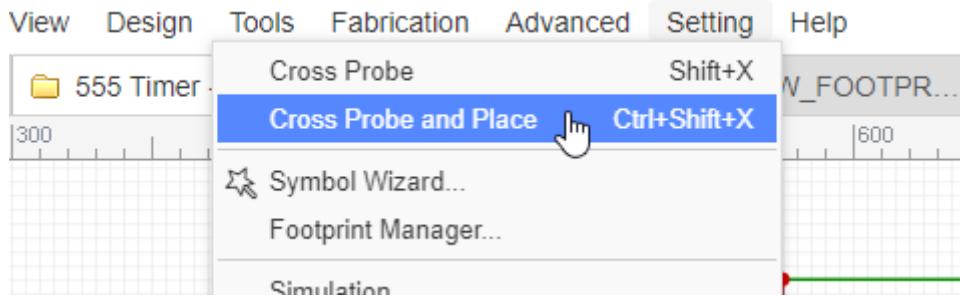
- You need to open PCB first before using cross probe in the schematic. And don't forget to use the hotkey `SHIFT+X`.
- After converting the schematic to PCB, for using this function please save the PCB first.
- If your project has many PCBs, when you use the cross probe please open the PCB what you need manually.

# Cross Probe And Place

---

If your schematic have a lot of components, it will be difficult to layout the PCB , so EasyEDA provides a powerful function "Cross Probe And Place".

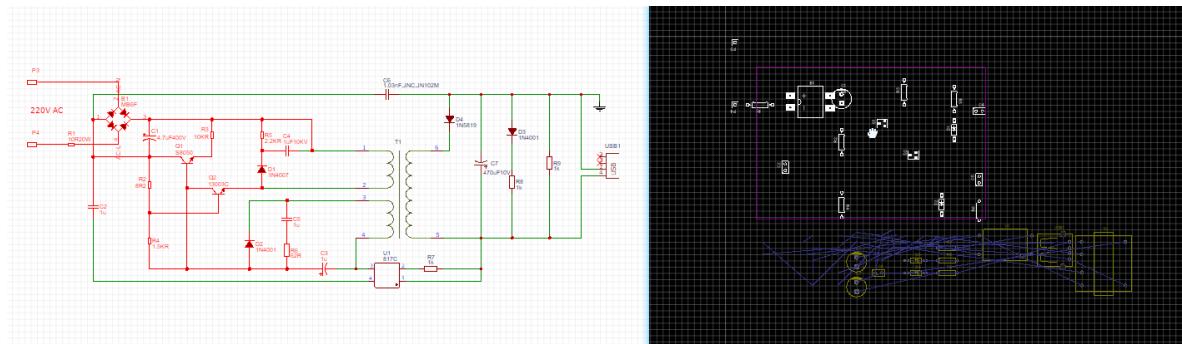
**Top Menu> Tools > Cross Probe And Place**



Cross Probe And Place will make the footprints' location match the schematic's parts' location as much as it possibly can.

#### How to use:

- Convert the schematic to PCB first, and save at current project.
- Frame select the components area by mouse in the schematic, and then click the "Cross Probe And Place", hotkey "CTRL + SHIFT + X".
- The editor will switch to the PCB, and choose the footprints as you selected for waiting for placing.
- Right click to place, and the mouse will keep the drag status, its easy for adjusting the footprints' location.



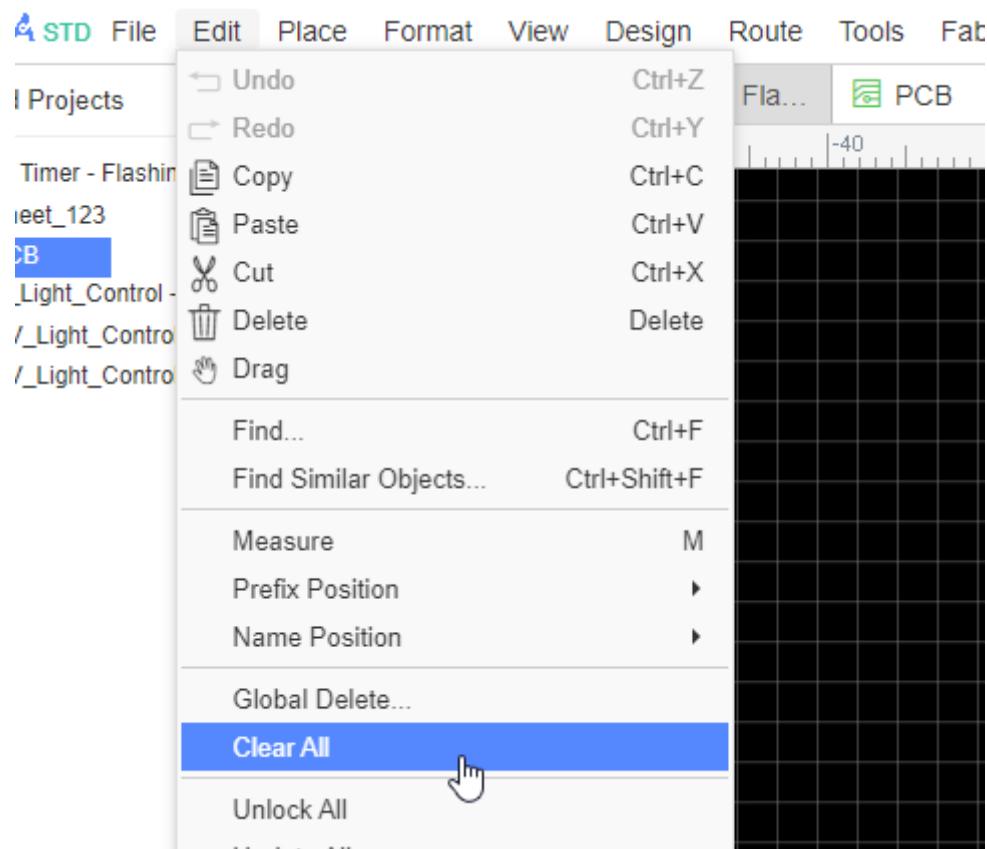
#### Notice:

- You need to open PCB first before using this function in the schematic

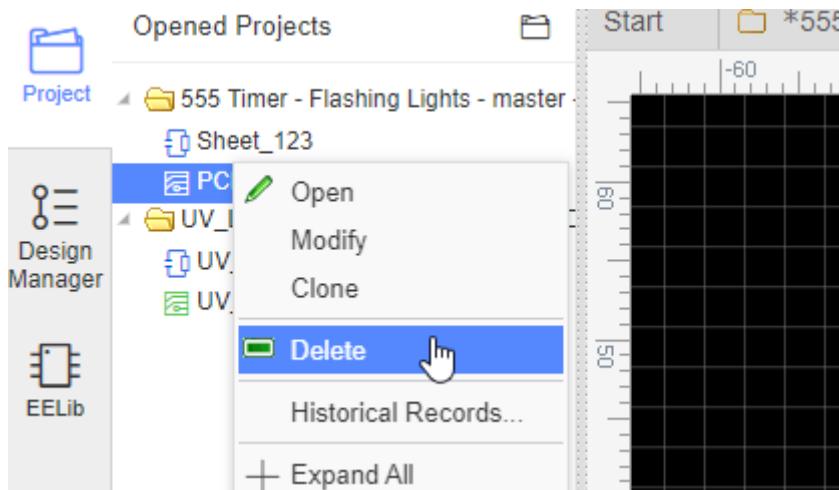
## Global Delete

If you feel your schematic or PCB is mess up, need delete objects in batch, you can:

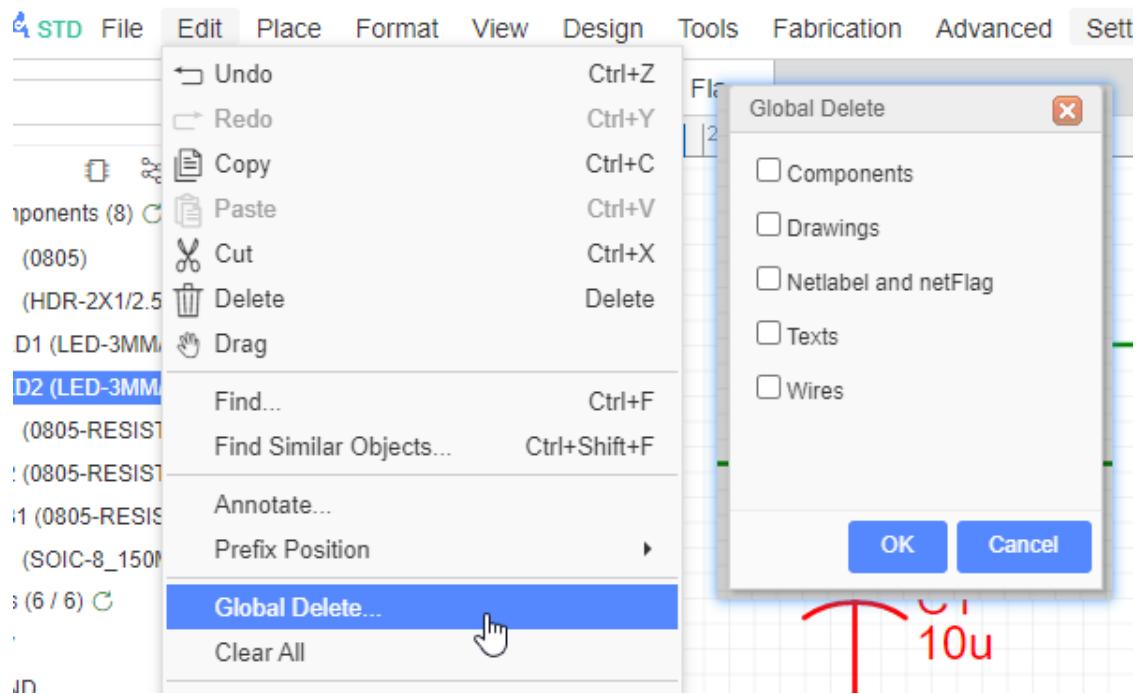
- Top Menu > Edit > Clear All, or CTRL + A select all and then press Delete key.



- Delete the document and create a new one.



- Using Top Menu > Edit > Global Delete, just delete what you want.



## Schematic Modules

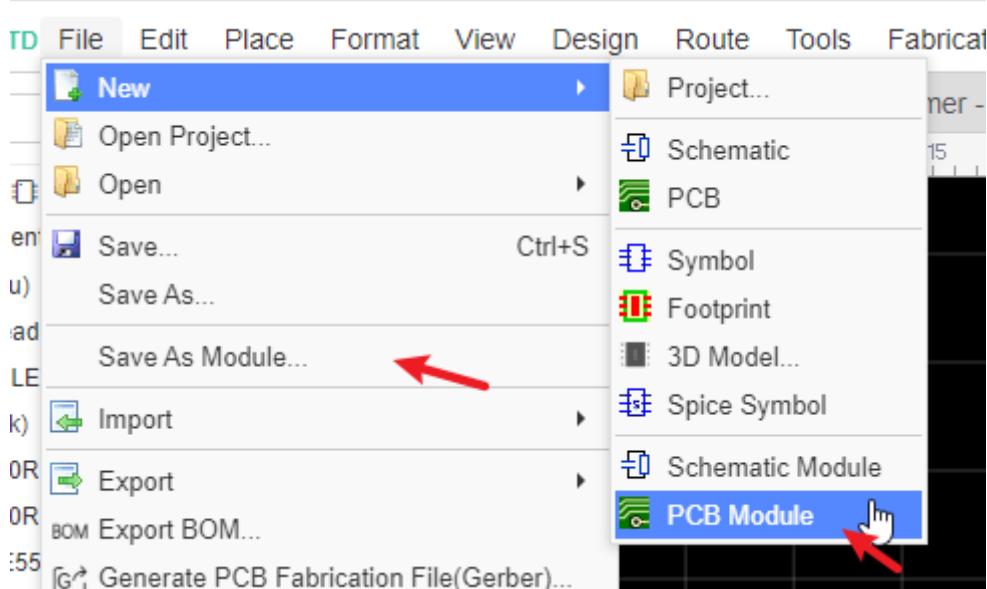
Copying codes is an easy job for coders, now copying and reusing a schematic or PCB is easy. Take a power supply unit for example, you can save this unit as a schematic module.

Via **File > Save as Module**:

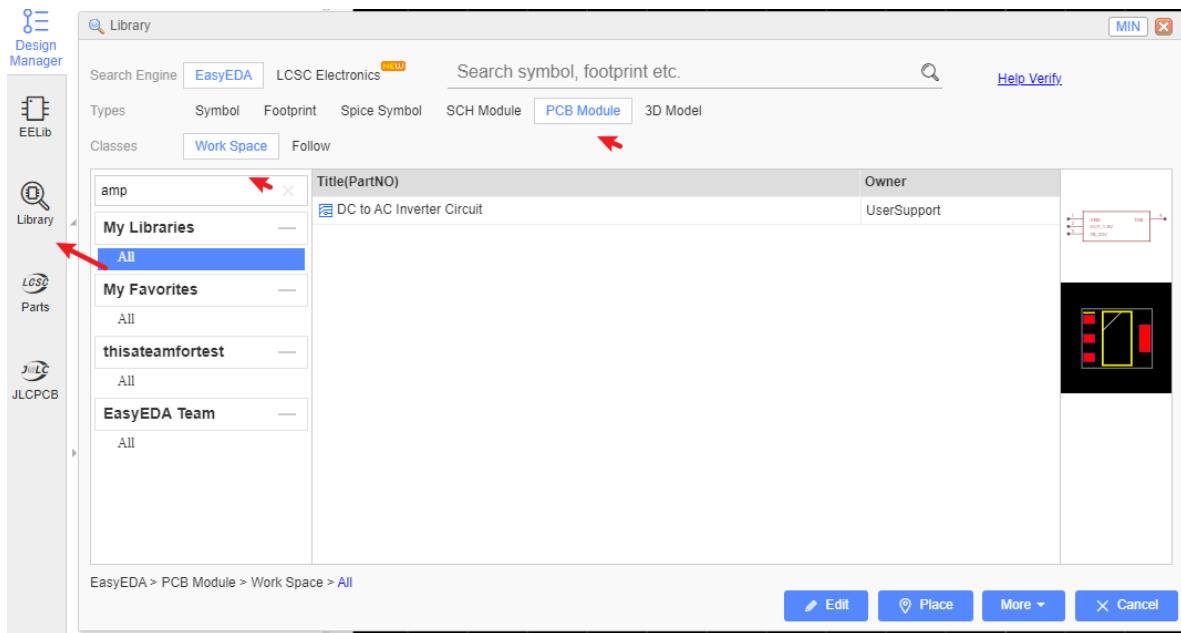
EasyEDA support create the PCB modules, it seems schematic module.

## How to Create

Via: **Save as Module** and **File > New > Schematic/PCB Module**.



## PCB module save at Library > Schematic/PCB module > Work Space > My Libraries

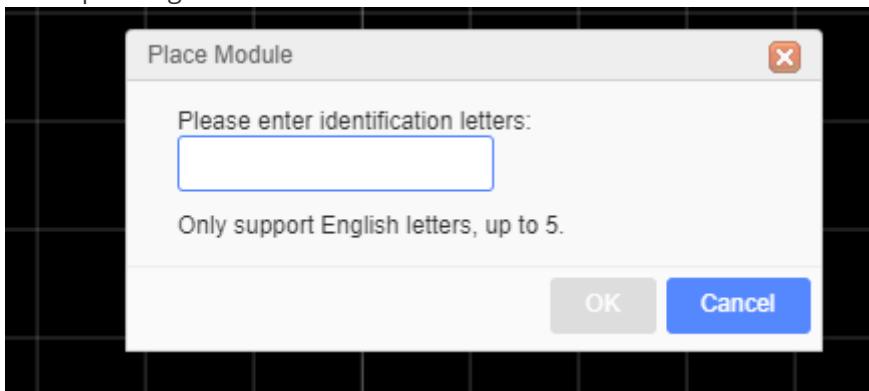


## How to use

Since v6.4.3, after placing schematic modules and PCB modules, after Import Changes, supports to keep the layout location.

How to use:

1. Draw schematic modules and PCB modules, and ensure that their component prefix are one to one, and the footprint is also corresponding. The module's component prefix can not have question marks and duplicate prefix, such as U? or two R1.
2. Open schematic and PCB at a same project.
3. Open "Library", select the module.
4. Click the "Place" button to place the previous saved schematic module and PCB module.
5. It will pop up a window to enter English letter. The letter of schematic module should keep corresponding with PCB modules.



For example: A component at schematic module is U2, enter letter K, press OK to place into canvas, it will be KU2, then PCB module has KU2 too.

Click "OK" and enter the placement mode. After each placement, the pop-up will continue to enter the identification letter. Make sure that the identification letters entered each time are unique.

6. When finish the module place, the PCB component unique ID will same as Schematic component unique ID, then after Import Changes, the component's location will be keep. and you can update the track's net follow the schematic netlabel too.

That implement the multiple channel placing.

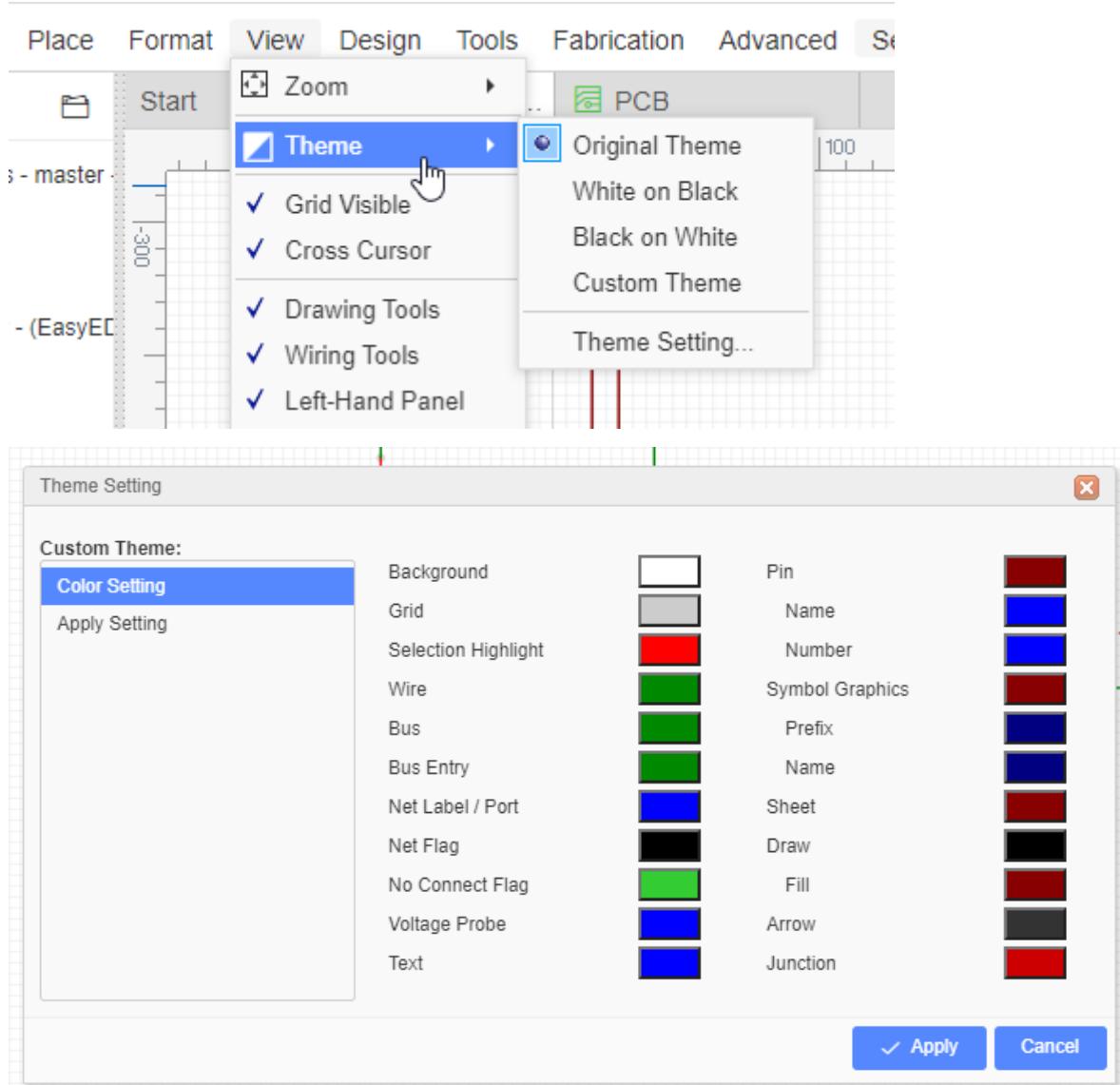
#### Notice:

- Module composes by tracks and components, it doesn't same as symbol binding footprint, the schematic module can not binding PCB module, after placing, the module will be separated by many objects, only the symbol and footprint can be corresponding via component ID, that is why you need to make the identification letter unique for placing each time to make sure schematic module corresponding with PCB module.

## Schematic Theme

EasyEDA support a powerful theme feature for the schematic design.

Via: Top Menu - View - Theme.



**Original Theme:** The default theme, only works for the new part placing.

**White on Black:** White on Black, the objects will be white, the background will be black.

**Black on White:** Black on White.

**User Defined:** When change to this theme style, the schematic will follow your theme options "My theme".

**My Theme:** Custom theme, which is stored locally in the browser and it will be synchronized to the server. When click apply, this theme will be applied to the current schematic. Next time you open the schematic, the theme of the schematic will be a custom theme.

**My theme Settings:** You can apply "My theme" on: 1. Creating New Schematic, 2. Opening Existed Schematic.

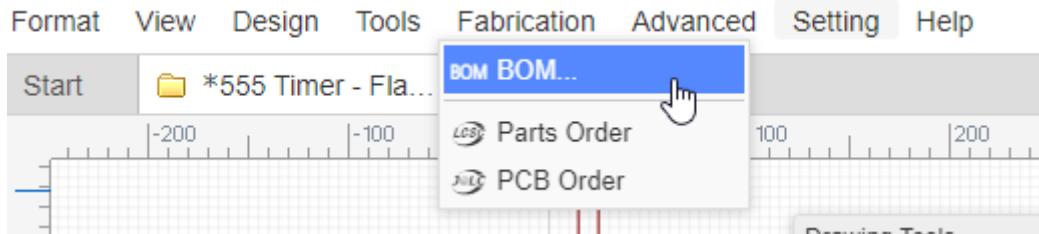
If you used any theme for the schematic, you need to UNDO to go back previous color theme. The "Original Theme" can't help.

Your schematic theme will synchronize to the server by default.

## Export BOM

---

You can export the Bill of Materials (BOM) for the schematic (Document) and PCB, via: "Top Menu - File - Export BOM", or "Top Menu - Fabrication - BOM".



After clicking the BOM export option, the dialog below will open.

In this dialog, you can click the button to assign LCSC part's order code for your components.

Export BOM

| ID | Name        | Designator | Footprint      | Qu... | Manufacturer Part  | Manufactu... | Supplier | Supplier Part | Price      |
|----|-------------|------------|----------------|-------|--------------------|--------------|----------|---------------|------------|
| 1  | 47k         | R1         | 0805-RESISTOR  | 1     | ?                  |              |          |               |            |
| 2  | 470R        | R2         | 0805-RESISTOR  | 1     | ?                  |              |          |               |            |
| 3  | 220R        | R31        | 0805-RESISTOR  | 1     | ?                  |              |          |               |            |
| 4  | 10u         | C1         | 0805           | 1     | ?                  |              |          |               |            |
| 5  | dddd        | U1         | SOIC-8_150MIL  | 1     | NE555DR            | TI           | LCSC     | C7593         | \$0.143... |
| 6  | Header-M... | H1         | HDR-2X1/2.54   | 1     | 826629-2           | TE Conne...  | LCSC     | C86471        | \$0.20275  |
| 7  | LED-3MM     | LED1       | LED-3MM/2.54   | 1     | 204-10SURD/S530-A3 | EVERLIGHT    | LCSC     | C99772        | \$0.0308   |
| 8  | LED-3MM     | LED2       | LED-3MM/2.5... | 1     | 204-10SURD/S530-A3 | EVERLIGHT    | LCSC     | C99772        | \$0.0308   |

2.5

After clicking on the assign icon, the components and footprints search dialog will pop up, and you can choose which component you want to assign.

Library

Search Engine: EasyEDA | 1k |

Types: Symbol | Spice Symbol

Classes: LCSC | JLCPCB Assembled

amp

Capacitors

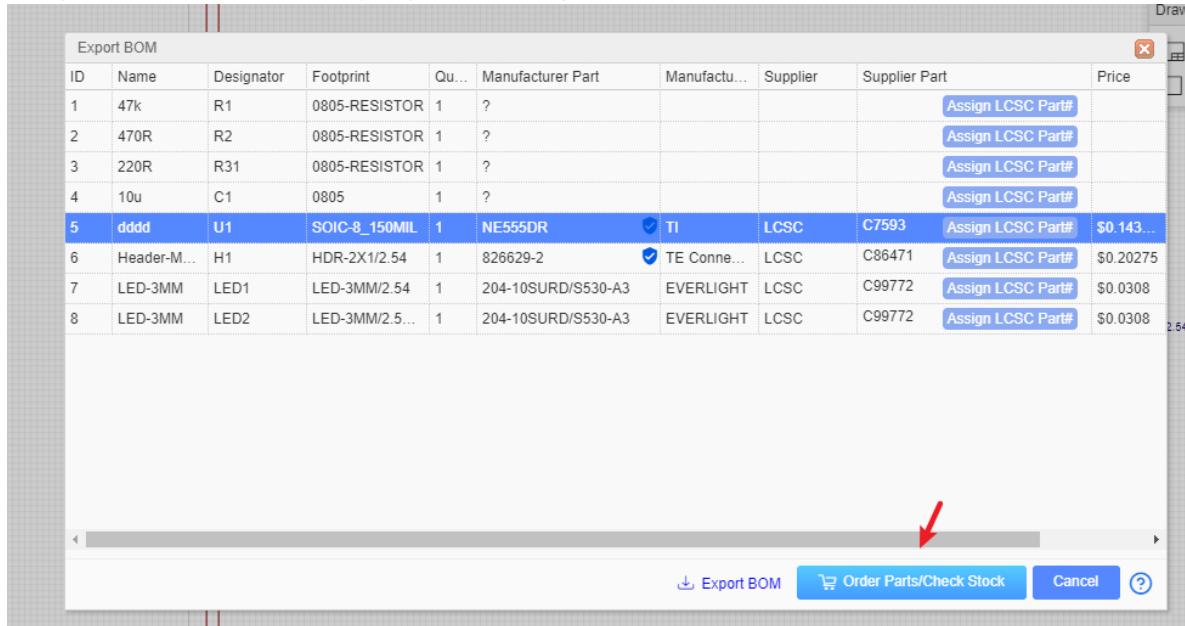
- CL21 Capacitor
- Mylar Capacitor
- Niobium Oxide Capacitors
- Capacitor Networks, Arrays
- Aluminum Electrolytic Ca...
- Polyester Film Capacitors
- Trimmers, Variable Capac...
- Aluminum Electrolytic Ca...
- Ceramic Disc Capacitors
- CBB Capacitors(polyprop...
- Multilayer Ceramic Capac...

| Title(PartNO)   | Footprint | Capacitance | Inductanc... |
|-----------------|-----------|-------------|--------------|
| NTCG164BH103JT1 | R0603     |             |              |
| ERTJ0EV104GM    | R0402     |             |              |
| ERTJ1VV154J     | R0603     |             |              |
| ERTJ1VR223G     | R0603     |             |              |
| ERTJ0EP333H     | R0402     |             |              |
| ERTJ1VA220H     | R0603     |             |              |
| ERTJ1VG103HA    | R0603     |             |              |
| ERTJZER104H     | R0201     |             |              |
| ERTJZEP473G     | R0201     |             |              |
| ERTJ1VT202H     | R0603     |             |              |
| ERTJ0EA680H     | R0402     |             |              |

EasyEDA > Symbol > LCSC > NTC Thermistors > NTCG164BH103JT1

\$0.0769

When you click the "Order Parts/Check Stock" button, we will help you to list all the components of your BOM at LCSC.com (If you haven't login LCSC, you have to login first). If you want to buy the components from LCSC, and you just need to put them to the cart and check out.



You can open the BOM in any text editor or spreadsheet.

|   | A  | B              | C        | D              | E          | F                        | G            | H        | I      | J      |
|---|----|----------------|----------|----------------|------------|--------------------------|--------------|----------|--------|--------|
| 1 | id | value          | quantity | package        | components | Manufacturer Part        | Manufacturer | Supplier | LCSC   | price  |
| 2 | 1  | 150            | 2        | AXIAL-0.3      | R1,R4      | 25121WJ020KT4F           | UniOhm       | LCSC     | C45278 | \$0.02 |
| 3 | 2  | 22k            | 2        | AXIAL-0.3      | R2,R3      | 25121WF300LT4F           | UniOhm       | LCSC     | C16074 | \$0.03 |
| 4 | 3  | 22u            | 2        | CAP-D3.0XF1.5  | C1,C2      | 1812B225K500NT           | FH           | LCSC     | C28503 | \$0.28 |
| 5 | 4  | 204-10UYC/S531 | 2        | LED-3MM/2.54   | LED1,LED2  | 67-21S/KK3C-H2727QAR3LEC | EVERLIGHT    | LCSC     | C73540 | \$0.04 |
| 6 | 5  | 2N3904         | 2        | TO-92(TO-92-3) | Q1,Q2      | MURA220T3G               | ON           | LCSC     | C37995 | \$0.17 |
| 7 |    |                |          |                |            |                          |              |          |        |        |

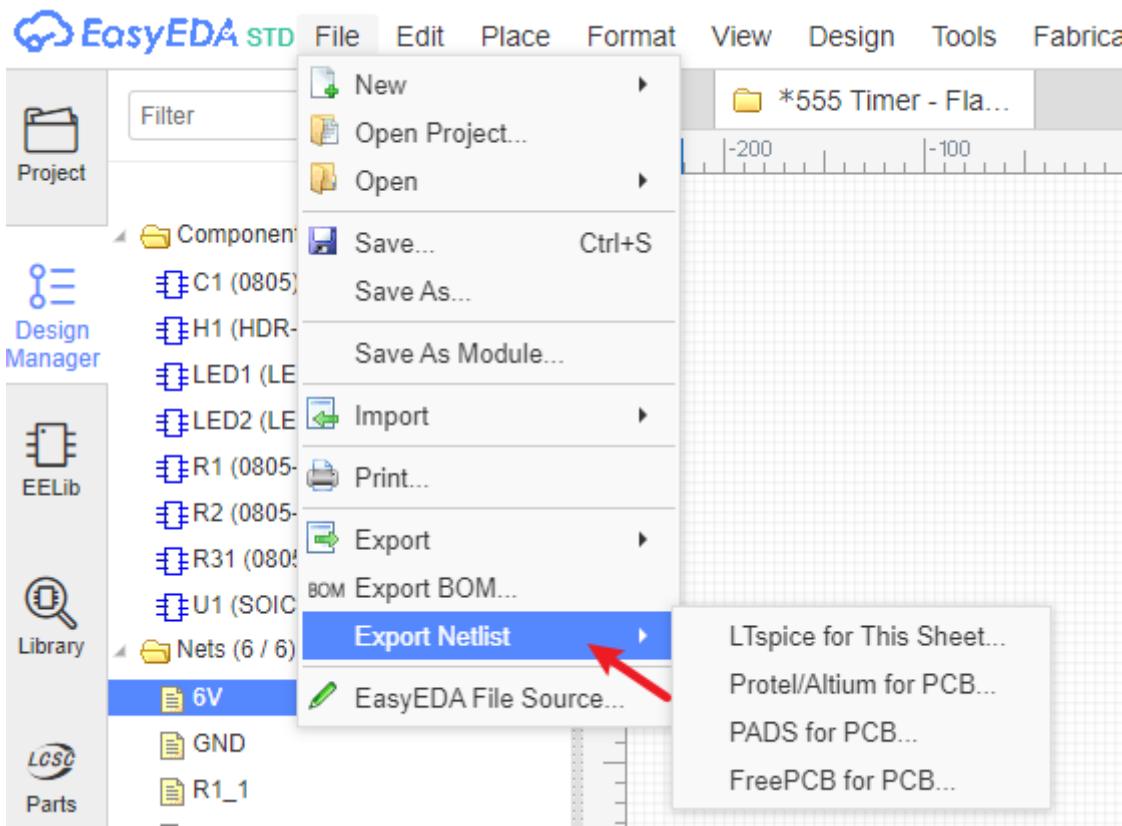
#### Notice:

- If your project has schematic and PCB, the BOM data will come from schematic; if the project only has PCB, the BOM data will come from PCB.
- In order to support multiple languages, BOM and coordinate files (CSV file) are UNICODE encoded and tab-based. If the CSV file cannot be read by your components vendor or PCB manufacturer, please convert the encoding and change the delimiter.
- Recommended solution: Save as a new CSV file in Excel or WPS. For example, open a CSV file in Excel, click or select: Save As - Other Formats - CSV (Comma Separated) (\*.csv). You can also open the CSV file with any text editor (such as Windows Notepad) and save as ANSI or UTF-8 encoding. If necessary, replace all tabs with commas.

## Export NetList

EasyEDA can export the netlist for the whole active project:

**File > Export NetList > Spice...**



EasyEDA can export a netlist in a variety of formats:

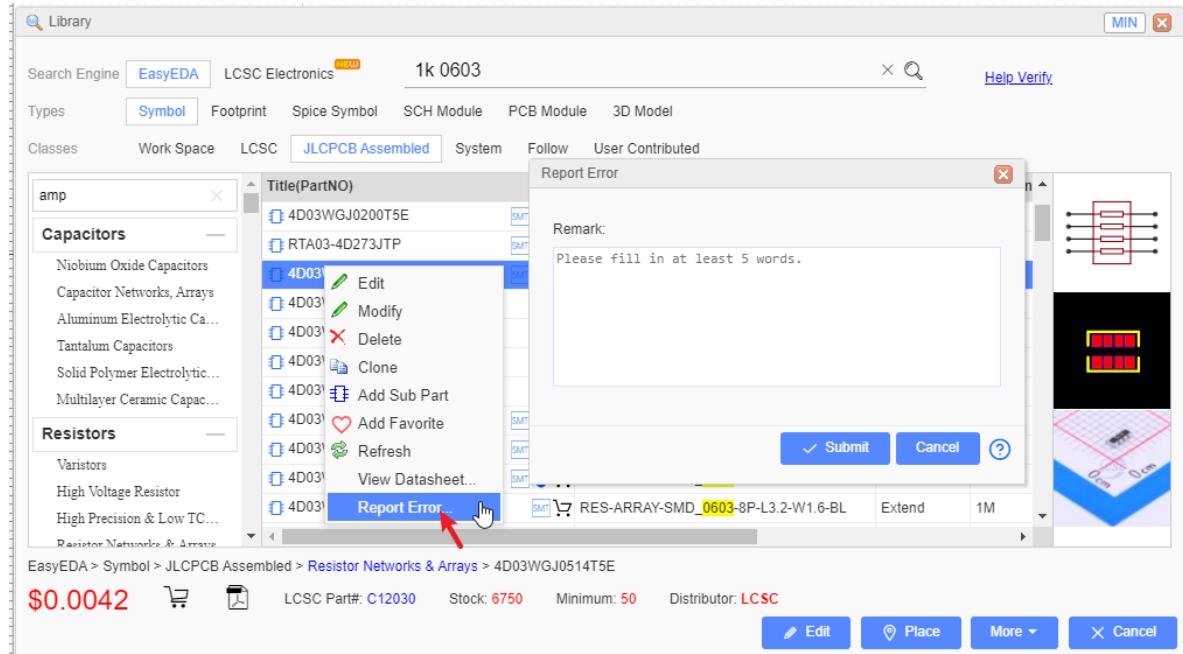
- **LTSpice for this Sheet:** this is a Spice compatible netlist generated by the simulation engine of EasyEDA, It is not normally used as the basis for as a PCB layout.
- **Protel/Altium for PCB:** a PCB netlist in a format that can be imported straight into Altium Designer and it's predecessor, Protel.
- **PADS for PCB:** a PCB netlist in a format that can be imported straight into Pads PCB layout tools.
- **FreePCB for PCB:** a PCB netlist in a format that can be imported straight into FreePCB, a free, open source PCB editor for Windows.

## Report Error

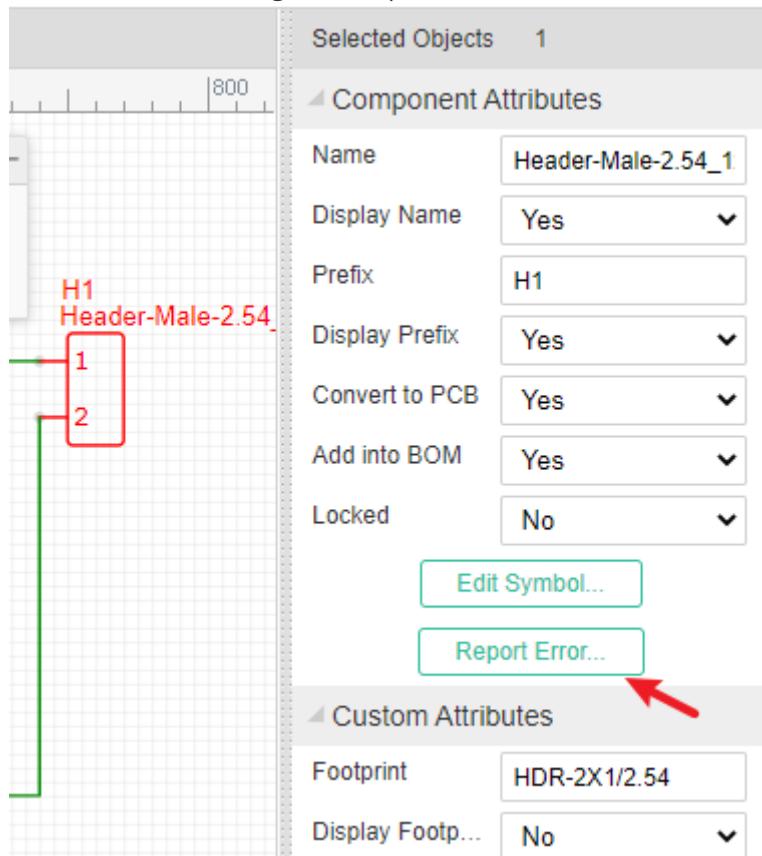
For EasyEDA official libraries, we have staffs to draw and maintain(LCSC & JLCPCB Assembled part) and we will try to keep them correctly as we can, but EasyEDA(System part) included a lot of open source of the libraries and the official drawing of the libraries, that can not avoid the wrong situation 100%, so when you meet a incorrect library, Please inform us in time, we will fix it as soon as possible.

There are 3 ways to report error:

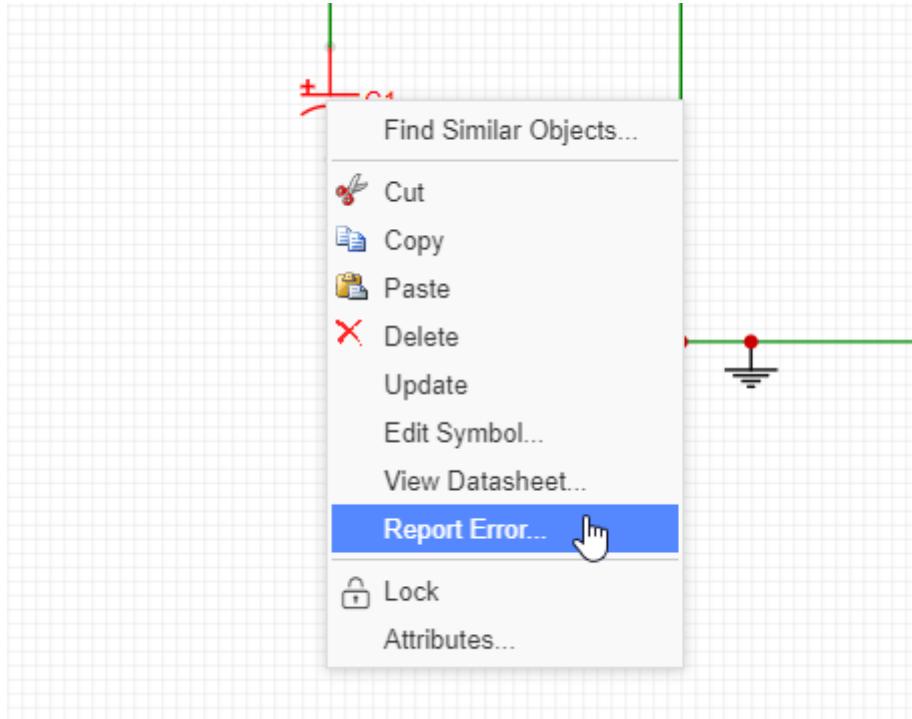
1.Right-click the official library and use the "Report Error" function on the "Libraries".



2.Select the official library on the canvas of the schematic/schematic module, click the "Report Error" button at the right-hand panel.



or right-click the component:



3. Send Email to us or post a topic at [Bug report](#)

[support@easyeda.com](mailto:support@easyeda.com)

# Create the Schematic Symbol

## Create the Schematic Symbol

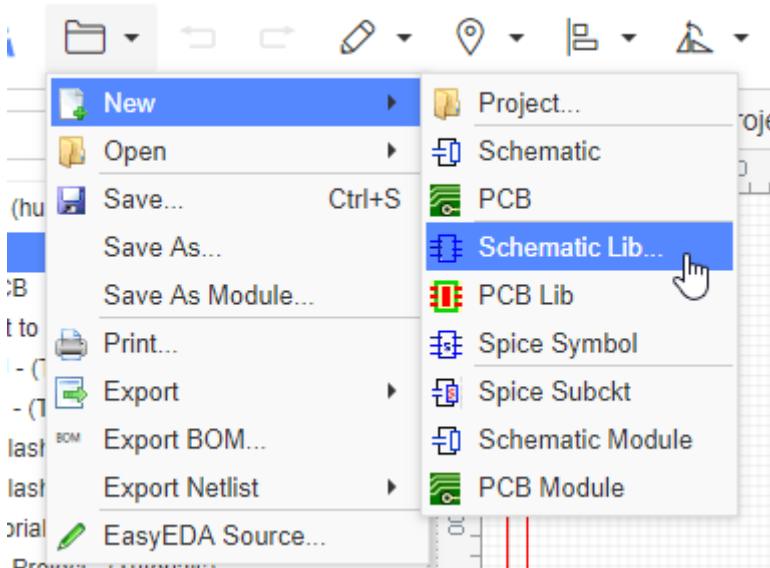
Using **Schematic Symbol Wizard** and **Group/Ungroup...** is a quick way to create schematic symbols but they are placed directly into the schematic that they are built in.

It is possible to reuse them by copying them (`CTRL+C` hotkeys) from the schematic they were created in and then cross-document-pasting them (`CTRL+SHIFT+V` hotkeys) into a different schematic but this quickly gets messy if you need to copy symbols that were created in several different schematics.

OK, you could keep copying new symbols into a dedicated “symbol library” schematic sheet to save searching for them but EasyEDA offers you an easier way to create and manage your symbols in a library.

Start a new Schematic Lib as shown below or by doing:

## 1. File > New > Symbol



This opens the New SchematicLib symbol editor.

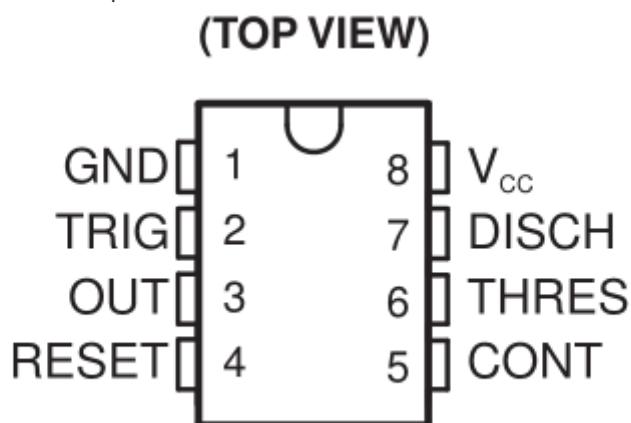
## 2. Create the symbol

### • Get the Datasheet

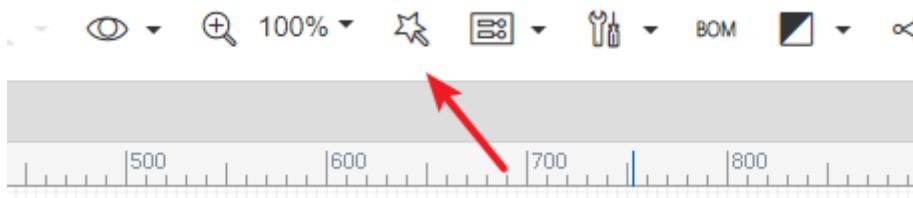
For example, using the NE555DR, the datasheet you can refer [LCSC: NE555DR](#).

And then create the symbol and place the pins for the library base on the datasheet.

This component have 8 pins and names.



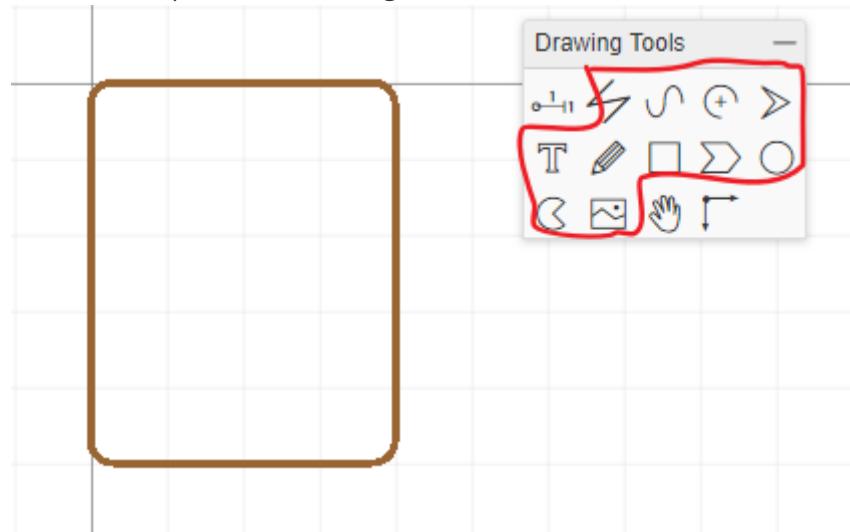
### • Create via Schematic Symbol Wizard



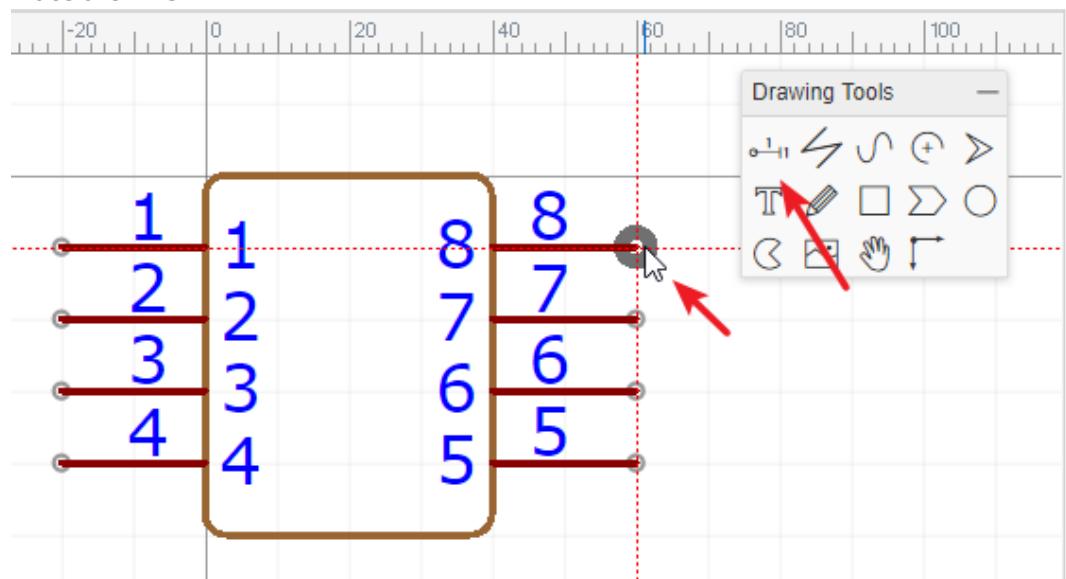
The more information of **Schematic Symbol Wizard** please refer next section.

### • Create by Manually

- Draw the shape via the Drawing Tools



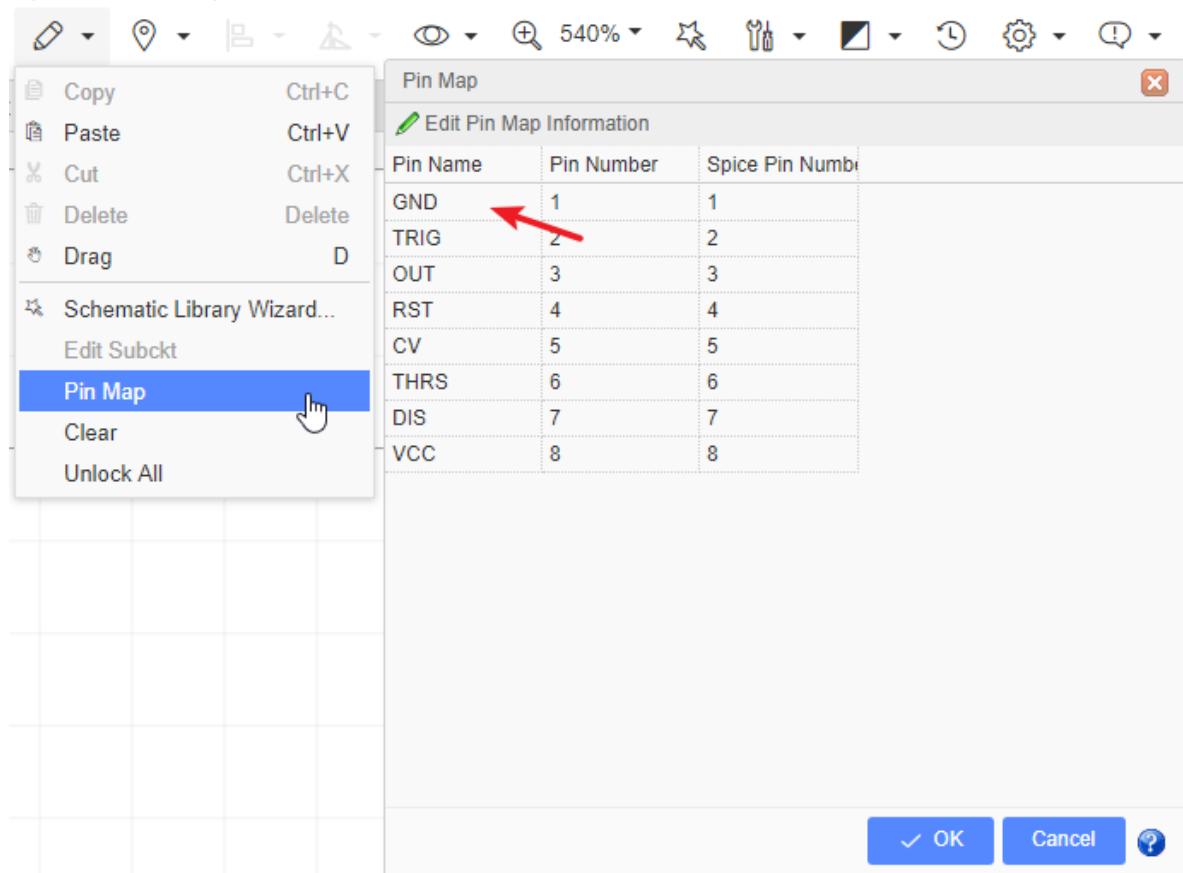
- Place the Pins



The Pin dot must keep out side as the image indicated, it is connecting with the wires.  
The more information please refer **SchematicLib Attributes - Pins** Section.

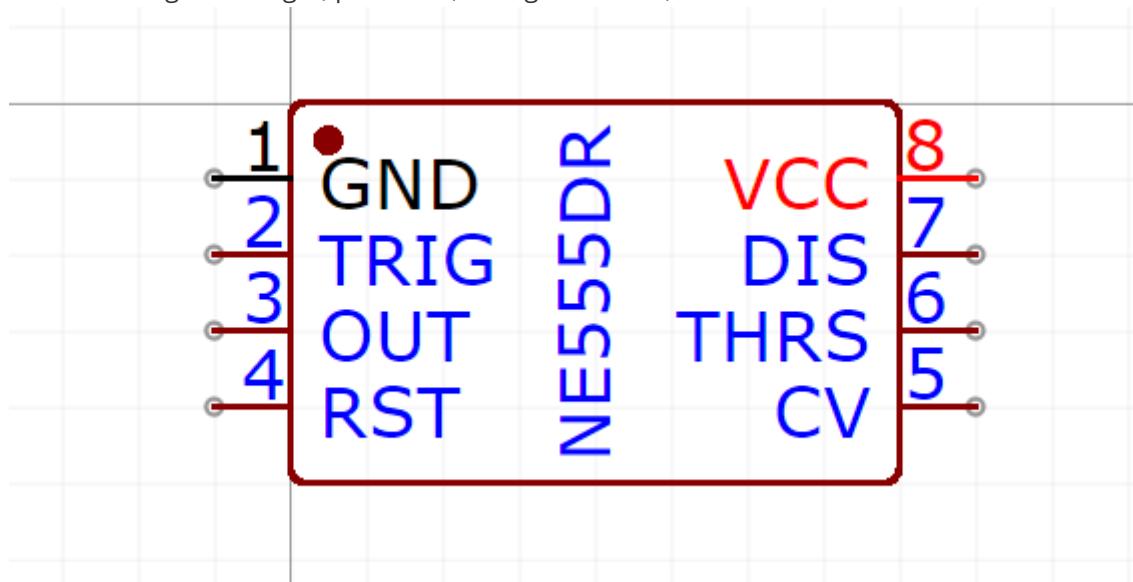
### 3. Edit the pin map

Via **Edit > Pin Map...**, change Pin names and Pin numbers. For some complicated IC, will use the alphabet for the pin number.



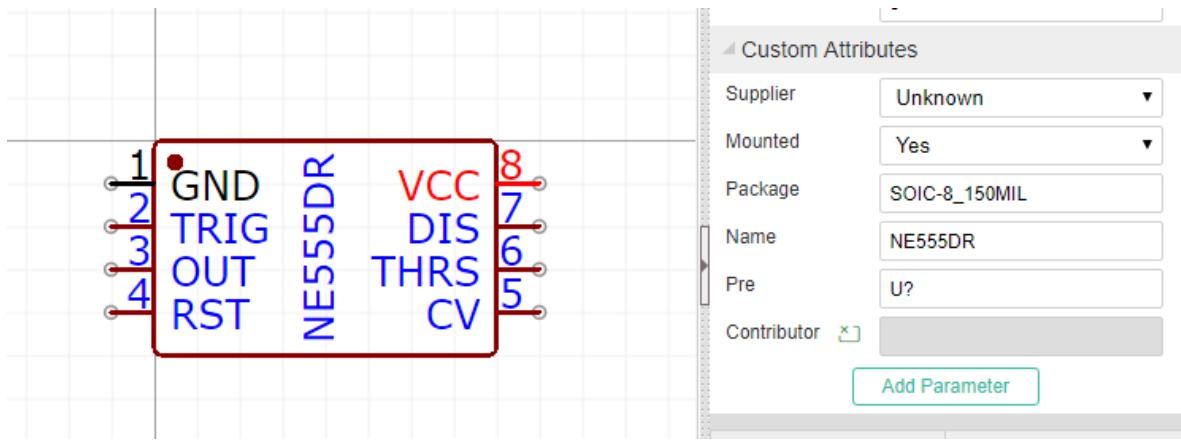
### 4. Modify the Detail

such as change Pin length, place text, change Pin color, Pin attributes etc.



### 5. Set Custom Attributes

You can set the supplier, footprint(Suggested, you must assign the footprint via "Footprint Manager"), Name(Required), Prefix(Required) for it, the more detail of attributes please refer below section: **Custom Attributes**



If the schematiclib need to assign the packahe, the Pin number should match the footprint's Pad number. The detail of the footprint assign please refer the **Footprint Manager** section at previous.

- If the part's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

## 6. Set the Origin

You can via: "Top Menu - Place - Set Canvas Origin - By Center Grid of Symbols" to set the origin.

## 7. Save your SchameticLib

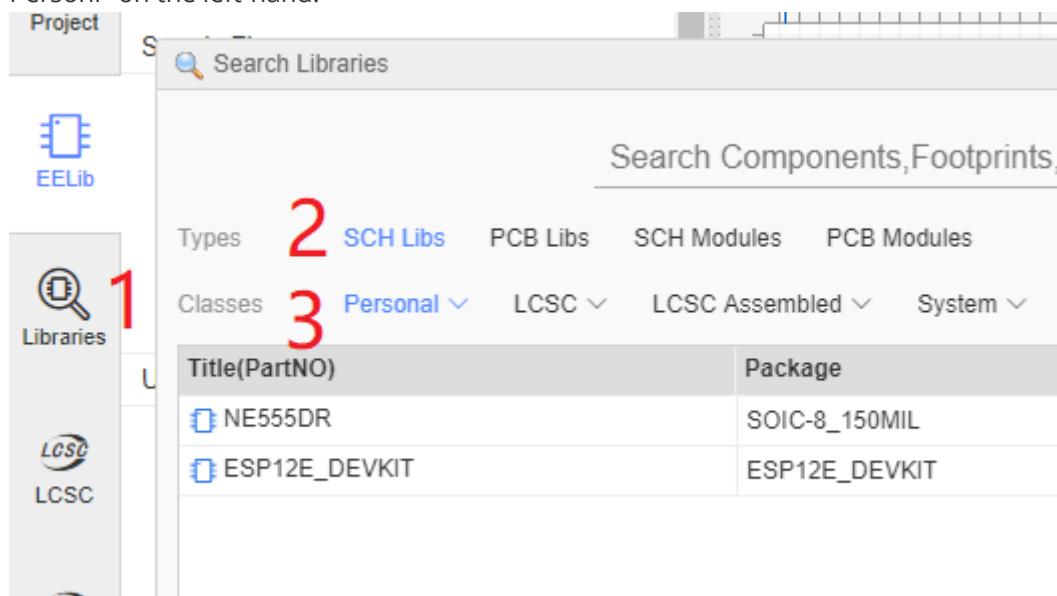
You can set this library's owner, datasheet link and tags etc.

Save as a schematic Lib

|                       |                                                                                                     |                             |
|-----------------------|-----------------------------------------------------------------------------------------------------|-----------------------------|
| Owner:                | Tutorials                                                                                           | <a href="#">Create Team</a> |
| Title:                | NE555DR                                                                                             |                             |
| Manufacturer Part:    | NE555DR                                                                                             |                             |
| Supplier:             | Unknown                                                                                             | Or Others                   |
| Supplier Part Number: | 296-6501-2-ND                                                                                       |                             |
| Link:                 | <a href="http://www.ti.com/lit/ds/symlink/ne555.pdf">http://www.ti.com/lit/ds/symlink/ne555.pdf</a> |                             |
| Tags:                 | 555 Timer                                                                                           |                             |
| Description:          | 555 Timer                                                                                           |                             |

[✓ Save](#) [Cancel](#)

Then a Schematic Symbol is created finish. And the you can find it at "Libraries - SchematicLib - Personl" on the left-hand.



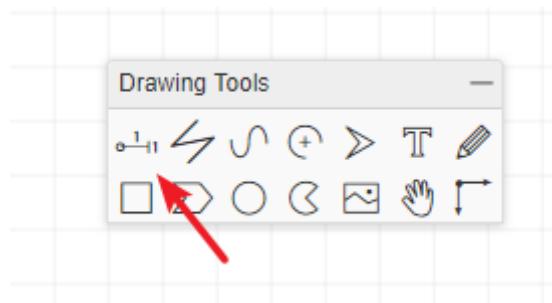
#### Notice:

- **Note the Origin Point.** To simplify rotating your symbols when they are placed into the canvas, make sure all of your symbols are created as near as possible centered around that point. Suggesting the first Pin/Pad or its center to be the origin point.
- *Please make sure all pins dot are placed on the grid, otherwise, when place the library on the schematic will causing the wiring difficult.*

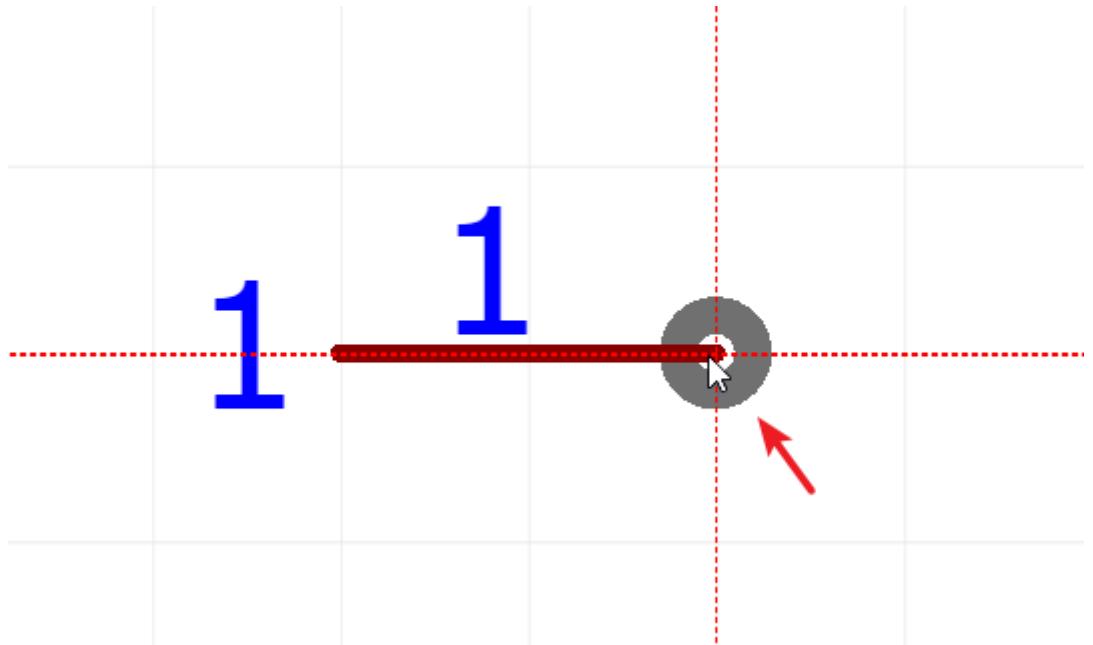
## Pin Attributes

Symbols pins are the most important part of any Schematic Lib symbol. They are the things that allow wires to be attached to symbols to connect up your circuit.

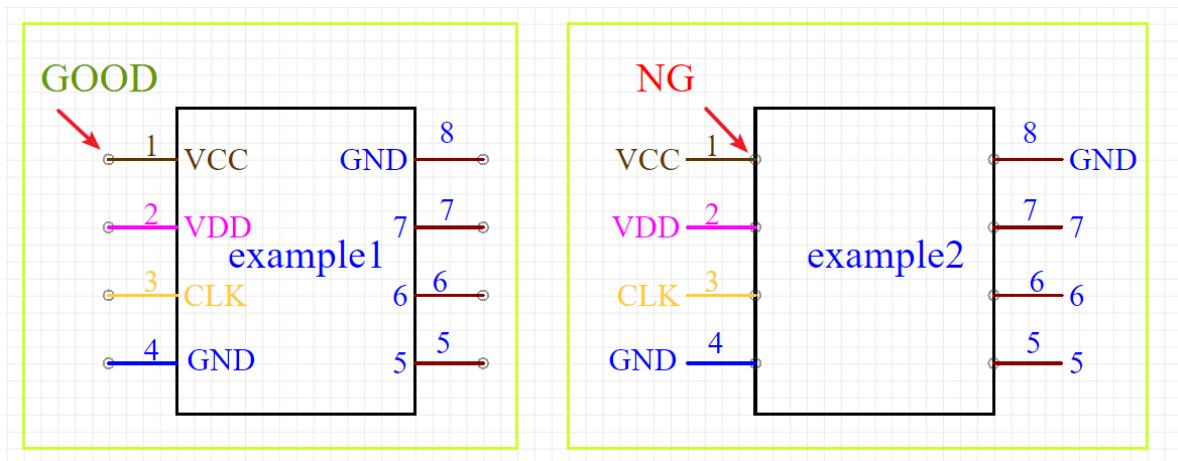
You can use the **P** hotkey to add a Pin or from the Draw Tools palette:



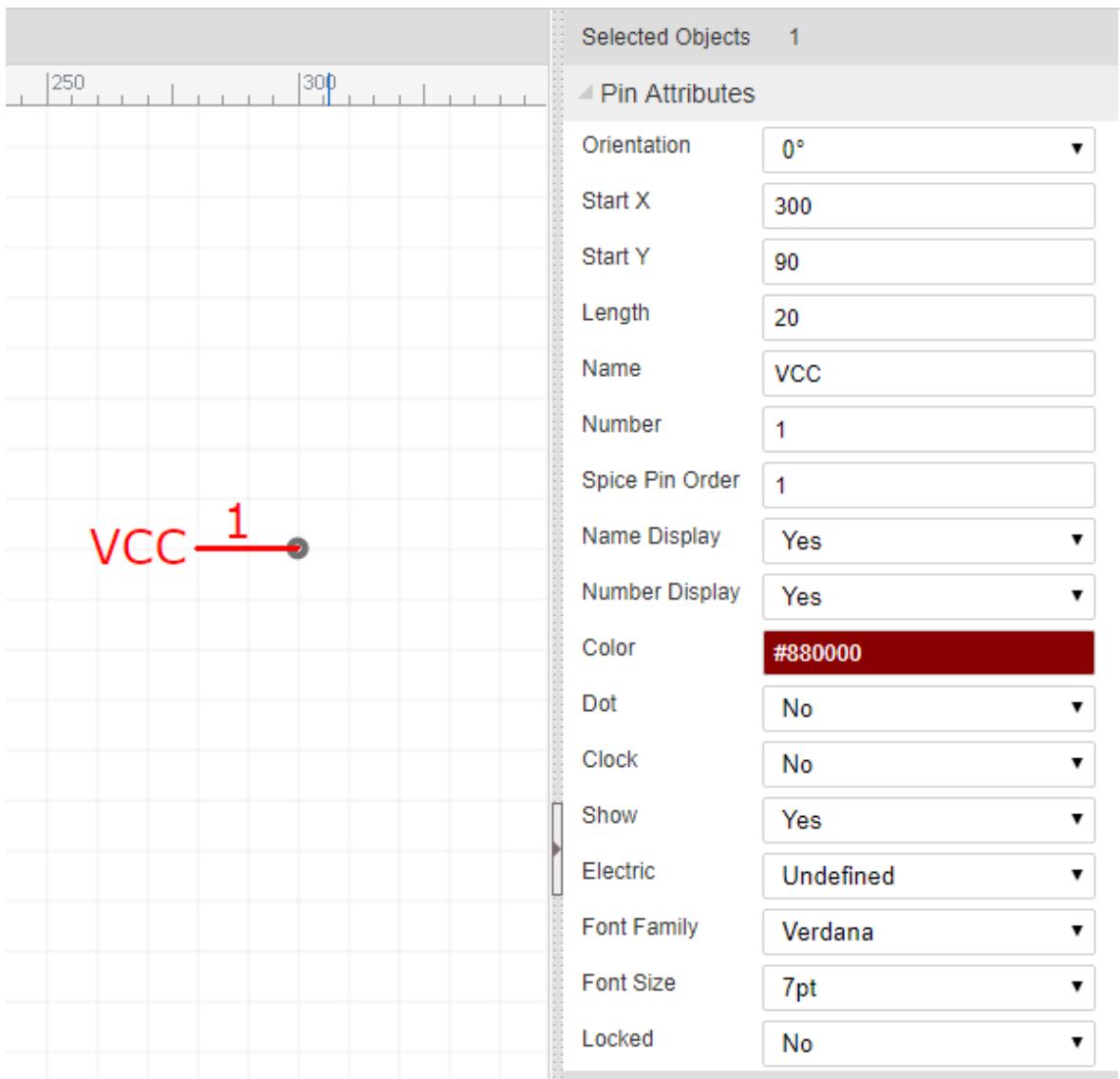
Before placing it on the canvas, you can use the rotation hotkey or rotate and flip from the menu to rotate it to the right orientation. Make sure the **Pin Dot(black dot)** is in the right position. The **Pin Dot** will be used to connect your wires or netlabels. Whenever a PIN is either placed as directly onto the canvas or as part of a symbol, the mouse has to point to the **Pin Dot** position to automatically start the Wire mode or to join a wire to it.



Whenever a Pin is placed as part of a symbol, the **Pin dot** should be **outside** of — and pointing away from — the symbol like in example 1(correct position), inside or pointing towards the symbol as shown in example 2(wrong position).



When you select a single Pin, the **Pin attributes** will be shown in the right hand **Properties** panel:



**Orientation:** 0°, 90°, 180° and 270°. If you want to create a 45° pin, you need to set its length as 0, and draw a line with 45°.

**Start-X and Start-Y:** The pindot position. Sometimes it may be difficult to move the pin to the desired position using the mouse, so you can move the pin via Start-X and Start-Y.

**Length:** Pin length.

**Name:** In this example, VCC is the name of the Pin.

**Number:** In this example, 1 is the number of the Pin. This number is the pin number of the device in a physical footprint.

Note that you can use alphanumeric identifiers such as; A1, B1, C1, A2, B2 and so on as the Number.

**Spice Number:** These are the pin numbers used to connect your symbol to the corresponding pins defined by the .model or .subckt used to simulate your device. The pin numbers of the simulation model may be different from the physical footprint pin numbers and - unless the model is specifically created to model multiple devices in a single footprint - do not change for different instances of a device in a multi-device footprint. The Spice Pin order must be **numerical** only.

**Name Display:** If you don't want to show VCC, switch it to NO.

**Number Display:** If you don't want to show 1, switch it to NO.

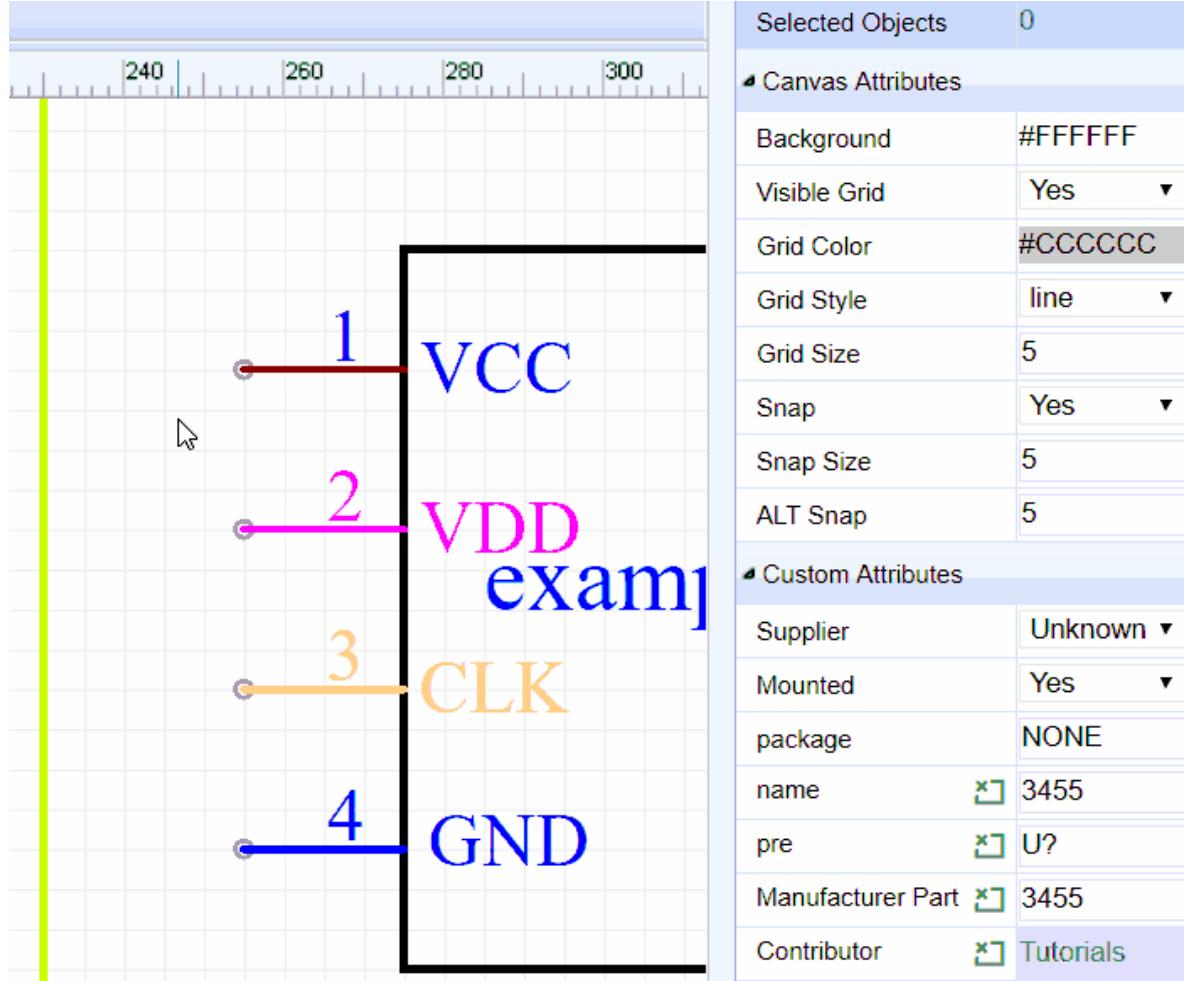
You can adjust the Name or Number position using your mouse but note that rotate and flip applies to the whole pin including the name and pin number; these items cannot be rotated and flipped independently of the pin itself.

Note also that rotate and flip actions do not result in upside down or mirrored pin number or names.

**Color:** You can set the Pin to different colours, such as *PIN3:CLK* as orange and *PIN4:GND* as blue. In this example, the *PIN1* is set as color `#880000`, but it shows as red, because it is selected. After deselecting it, the pin will appear color `#880000`.

**Dot:** adds a circle to the inside end of the pin to indicate logical (or analogue) inversion.

**Clk:** adds a `>` to the inside end of the pin to indicate that the pin is logical clock input.



**Show:** YES/NO. Allows you to hide the pin. When set it to NO, this Pin will be hidden when the symbol is placed on the schematic editor canvas, and then create a net which name same as this pin name.

Note that the pin is not hidden here in the Schematic Lib symbol editor canvas because if it was, it would disappear from view and so how would you find it to make it visible again? For the same reason this option has no effect in symbols made using Group/Ungroup...

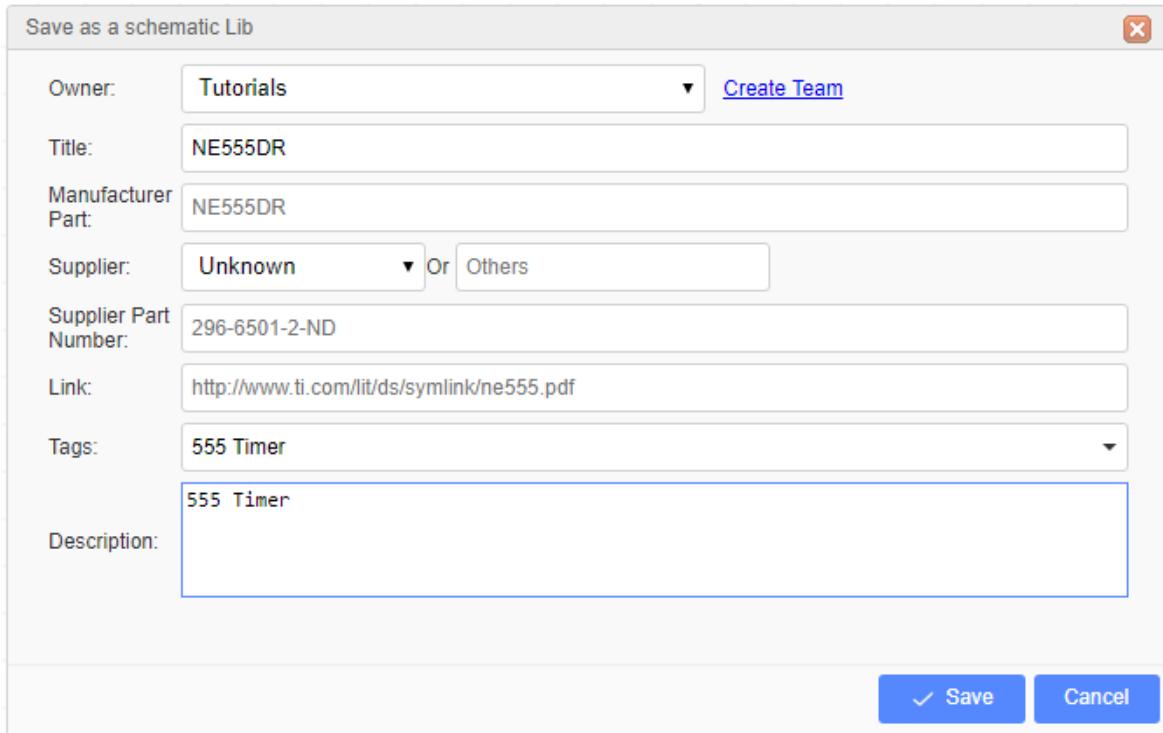
*We may not have thought of everything in EasyEDA but we do try. :)*

**Electric:** [Undefined, Input, Output, I/O, Power]

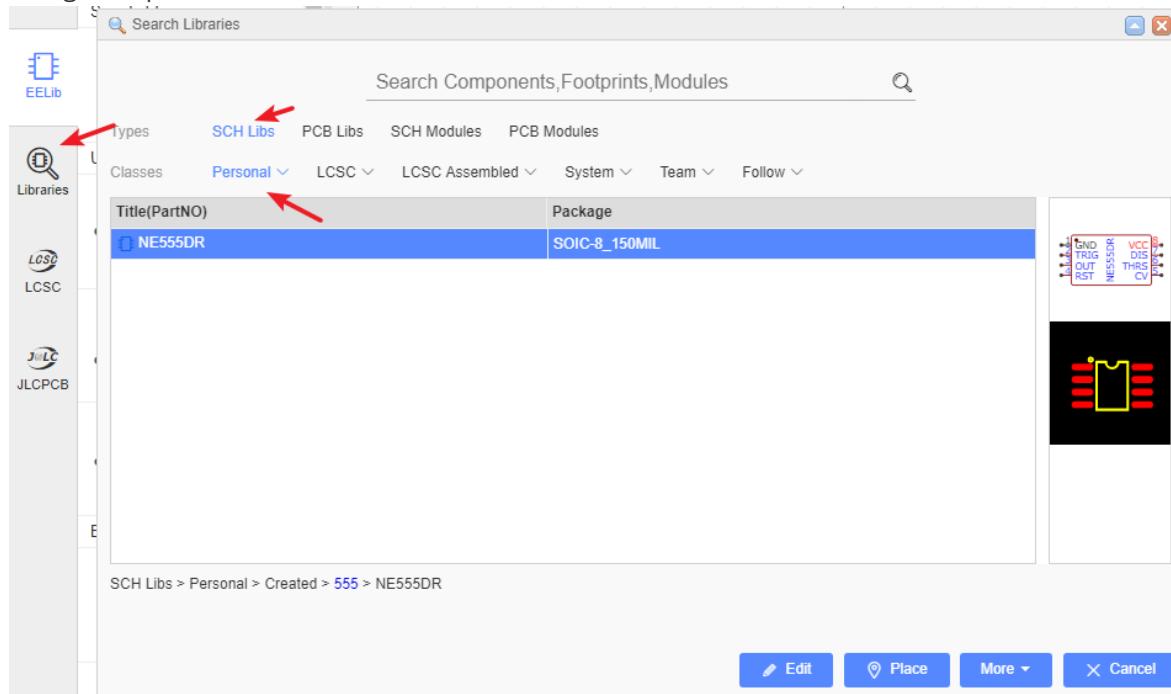
EasyEDA provides Electrical Rules Checking (ERC) right now, But you still need to set electric of your Schematic libs.

If you set the PIN as Power and set the pin to be hidden, then the Pin will be connected by Name which is the NetLabel. If the Name is VCC, it will be connected to the net in your circuit with the NetLabel or NetFlag VCC. This helps to keep the schematic clear and uncluttered when using Multi-part Components.

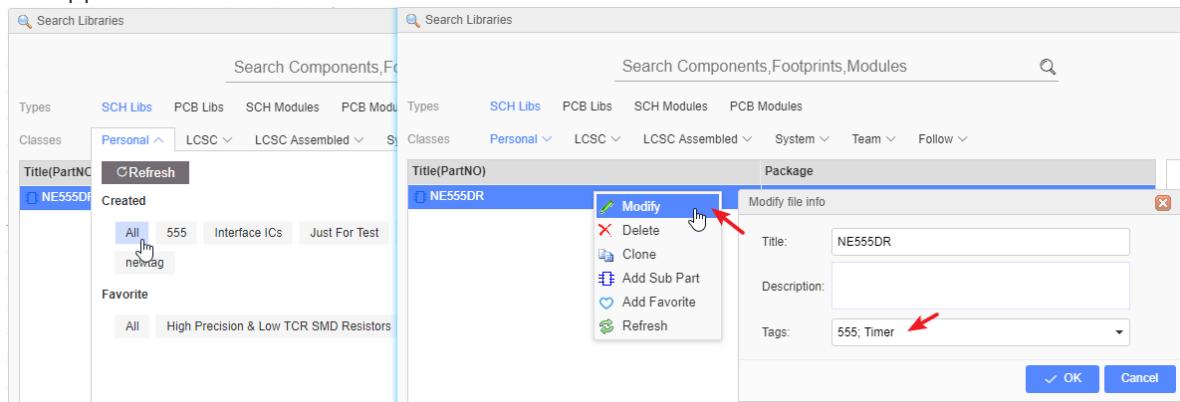
After creating the Lib, use **CTRL+S** will open the save dialog:



After clicking **Save**, you will see it appears in **Libraries > Symbols > Personal** of the left hand Navigation panel.

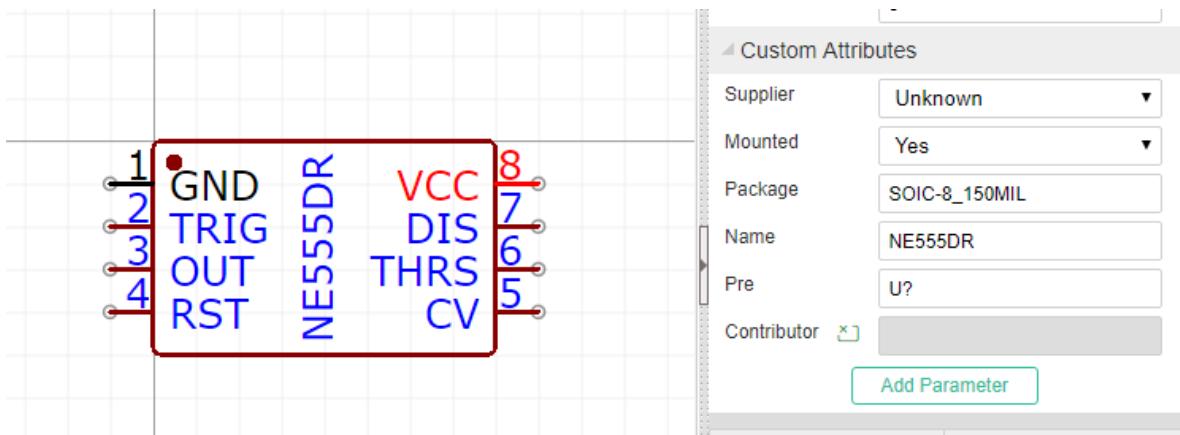


If you want to modify the tag for your new symbol: **Libraries > Symbols > Personal > Select New Lib > More > Modify**, or right-click new Lib > Modify, if your Lib doesn't have the tags it will appears on All.



## Custom Attributes

In the Schematic Lib editor's canvas Properties panel, you will find a **Custom Attributes** section:



- **footprint**

How to change Schematic Symbol's footprint? If you would like to built a PCB, you need to assign a footprint for your Schematic symbol. Although there are other ways to do this in EasyEDA, here is the right place to do it. When you set a footprint, **the footprint's pad numbers must match the schematic Lib's pin number**, otherwise, when you convert the schematic to PCB, there will miss several nets.

Click in the **footprint** input box, and the **Footprint Manager** dialog will open as used to do this task in the Schematic Editor.

The more information please refer to **Schematic - Footprint Manager** section.

**Notie:**

*You have to assign the footprint via the Footprint Manager, otherwise, the Schematic lib will not get the footprint correctly. The footprint is linked with SchematicLib by global unique ID not the title.*

- **Prefix**

The default Schematic symbol Prefix is **U?** If you create a resistor, you can set the Prefix to **R?**. It is filled required.

- **Name**

You can change the schematic lib's name here, it is can be different from the part's file name.

- **Contributor**

This is your registered user name. When Other EasyEDA's users use your libraries, they will remember your contributions!

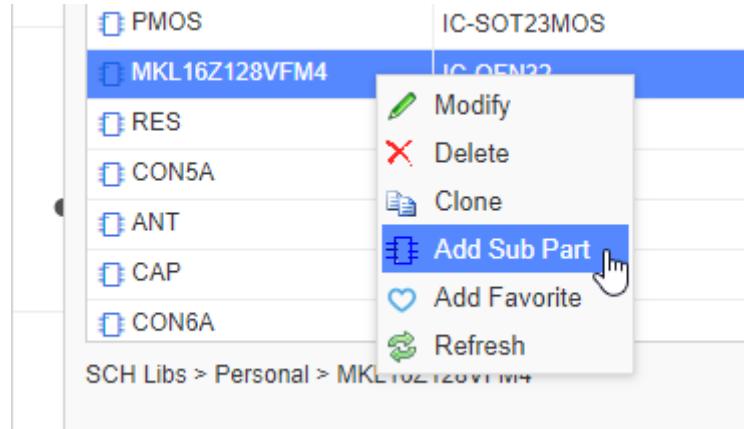
## Symbol Subparts

We have already touched on how EasyEDA can support **Multi-part/Subpart Components**, but how do you create **multi-part components**?

EasyEDA provides a sub parts facility to do this.

After creating a part, you can right-click the part in the **Library > Symbols > Work Space > Created** section to pop up the content menu.

Suppose you have created your own symbol for a 74HCT04 hex inverter.



Right Click **Add sub part** and that will add 74HCT04.1,

Click again to add 74HCT04.2 , up to 74HCT04.6.

Then double click on each sub part in turn to modify the Pin Name and Number attributes.

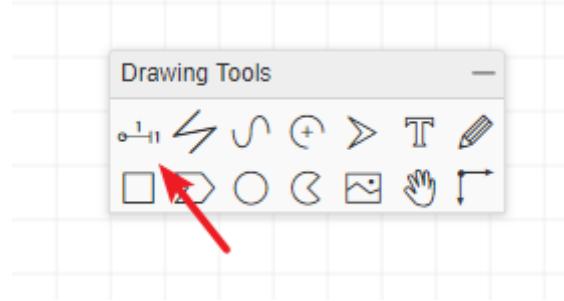
Easy or what?

## Schematic Symbol Attributes

### Pin Attributes

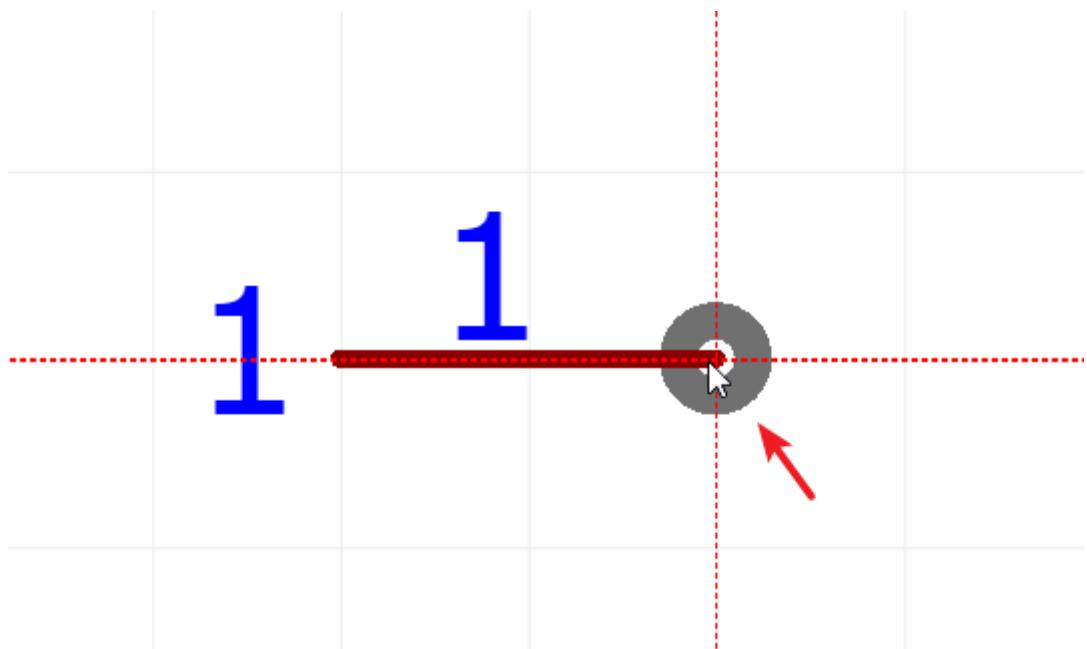
Symbols pins are the most important part of any Schematic Lib symbol. They are the things that allow wires to be attached to symbols to connect up your circuit.

You can use the **P** hotkey to add a Pin or from the Draw Tools palette:

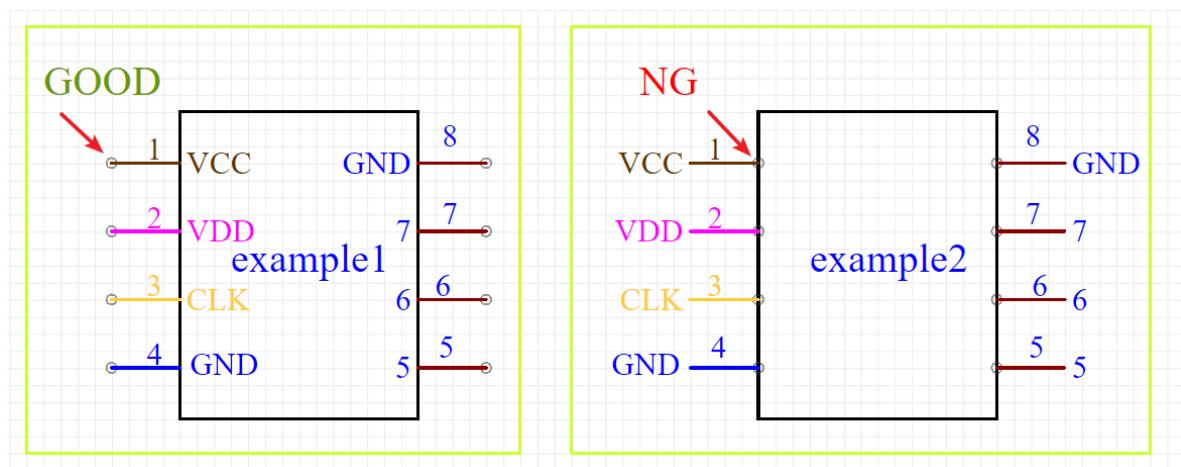


## Pin Orientation

Before placing it on the canvas, you can use the rotation hotkey or rotate and flip from the menu to rotate it to the right orientation. Make sure the **Pin Dot(black dot)** is in the right position. The **Pin Dot** will be used to connect your wires or netlabels. Whenever a PIN is either placed as directly onto the canvas or as part of a symbol, the mouse has to point to the **Pin Dot** position to automatically start the Wire mode or to join a wire to it.

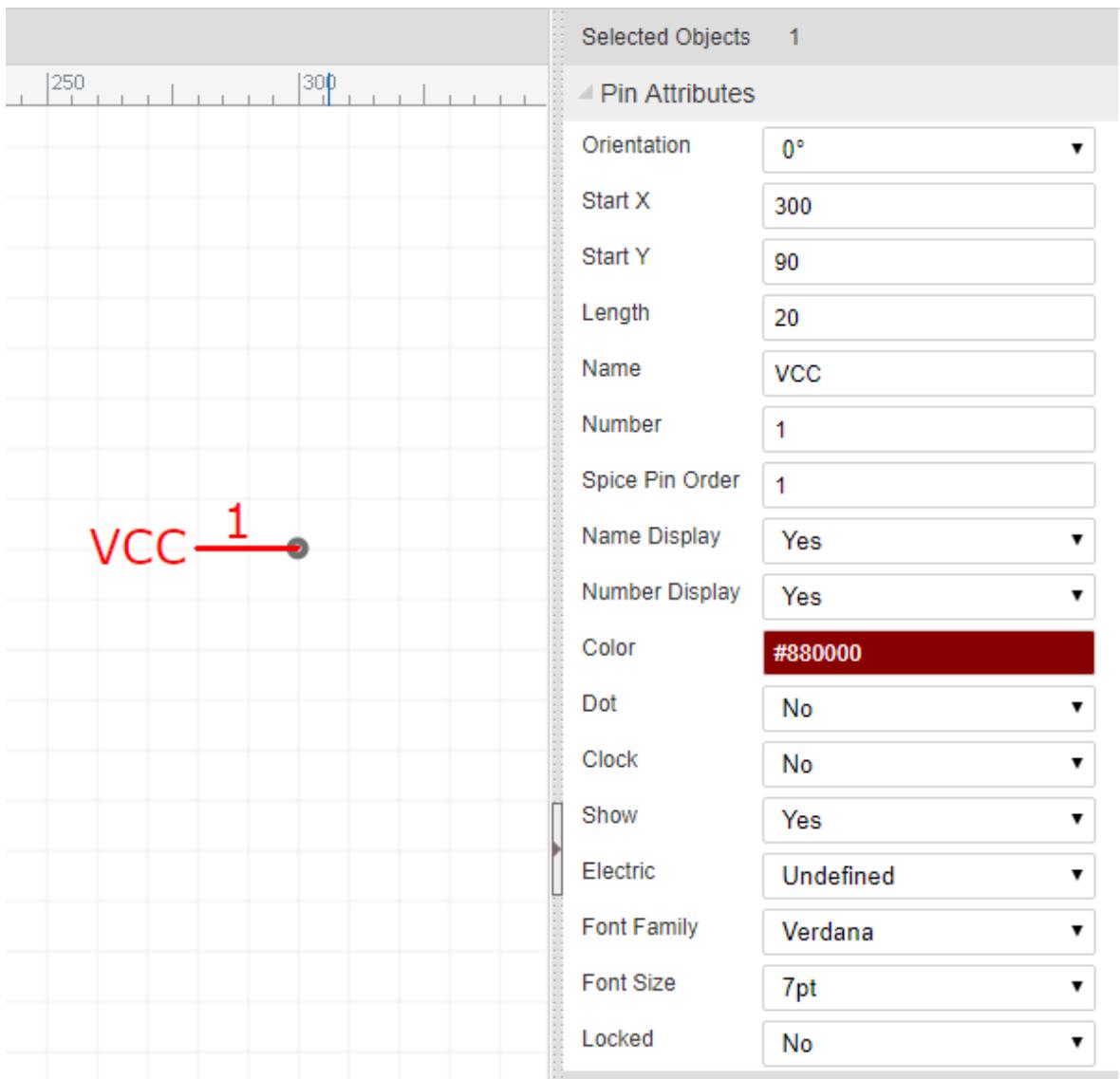


Whenever a Pin is placed as part of a symbol, the **Pin dot** should be **outside** of — and pointing away from — the symbol like in example 1(correct position), inside or pointing towards the symbol as shown in example 2(wrong position).



## Pin Attributes

When you select a single Pin, the **Pin attributes** will be shown in the right hand **Properties** panel:



**Orientation:** 0°, 90°, 180° and 270°. If you want to create a 45° pin, you need to set its length as 0, and draw a line with 45°.

**Start-X and Start-Y:** The pindot position. Sometimes it may be difficult to move the pin to the desired position using the mouse, so you can move the pin via Start-X and Start-Y.

**Length:** Pin length.

**Name:** In this example, VCC is the name of the Pin.

**Number:** In this example, 1 is the number of the Pin. This number is the pin number of the device in a physical footprint

Note that you can use alphanumeric identifiers such as; A1, B1, C1, A2, B2 and so on as the Number.

**Spice Number:** These are the pin numbers used to connect your symbol to the corresponding pins defined by the .model or .subckt used to simulate your device. The pin numbers of the simulation model may be different from the physical footprint pin numbers and - unless the model is specifically created to model multiple devices in a single footprint - do not change for different instances of a device in a multi-device footprint. The Spice Pin order must be **numerals** only.

**Display Name:** If you don't want to show VCC, switch it to NO.

**Display Number:** If you don't want to show 1, switch it to NO.

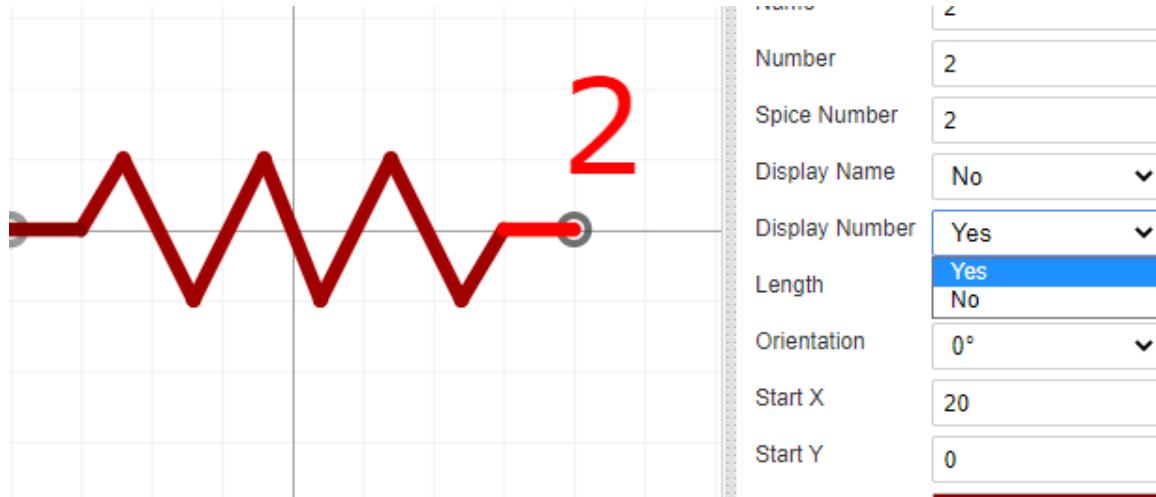
You can adjust the Name or Number position using your mouse but note that rotate and flip applies to the whole pin including the name and pin number; these items cannot be rotated and flipped independently of the pin itself.

Note also that rotate and flip actions do not result in upside down or mirrored pin number or names.

**Color:** You can set the Pin to different colours, such as *PIN3:CLK* as orange and *PIN4:GND* as blue. In this example, the PIN1 is set as color `#880000`, but it shows as red, because it is selected. After deselecting it, the pin will appear color `#880000`.

**Dot:** adds a circle to the inside end of the pin to indicate logical (or analogue) inversion.

**Clock:** adds a `>` to the inside end of the pin to indicate that the pin is logical clock input.



**Show:** YES/NO. Allows you to hide the pin. When set it to NO, this Pin will be hidden when the symbol is placed on the schematic editor canvas.

Note that the pin is not hidden here in the Schematic Lib symbol editor canvas because if it was, it would disappear from view and so how would you find it to make it visible again? For the same reason this option has no effect in symbols made using Group/Ungroup...

*We may not have thought of everything in EasyEDA but we do try. :)*

**Electric:** [Undefined, Input, Output, I/O, Power]

EasyEDA provides Electrical Rules Checking (ERC) right now, But you still need to set electric of your Schematic libs.

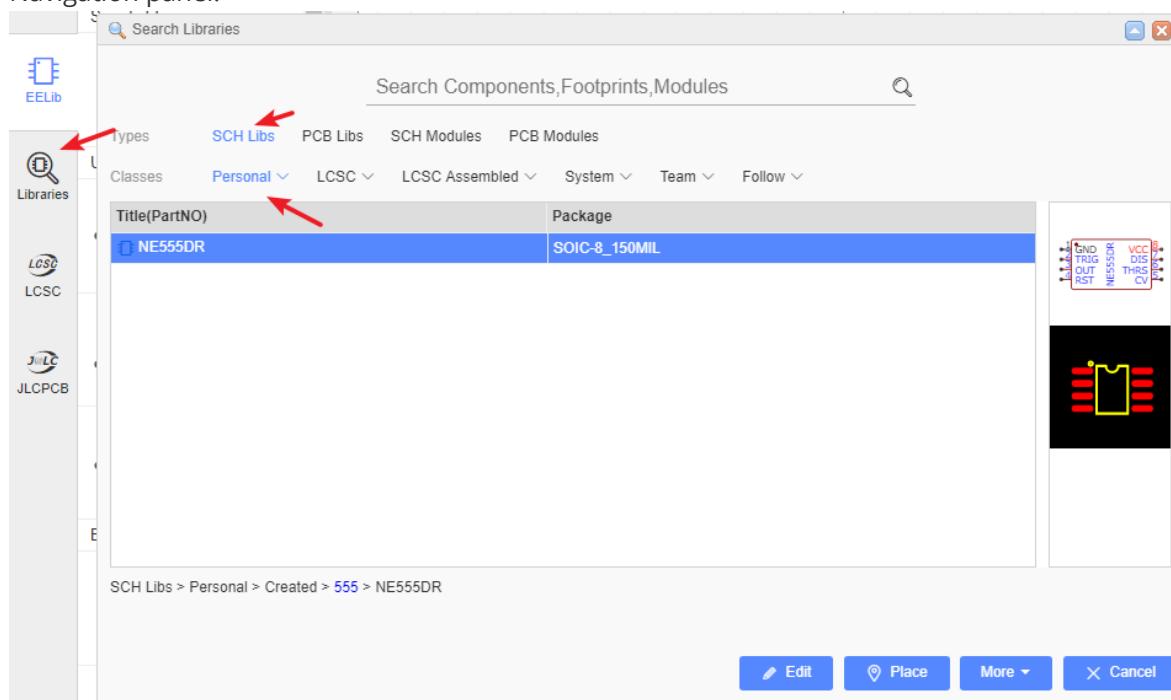
If you set the PIN as Power and set the pin to be hidden, then the Pin will be connected by Name which is the NetLabel. If the Name is VCC, it will be connected to the net in your circuit with the NetLabel or NetFlag VCC. This helps to keep the schematic clear and uncluttered when using Multi-part Components.

After created the Lib, use `CTRL+S` will open the save dialog:

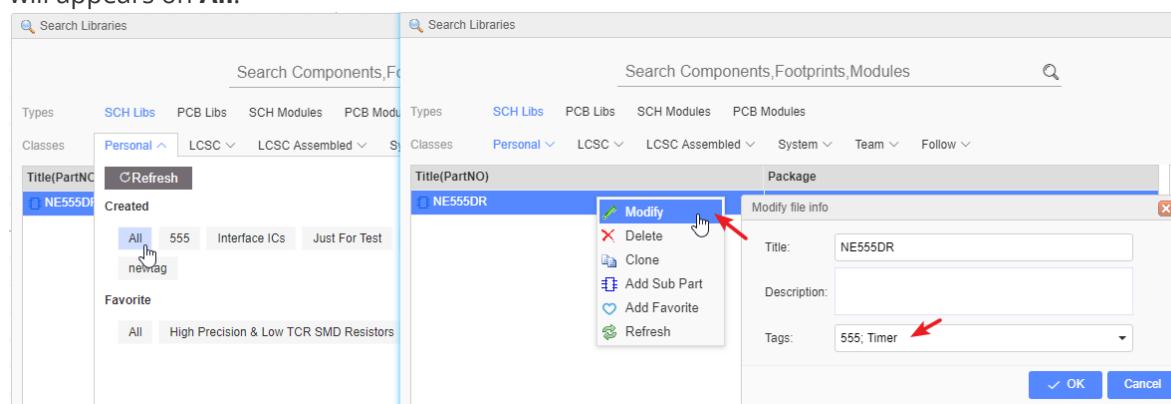
Save as a schematic Lib

|                                                                             |                                                                                                     |                             |
|-----------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------|-----------------------------|
| Owner:                                                                      | Tutorials                                                                                           | <a href="#">Create Team</a> |
| Title:                                                                      | NE555DR                                                                                             |                             |
| Manufacturer Part:                                                          | NE555DR                                                                                             |                             |
| Supplier:                                                                   | Unknown                                                                                             | Or Others                   |
| Supplier Part Number:                                                       | 296-6501-2-ND                                                                                       |                             |
| Link:                                                                       | <a href="http://www.ti.com/lit/ds/symlink/ne555.pdf">http://www.ti.com/lit/ds/symlink/ne555.pdf</a> |                             |
| Tags:                                                                       | 555 Timer                                                                                           |                             |
| Description:                                                                | 555 Timer                                                                                           |                             |
| <input type="button" value="✓ Save"/> <input type="button" value="Cancel"/> |                                                                                                     |                             |

After clicking **Save**, you will see it appears in **Libraries > Symbols > Personal** of the left hand Navigation panel.

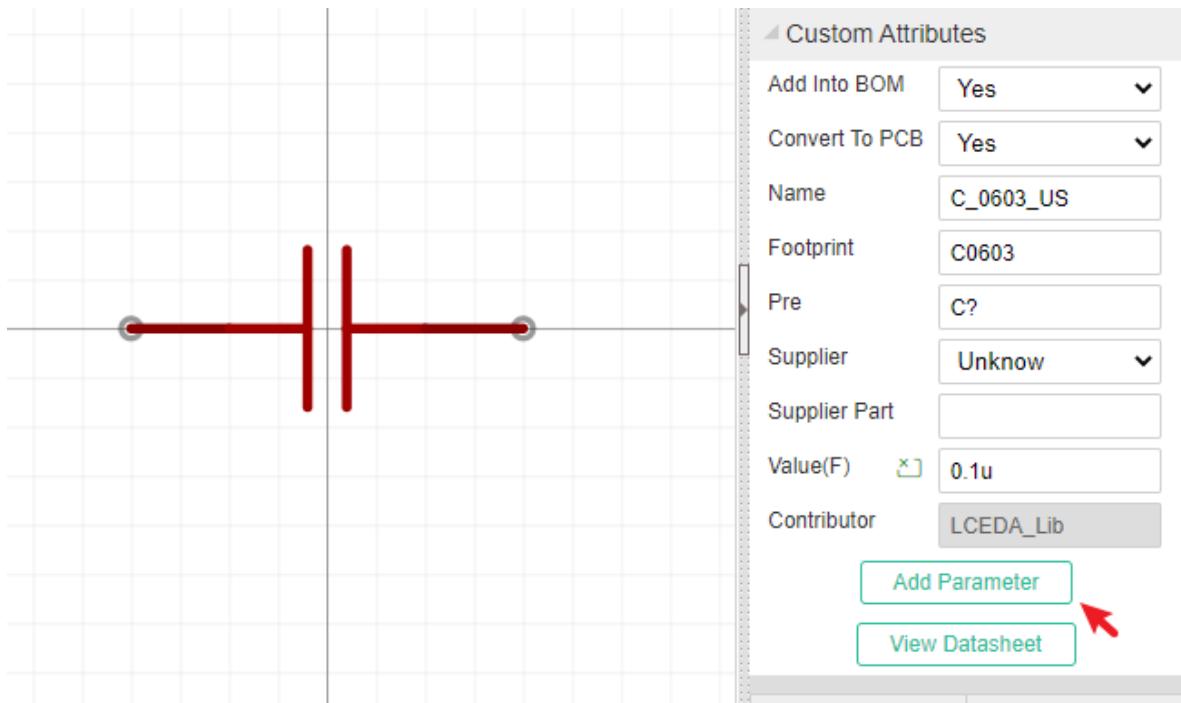


If you want to modify the tag for your new symbol: **Libraries > Symbols > Personal > Select New Lib > More > Modify**, or **right-click new Lib > Modify**, if your Lib doesn't have the tags it will appears on All.



# Symbol Custom Attributes

In the Schematic Lib editor's canvas Properties panel, you will find a **Custom Attributes** section:



- **Add into BOM**

This part display at BOM or not.

- **Convert to PCB**

If you set it as No, this part will not display at Footprint Manager and can't not convert to PCB.

- **footprint**

To assign a footprint for this part. Only assign one footprint.

The more information please refer to **Schematic - Footprint Manager** section.

**Notie:**

*You have to assign the footprint via the Footprint Manager, otherwise, the Schematic Symbol will not corresponding the Footprint correctly. The Footprint is linked with Symbol by global unique ID not the title.*

- **Pre**

The default Schematic symbol Prefix is **U?** If you create a resistor, you can set the Prefix to **R?**. It is filled required.

- **Name**

You can change the schematic lib's name here, it is can be different from the part's file name.

- **Contributor**

This is your registered user name. When Other EasyEDA's users use your libraries, they will remember your contributions!

## Show symbol value as component name when place component at schematic

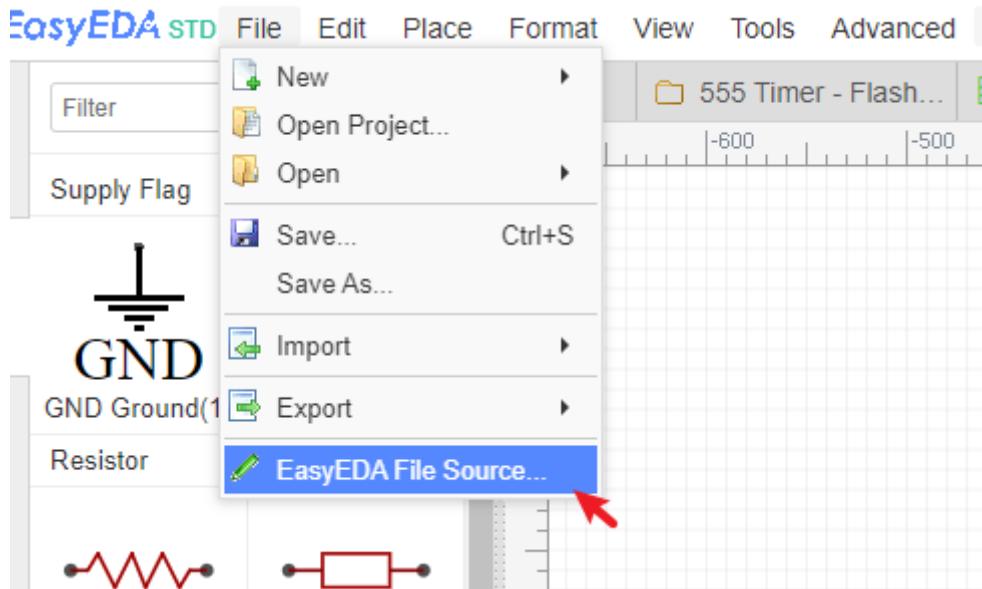
For example, a resistor symbol vaule is  $2K\Omega$ , name is ABC, but when place it at schematic, it will not show  $2K\Omega$  as component name, the name is ABC. You can change name to  $2K\Omega$ , but it not very well.

EasyEDA doesn't support common function to support this feature yet.

But, we can edit the symbol file source to implement this feature.

How do it works:

1. Finish symbol and parameter edit.
2. Open file source. via: Top Menu - File - EasyEDA File Source.



3. Add or modify the parameter: nameAlias.

The screenshot shows a 'EasyEDA Source' dialog box. It contains a text area with JSON code for a component symbol. A red box highlights a section of the code where 'nameAlias' is set to 'Value(Ω)' and 'Value(Ω)' is set to '1k'. The JSON code is as follows:

```
{
 "head": {
 "docType": "2",
 "editorVersion": "6.4.3",
 "newgId": true,
 "c_para": {
 "package": "R0603",
 "name": "R_0603_US",
 "pre": "R?",
 "nameAlias": "Value(Ω)",
 "Value(Ω)": "1k",
 "BOM_Supplier Part": "",
 "BOM_Supplier": "",
 "Contributor": "LCEDA_Lib"
 },
 "c_spiceCmd": null,
 "hasIdFlag": true,
 "---"
 }
}
```

This symbol will show 1k as component name after placing at schematic.

4. Apply after modified, and save.

You can double click the EElab resistor symbol to get an example.

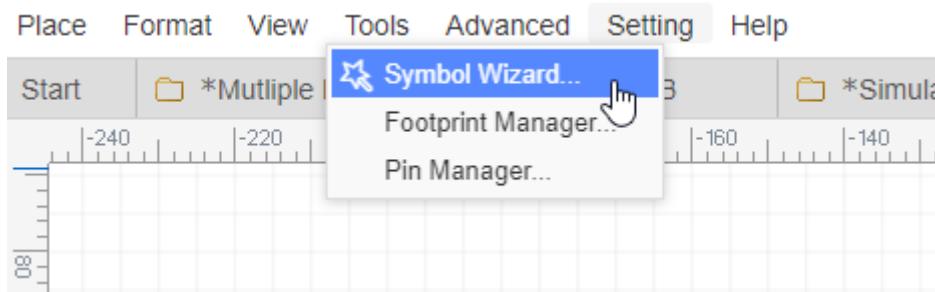
EasyEDA will provide it as a feature in the future.

## Schematic Symbol Wizard

How many times have you hit a schematic capture roadblock because you couldn't find a component symbol?

Well, in EasyEDA that would be never because the **Schematic Symbol Wizard** provides a quick and easy way to create a general Schematic Symbol symbol.

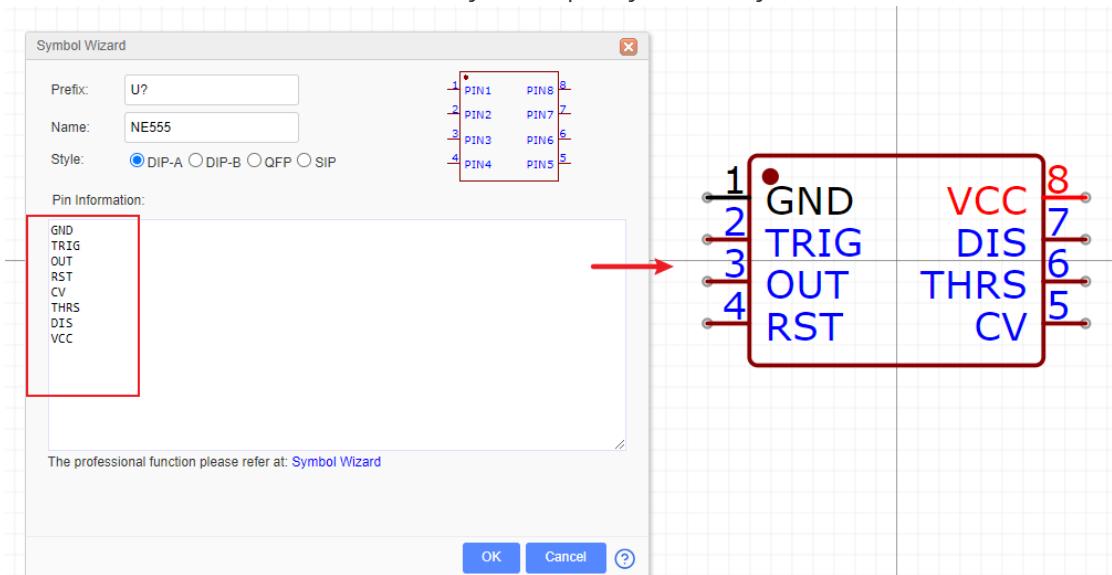
Via: Top Menu - Tools - Symbol Wizard



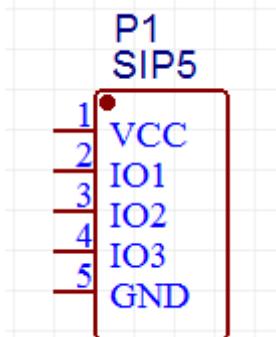
## Basic Function

### Input the Pins' name Only

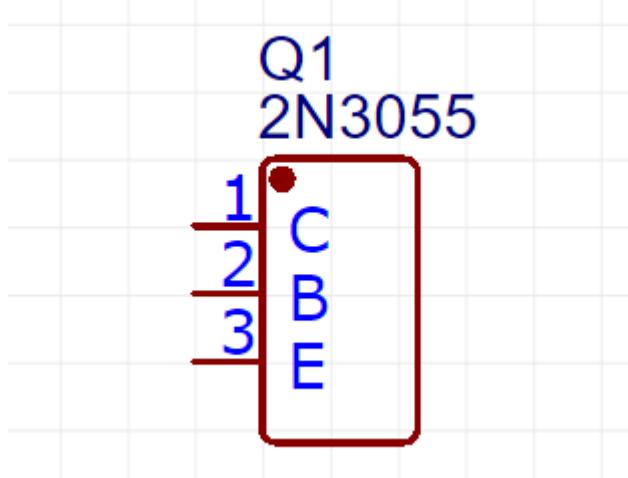
1. Using the **NE555** timer as an example: this device is available in a **DIP8** package so select **DIP**. Then enter the NE555 pin names into the **Pin Names** text box separated by new line or space, Then press OK. Abracadabra! As if by magic, you will find a perfectly formed dual in line 8 pin symbol for the NE555 attached to your mouse cursor, ready to be placed! You just need a few seconds to build a NE555 symbol, quickly and easily.



2. The EasyEDA Schematic Symbol Wizard allows you to create DIP, QPF or SIP styles symbols. If you are designing Arduino Shields then you will need lots of SIP symbol, so you can create a SIP symbol like the one shown below in a few seconds.

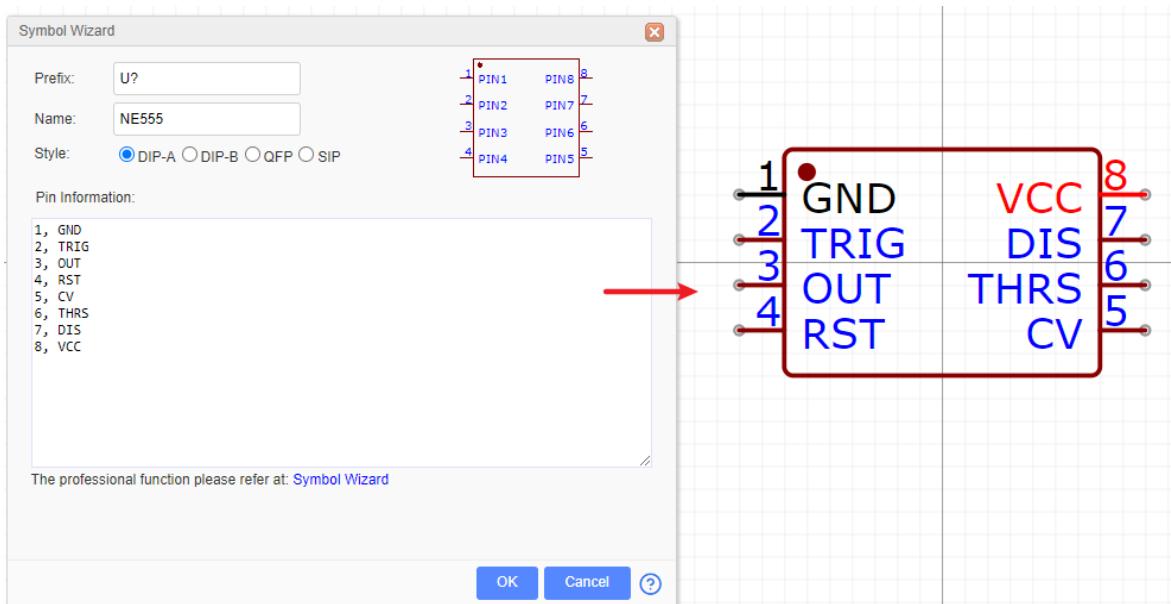


3. If you are not too worried that the symbols may not look quite the way people might expect and that they may not look anything like the Type you select, then of course you can use the wizard to create symbols for any component:



## Input the Pins' number and name

Schematic Symbol wizard support you input the pins' number and name. As below example, setting every pin's number is easily.



## Professional Function

Schematic Symbol Wizard support the professional function, it is easier to create the large and complex and more convenient Schematic Symbol.

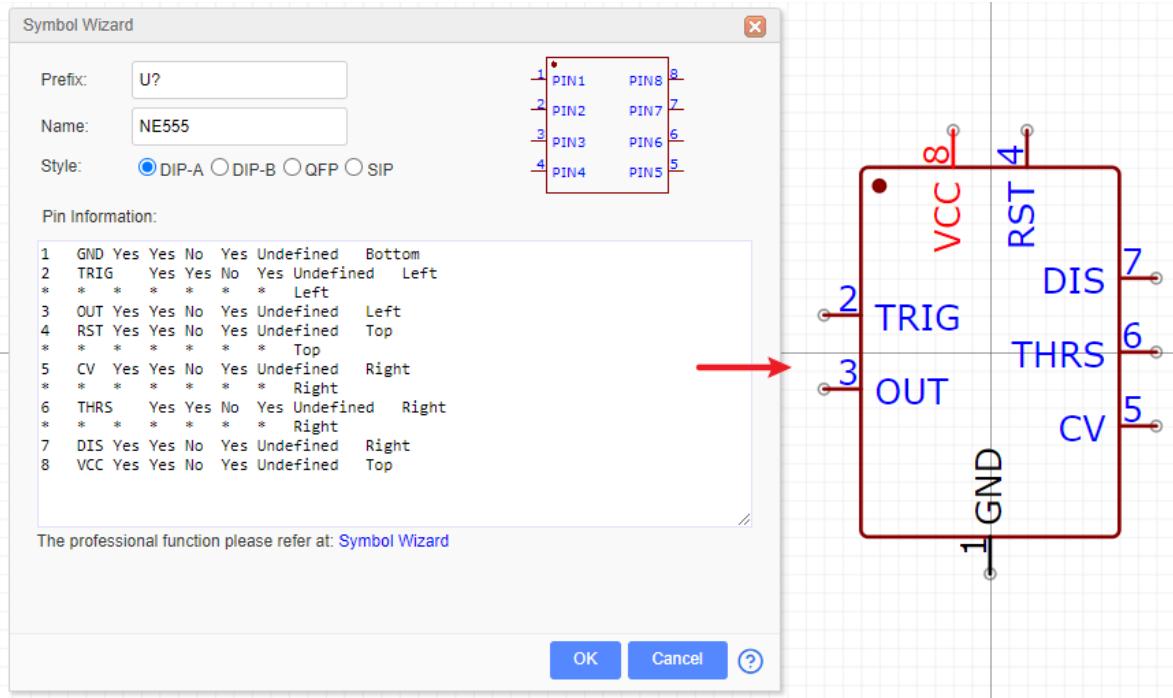
1. Download [Schematic Symbol Wizard Template.xlsx](#)

2. Open it via Excel or WPS, and edit each Pins attributes and position, and then copy the content and paste in wizard dialog without content title.

Tip: If you want to create the gap between Pin and Pin, you can use the as below image.

|    | A      | B    | C              | D            | E     | F    | G         | H        | I |
|----|--------|------|----------------|--------------|-------|------|-----------|----------|---|
| 2  | Number | Name | Number Display | Name Display | Clock | Show | Electric  | Position |   |
| 3  | 1      | GND  | Yes            | Yes          | No    | Yes  | Undefined | Bottom   |   |
| 4  | 2      | TRIG | Yes            | Yes          | No    | Yes  | Undefined | Left     |   |
| 5  | *      | *    | *              | *            | *     | *    | *         | Left     |   |
| 6  | 3      | OUT  | Yes            | Yes          | No    | Yes  | Undefined | Left     |   |
| 7  | 4      | RST  | Yes            | Yes          | No    | Yes  | Undefined | Top      |   |
| 8  | *      | *    | *              | *            | *     | *    | *         | Top      |   |
| 9  | 5      | CV   | Yes            | Yes          | No    | Yes  | Undefined | Right    |   |
| 10 | *      | *    | *              | *            | *     | *    | *         | Right    |   |
| 11 | 6      | THRS | Yes            | Yes          | No    | Yes  | Undefined | Right    |   |
| 12 | *      | *    | *              | *            | *     | *    | *         | Right    |   |
| 13 | 7      | DIS  | Yes            | Yes          | No    | Yes  | Undefined | Right    |   |
| 14 | 8      | VCC  | Yes            | Yes          | No    | Yes  | Undefined | Top      |   |
| 15 |        |      |                |              |       |      |           |          |   |
| 16 |        |      |                |              |       |      |           |          |   |

3.The Wizard will create the symbol follow your content. The types you chosen will be ignored.



#### Notice:

- If the content you input wasn't one, two or eight columns, it will shown incorrect format.
- You can use the Key Space to separate the column data.

## Edit Exited Schematic Symbol

### Personal Libraries

When you **CTRL+S** to save the Schematic Symbol, will pop up a dialog, you can choose this library's owner:

Save as Symbol

|                                                                             |                                                                                                     |                             |
|-----------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------|-----------------------------|
| Owner:                                                                      | UserSupport                                                                                         | <a href="#">Create Team</a> |
| Title:                                                                      | C_0603_US                                                                                           |                             |
| Supplier:                                                                   | Unknow                                                                                              | Or Others                   |
| Supplier Part:                                                              | 296-6501-2-ND                                                                                       |                             |
| Manufacturer:                                                               | ReliaPro                                                                                            |                             |
| Manufacturer Part:                                                          | NE555DR                                                                                             |                             |
| Link:                                                                       | <a href="http://www.ti.com/lit/ds/symlink/ne555.pdf">http://www.ti.com/lit/ds/symlink/ne555.pdf</a> |                             |
| Tags:                                                                       | Split by ":" for multi tags                                                                         |                             |
| Description:                                                                |                                                                                                     |                             |
| <input type="button" value="✓ Save"/> <input type="button" value="Cancel"/> |                                                                                                     |                             |

After finish, you can find your library at the left panel: **Library > Symbols > Work Space > All**

The screenshot shows the EasyEDA Library interface. On the left, there's a sidebar with icons for EELib, LCSC Parts, and JLCPCB. The main area has tabs for Library, Symbols, and Work Space. A search bar at the top right contains the text "Search symbol, footprint etc." Below it, a search engine dropdown is set to "EasyEDA" and a supplier dropdown is set to "LCSC Electronics". The "Symbol" tab is selected. The search results table has columns for "Title(PartNO)" and "Footprint". The results listed are:

| Title(PartNO)  | Footprint         |
|----------------|-------------------|
| NCP1117ST18T3G | SOT230P700X180-4N |
| NE555          | SOP-8             |
| 2 HEADER COPY  | SIP220P-2         |
| R_1812_US      | R1812             |
| ESP12E_DEVKIT  | ESP12E_DEVKIT     |

A red arrow points to the "Work Space" tab in the top navigation bar.

## Tag

When you select it , right-click it and select the menu "modify", you can add a tag for it.

| Title(PartNO)  | Footprint         | Owner       | Description |  |
|----------------|-------------------|-------------|-------------|--|
| NCP1117ST18T3G | SOT230P700X180-4N | UserSupport |             |  |
| NE555          | SOP-8             | UserSupport |             |  |
| 2 HEADER COPY  | SIP220P-2         | UserSupport |             |  |
| R_1812_US      | R1812             | UserSupport | aaaaaa      |  |
| ESP12E_DEVKIT  | ESP12E_DEVKIT     | UserSupport |             |  |

> All > NCP1117ST18T3G

Edit Place More ▾ X Cancel

Edit Modify Delete Clone Add Sub Part

## Favorite

When you favorite a library, you can find it at **Library > Symbols > Work Space > Favorite**, If this library has a tag, the tag will show up too, but you can't edit that. But you can via "Clone" or "Edit and save" to create a new library to personal libraries.

| Title(PartNO)  | Footprint   |
|----------------|-------------|
| NCP1117ST18T3G | P700X180-4N |
| NE555          | -2          |
| 2 HEADER COPY  |             |
| R_1812_US      |             |
| ESP12E_DEVKIT  | DEVKIT      |

Edit Modify Delete Clone Add Sub Part Add Favorite Refresh View Datasheet... Report Error... View Owner View Detail

## Edit Symbol in the Library

When you feel the Schematic Libs can not be satisfied for you, you can edit it.

Via "**Library**" > "**Search Part/Work Space/LCSC/System**" > **Select Symbol** > **Edit**

or you can click the preview image

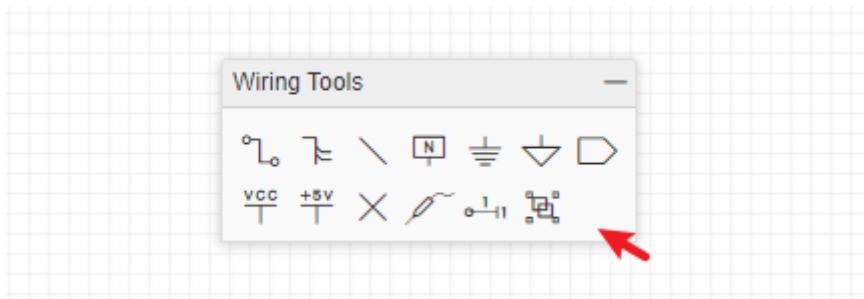
The screenshot shows the EasyEDA Library search interface. The search bar at the top contains the text "amp". Below it, the "Types" dropdown is set to "Symbol". The "Classes" dropdown is set to "Work Space". The search results table has columns for "Title(PartNO)", "Footprint", "Owner", and "Description". The first result, "NCP1117ST18T3G", is highlighted. To the right of the table, there is a detailed view of the component's symbol and pinout. The bottom of the interface shows a breadcrumb navigation path: "EasyEDA > Symbol > Work Space > All > NCP1117ST18T3G". Below the table are buttons for "Edit", "Place", "More", and "Cancel".

when you finish and save, it will be saved to your personal libraries **Work Space** and become your personal libraries.

## Edit Symbol in the Schematic

If you want to edit a symbol in the schematic, you can use the Ungroup/Group function.

On the **Wiring Tools** palette there is the **Group/Ungroup Symbol...** button.



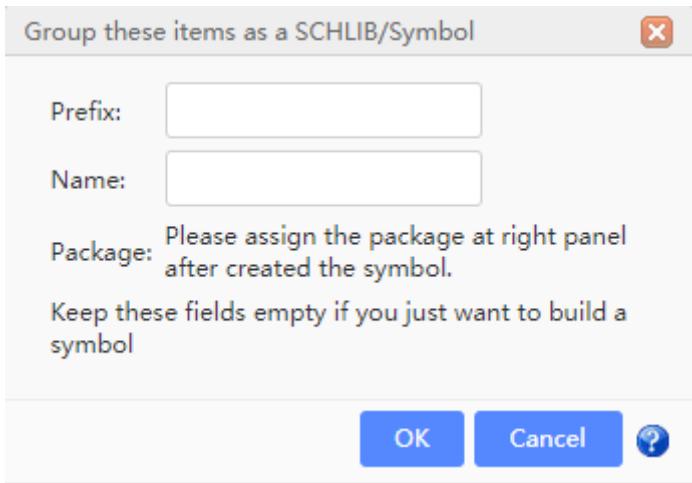
This tool is for you to quickly create or edit schematic library symbols.

1. Select a symbol
2. Click the **Group/Ungroup Symbol...** button

Up to this point you have a collection of separate pins, a drawn rectangle and some text that are all separate items with no particular association with each other.

3. Edit the shape or pin what you want to change
4. Select all of the items and click the **Group/Ungroup Symbol...** button.

A dialog will be opened:



After you click OK, all those separate elements will be grouped together to form your new symbol directly in the schematic.

Using the group function, you can create/edit any symbol in the schematic, easily and quickly.

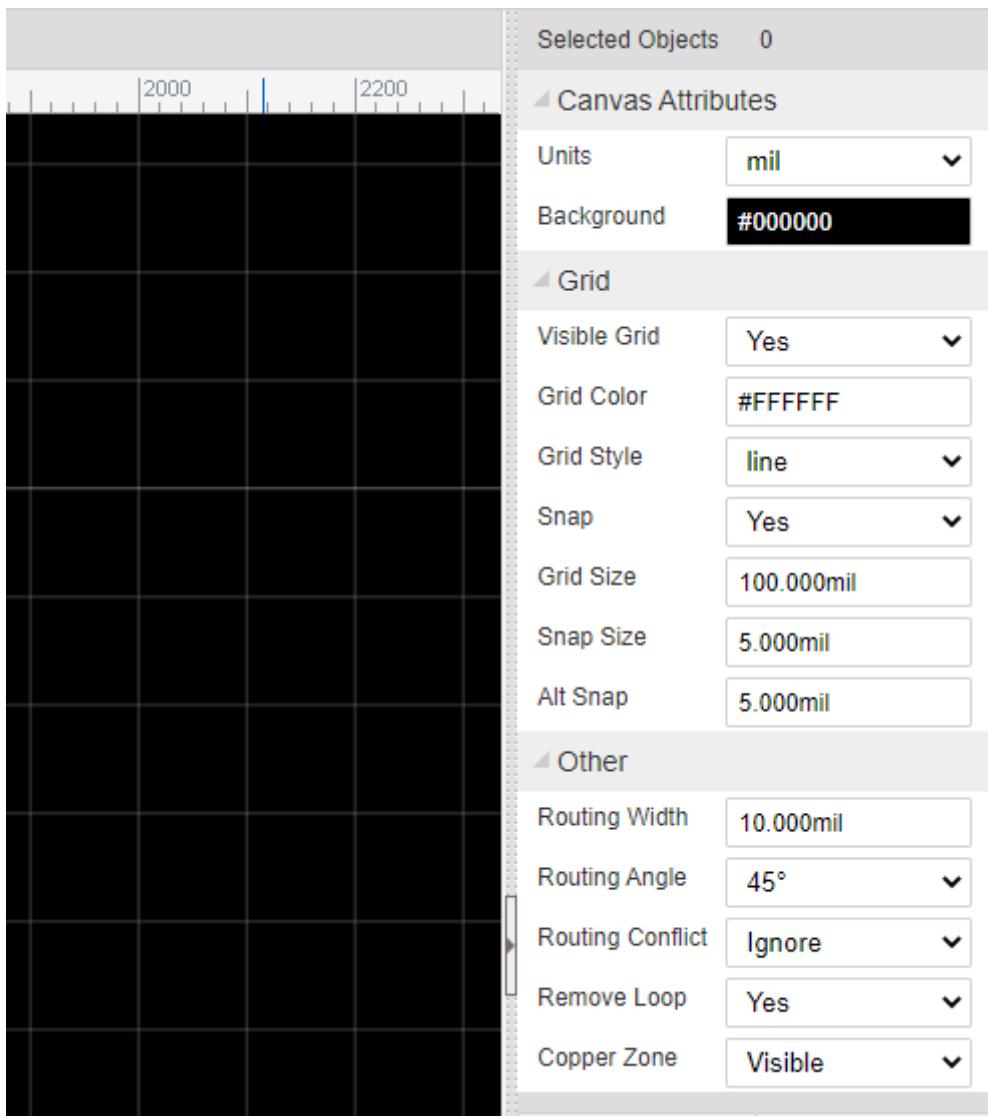
## Canvas Setting

---

After the initial conversion of a schematic to PCB, it is time to learn how to manage EasyEDA's PCB Design Editor.

### Canvas Attributes

Lots of PCB canvas attributes are the same as Schematic canvas attributes. The key is that you can set **units** in PCB canvas attributes.



When you select a object at the canvas, you can modify its attributes at the right panel.

**Snap Size:** The cursor snapping size.

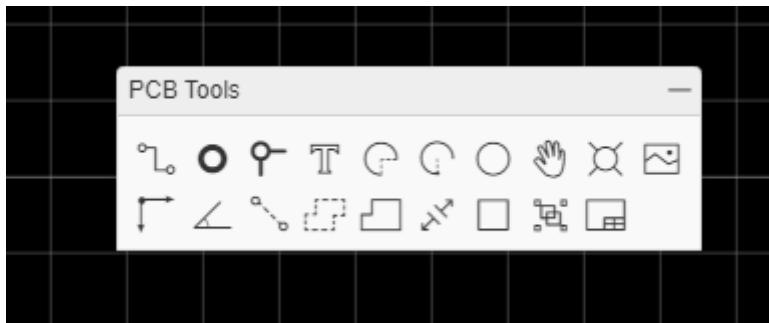
**Alt Snap:** When press hotkey ALT the cursor snapping size.

#### Other

- **Routing Width:** Setting the default routing width.
- **Routing Angle:** Setting the routing angle.
- **Routing Conflict:** When routing the track, what to do when impact the difference net objects.
  - **Ignore:** The track go through the objects.
  - **Block:** The track will stop when meet the difference net objects.
  - **RoundTrack:** The track will go aroud the difference net objects.
- **Remove Loop:** Remove the track loop.
- **Copper Zone:** Setting the copper zone visible or invisible.

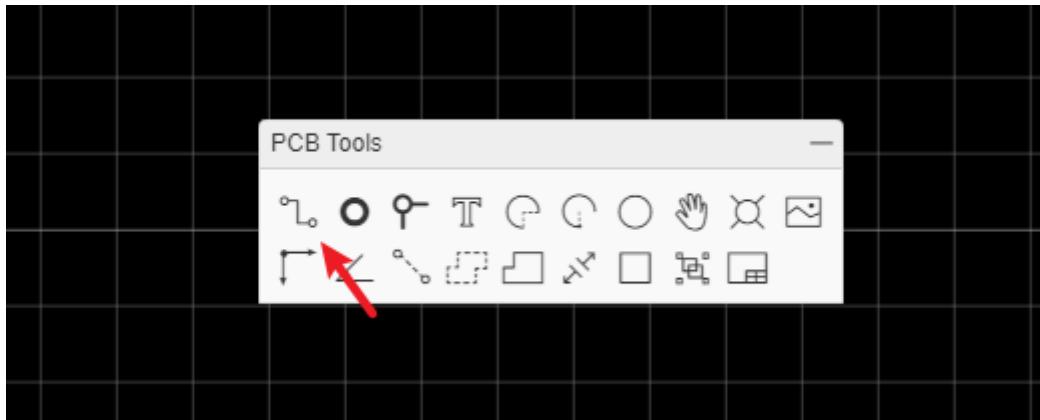
## PCB Tools

PCB tools provide many function to fulfill your PCB design requirement. Such as: Track, Pad, Via, Text, Arc, Circle, Move, Hole, Image, Canvas Origin, Connect Pad to Pad, Copper Area, Solid Region, Measure/Dimension, Rect, Group/Ungroup. etc.

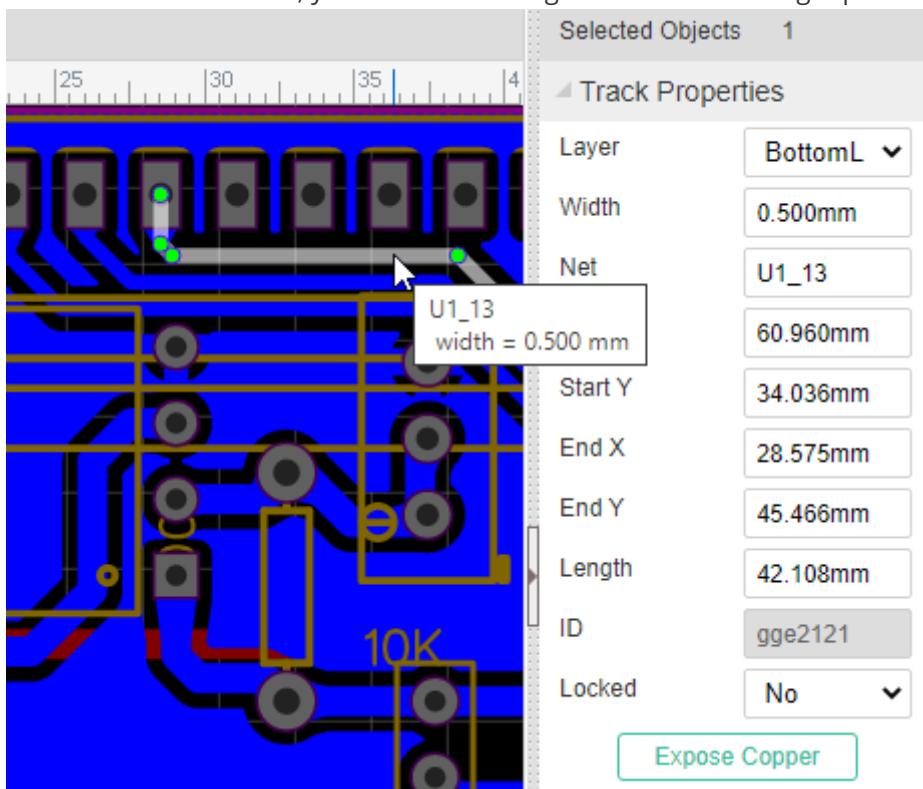


## Track

In the schematic editor, we use Wire or the **W** Hotkey to connect Pins, in a similar way in the PCB editor, we use Track to connect Pads. Track allows you to draw PCB tracks and can be found on the PCB Tools palette or using the **W** Hotkey (not T: see above!).



When a track is selected, you can find its Length attribute in the right panel.

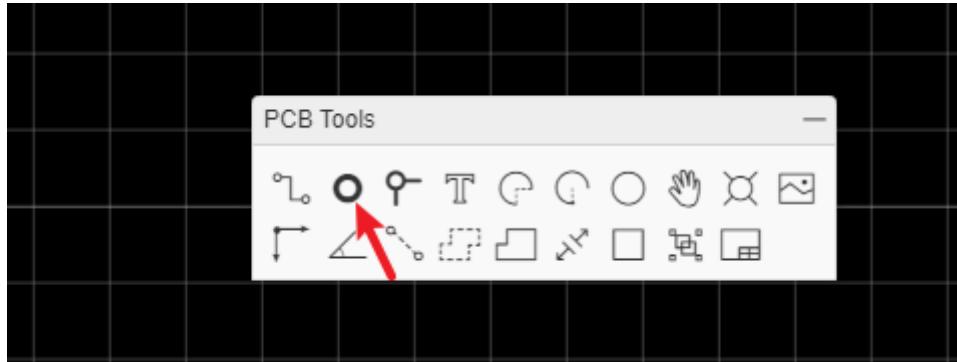


If you want to create solder mask for the track, you can click the "Expose Copper" button at the right-hand property panel.

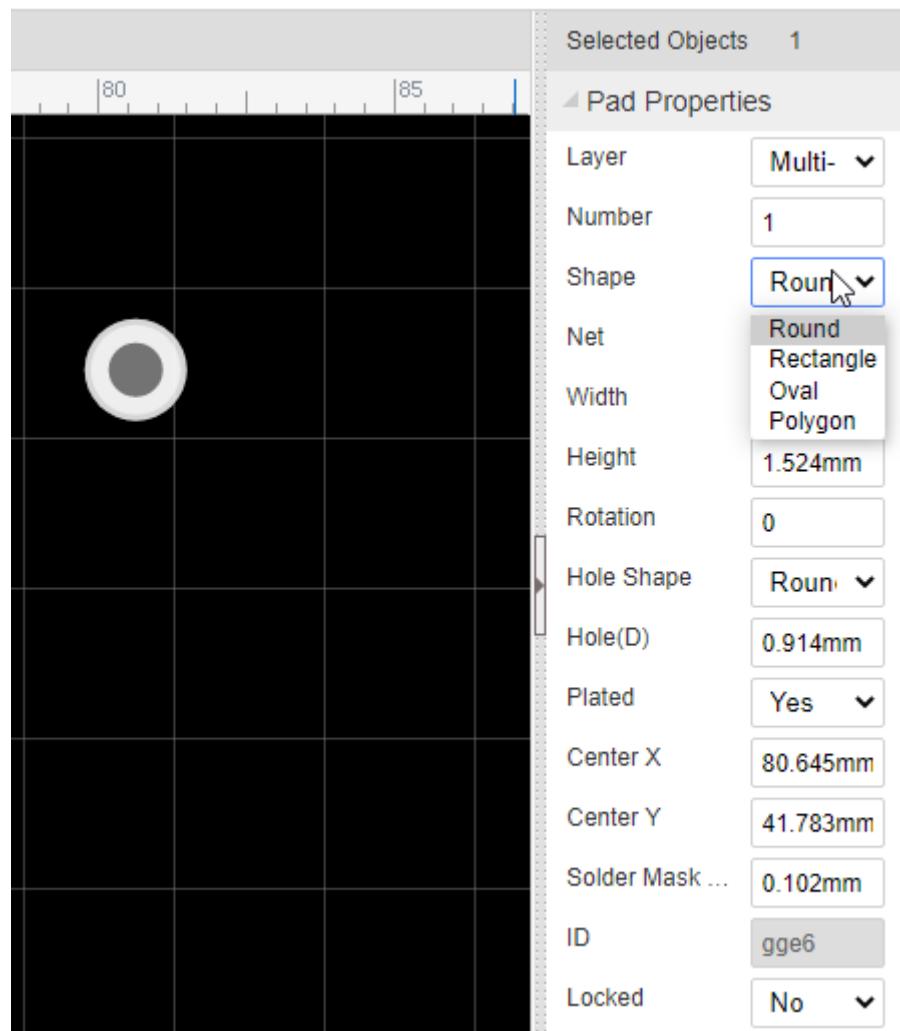
The more information of routing, please refer at [PCB: Route Tracks](#)

## Pad

You can add pads using the Pads button from the Footprint Tools palette or using the **P** hotkey.



After selecting one of the pads, you can view and adjust its attributes in the right hand Properties panel.



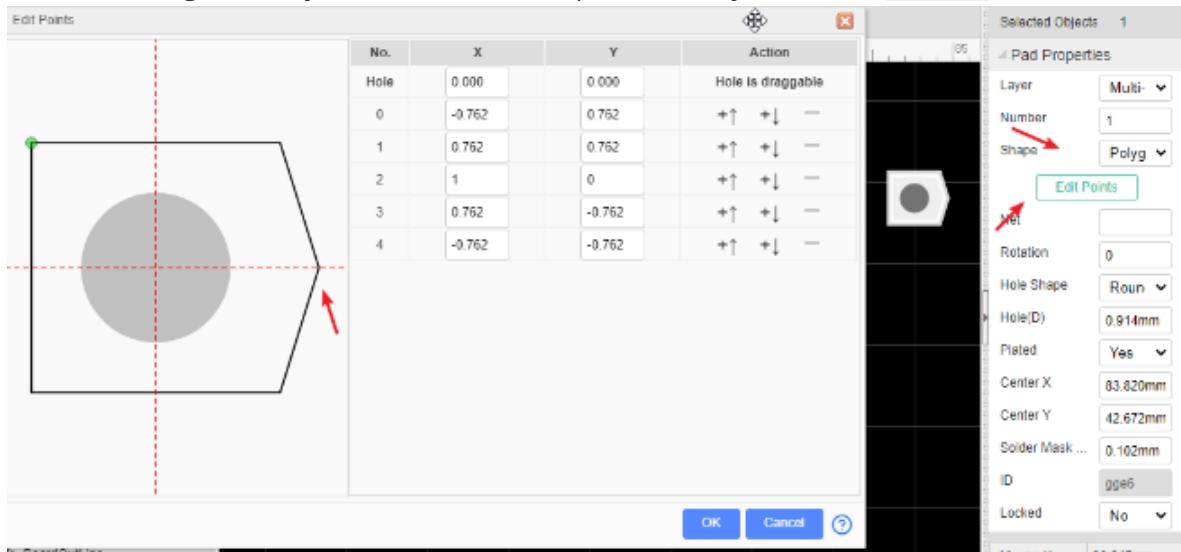
**Number:** Remembering the pin numbers you set in the schematic symbol in your Schematic Lib: to connect those schematic symbol pins to the pads in your PCB footprint, the pad numbers you set here in the Footprint footprint must be the same.

**Shape:** Round , Rectangular , Oval and Polygon.

EasyEDA supports four shapes: Round , Rectangular , OVAL and POLYGON .

- OVAL PAD will give you more space.
- POLYGON PAD will let you to create some strange pad.

Like in the image below, you can edit the PADs points when you select a POLYGON PAD



**Layer:** If the pads are part of a **SMD** footprint, you can set it to **Top layer** or **Bottom layer**. For through hole components you should set it to **Multi-Layer**. If it setting as mult-layer, it will connect with all copper layers.

**Net:** You don't need to enter anything here because at present this footprint is not connected to anything in a circuit.

**Width and Height:** When the shape is set to Round, Width will equal Height.

**Rotation:** Here you can set the Pad's rotation as you want.

**Hole(D):** This is the drill hole **diameter** for a through hole pad. For a SMD Pad, set its layer to **TopLayer or BottomLayer**.

**Hole Shape:** Round and Slot. When it is set as a slot, the Gerber is generated through the stitching of multiple drill holes in the corresponding position. If your hole is round, please do not set it as a slot, so as to avoid the overlapping error of holes during the production of DFM detection.

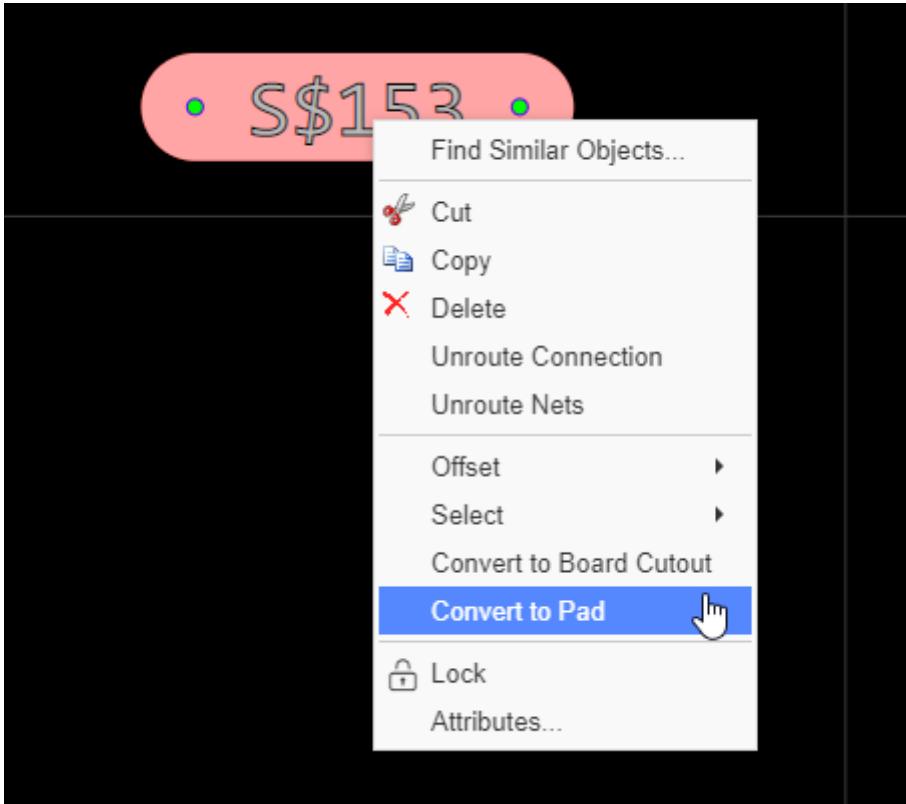
**Center-X and Center-Y:** using these two attributes, you can set the pad's position with more precision, compared to using the mouse.

**Plated:** Yes or No. When you set it as No, this pad Inner wall do not metallization.

**Paste Mask Expansion:** For single layer pad. This property affects the size of the tin area on the plate of the steel mesh. If you want to set a pad that is not open in the steel mesh, you can set the value to be negative, which is usually larger than the diagonal of the pad.

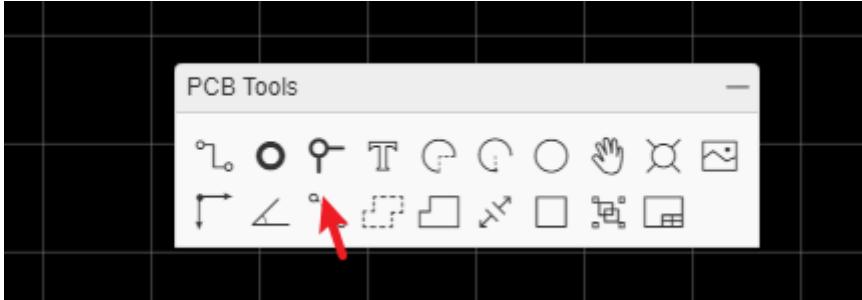
**Solder Mask Expansion:** This property affects the size of the green oil area cover on the pad. If you want to set a pad not open covered with green oil, you can set the value to be negative, the value is usually set larger than the diagonal of the pad.

And you can select a track/Solid Region, right-click it and convert to a pad.



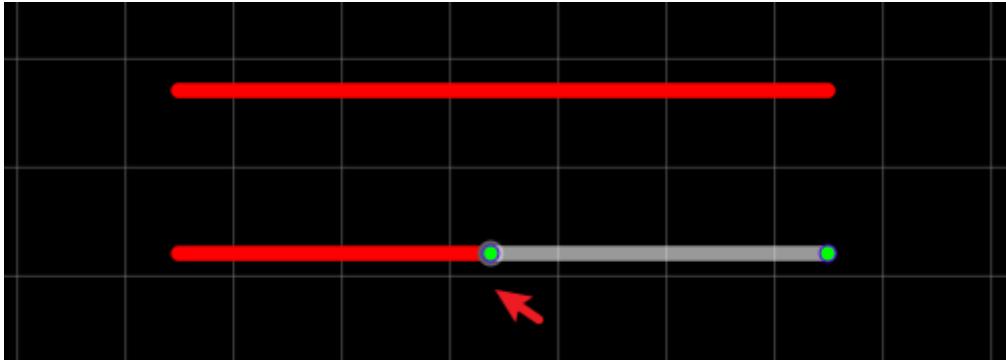
## Via

When you want to lay a multilayer PCB, you need to add Vias for nets getting through layer and layer.



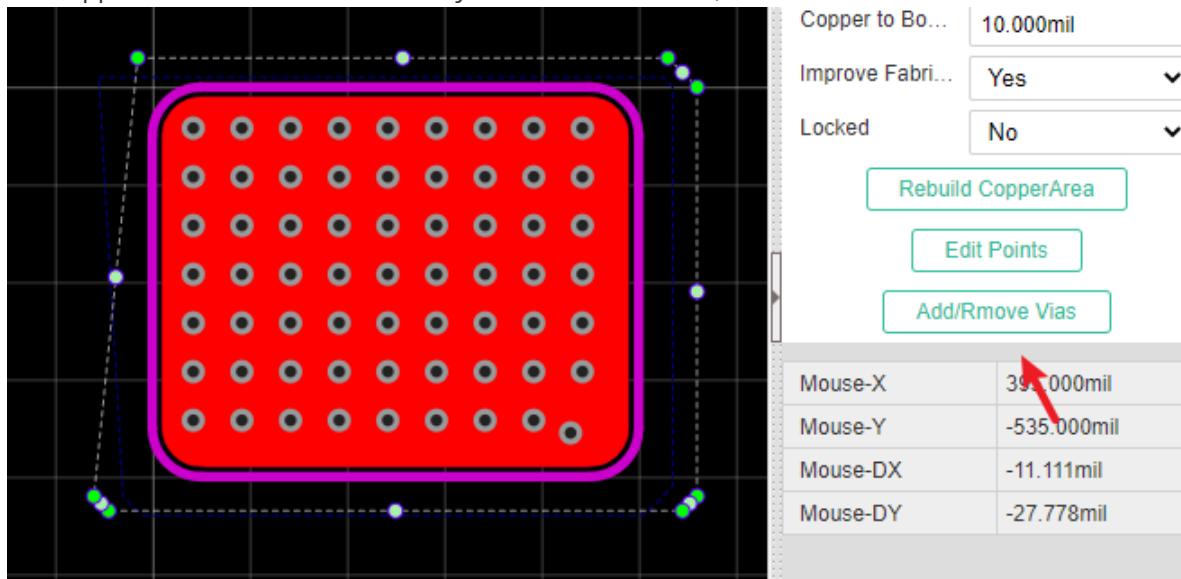
### Place a Via on a Track

When placing a [via](#) on a track, the track will be cut to two segments, and the via net will follow track's net. Placing two vias on a tracks, you will get three segments, then you can change one segment to other layer id, or remove one of them.



### Place Multiple Vias

Click the copper area outline, click the "Add/Remove Vias" button. this feature needs the same net copper areas on two and more layers in the same time, the cross area will add the vias.

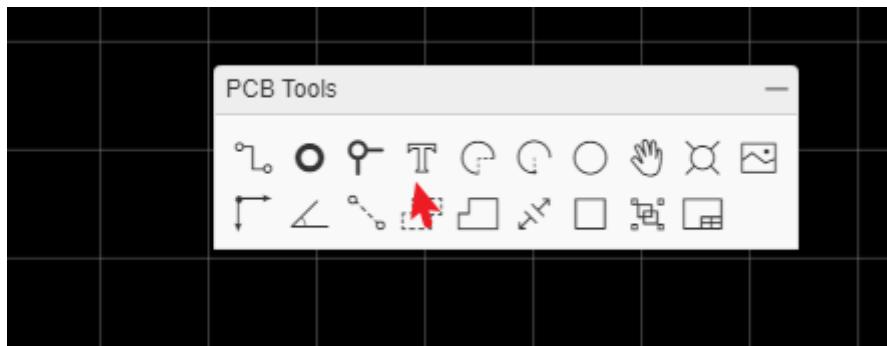


#### Notice:

- EasyEDA only support the through via for all layers, doesn't support the Buried Blind/via.

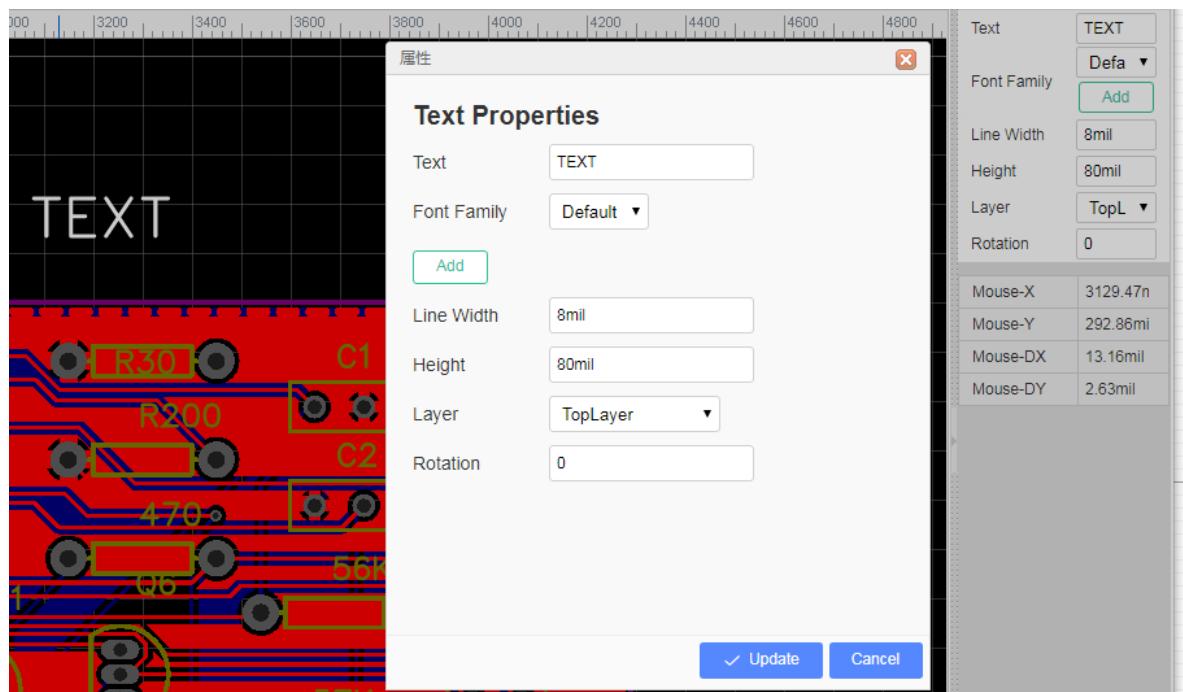
## Text

You can add more fonts from your computer or download some [free fonts](#):[www.1001freefonts.com](http://www.1001freefonts.com) and [free fonts: http://www.fontspace.com/](http://www.fontspace.com/).

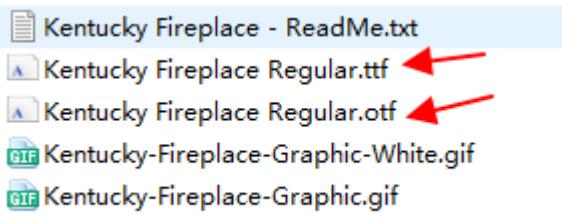


if you need Japanese or Korean you can use [Google Noto fonts](#)

Select the text, then you can find a Font-family attribute on the right panel like in the image below.



Click the add button, then choose the font, the font file must be `ttf` or `otf`.

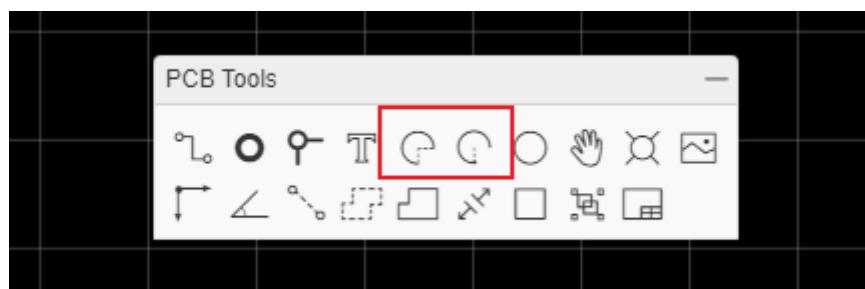


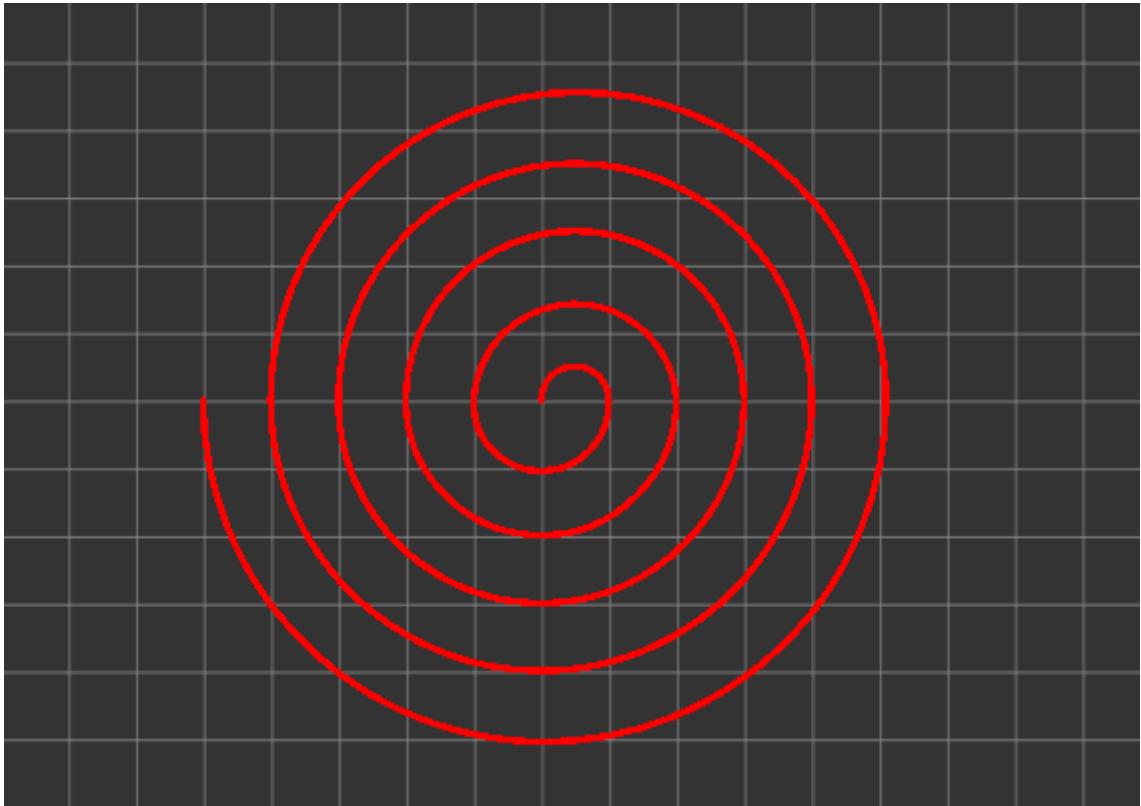
So you can add any fonts by yourself. EasyEDA doesn't cache the font on our server, so if you close the editor, you need to add the font again by yourself.

**Note:** If you use the other font, the `Linewidth` attribute is useless, because it will be automatically set by changing the `Height`.

## Arc

You can draw many Arcs with different sizes, it's easy to create a pretty cool PCB as you like.





EasyEDA provides two Arc tools:

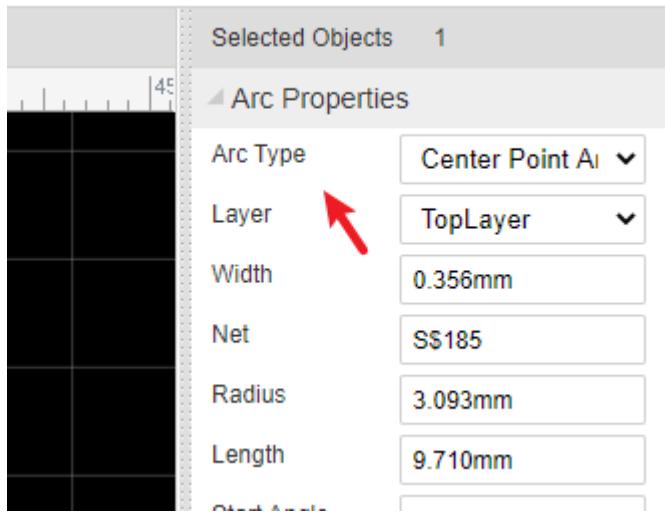
- Start point fixed, you can change the end point position and radius.



- Center point fixed, you can change the radius.

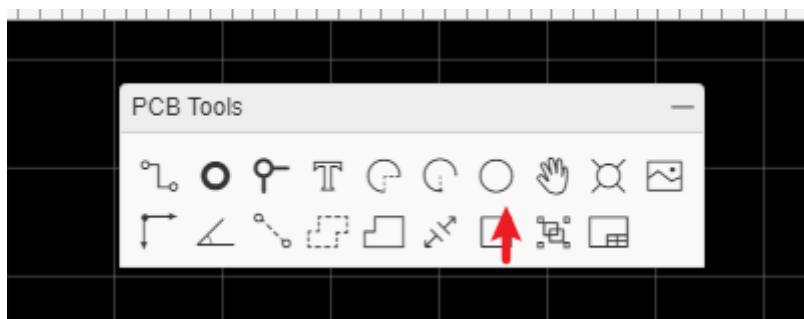


Select the arc, you can change the arc type at property panel, different arc type has different drag behavior.



## Circle

You can draw a circle in PCB. If you want to draw a circle at TopLayer or BottomLayer, please use Arc.



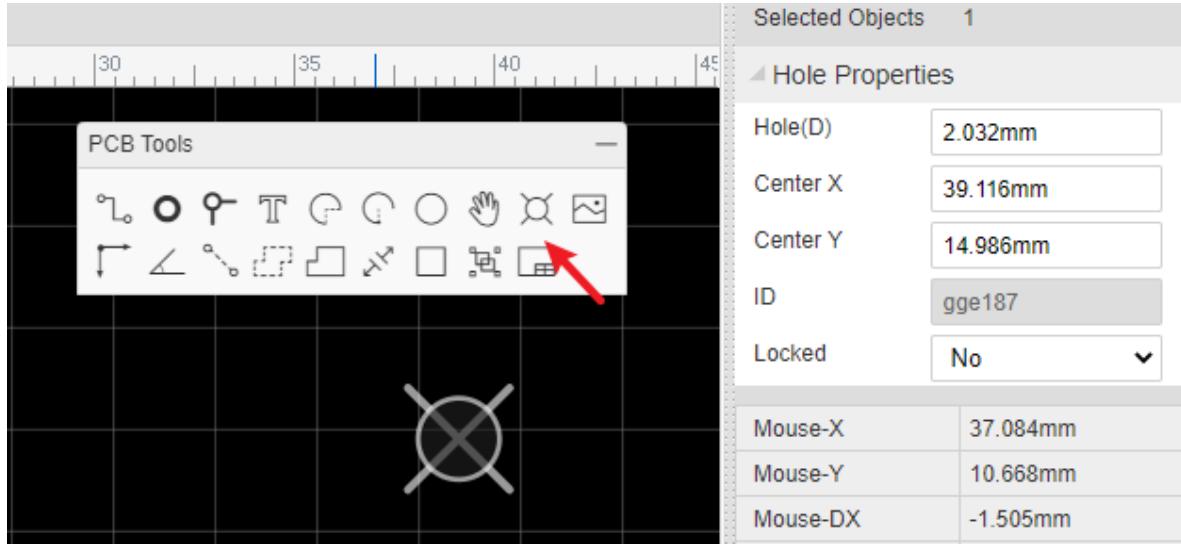
## Move

This option is same as schematic's drag.



## Hole

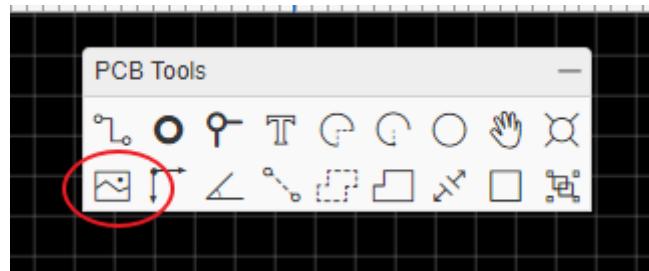
There were lots of users that didn't know how to use PAD or VIA as a HOLE, they asked EasyEDA for help, so EasyEDA added a HOLE TOOL in the PCB toolbar.



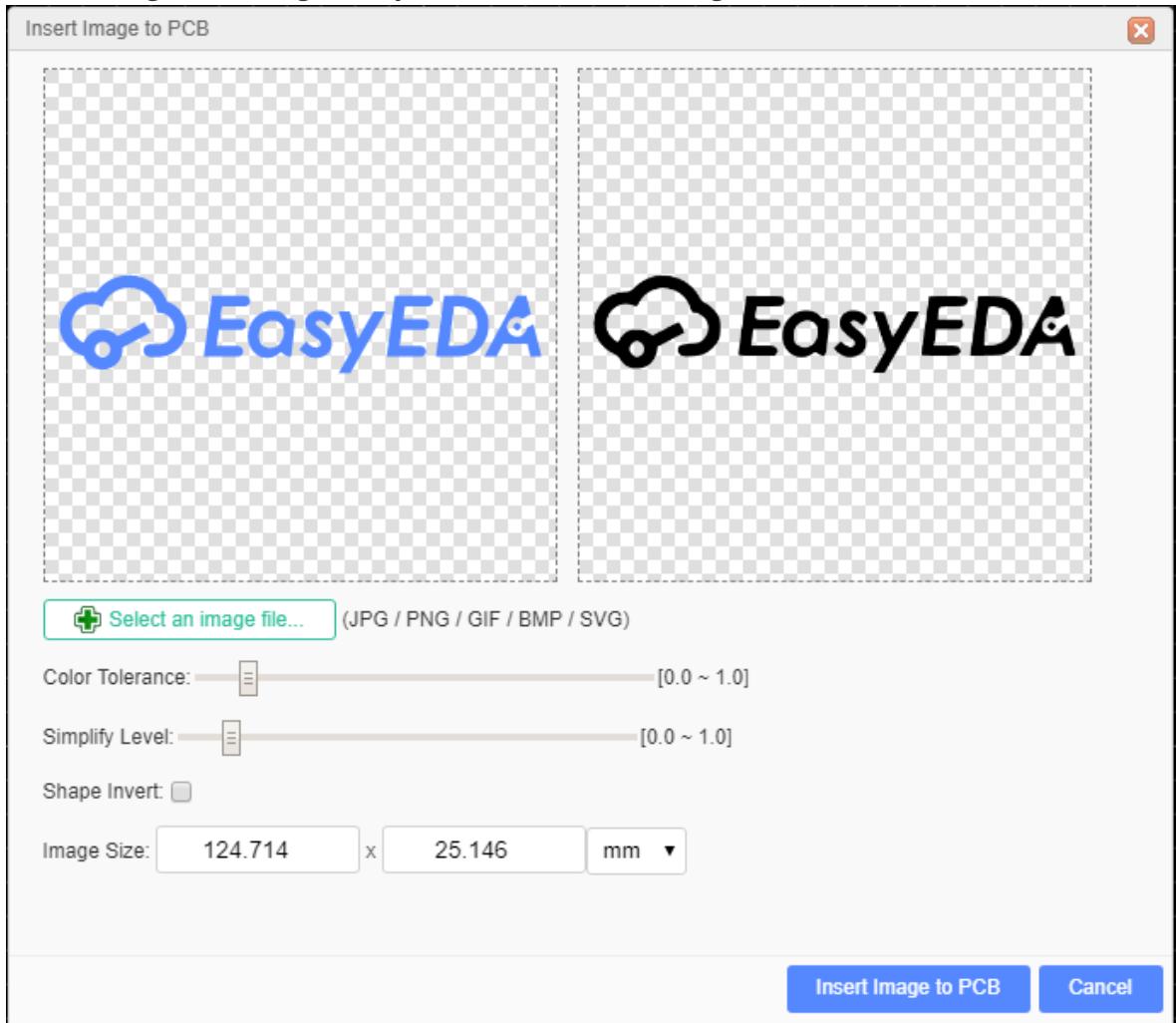
And if you want to create the slot hole, you can use solid region(Type: NPTH), or route a track, and then right-click the "Convert to NPTH" menu.

## Image

On PCB and Footprint editor, there is a nice feature on the PCB Tools bar.



After clicking on the image icon, you will see the Insert Image window as below.

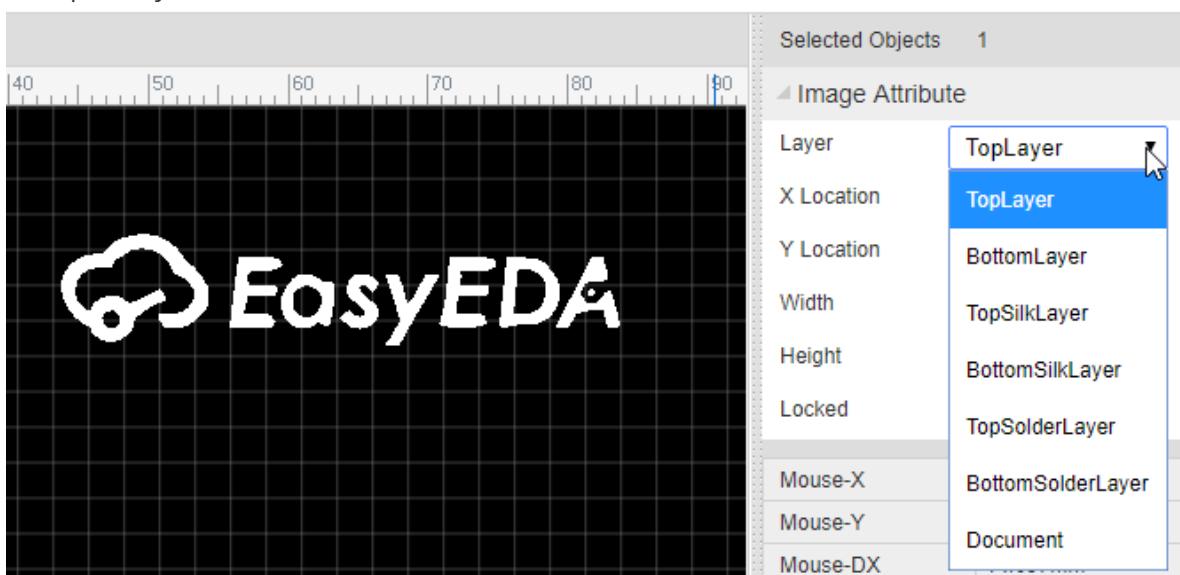


In this dialog, you can choose your favorite image, EasyEDA support [JPG](#), [BMP](#), [PNG](#), [GIF](#), and [SVG](#). Unlike some other EDA tools which only support a Monochrome Bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

You can adjust the color tolerance, simplify level and reset the image size there.

And you can select shape invert.

The image will be inserted to the active layer, if it is not right, you can change the attribute. Such as TopSilkLayer.



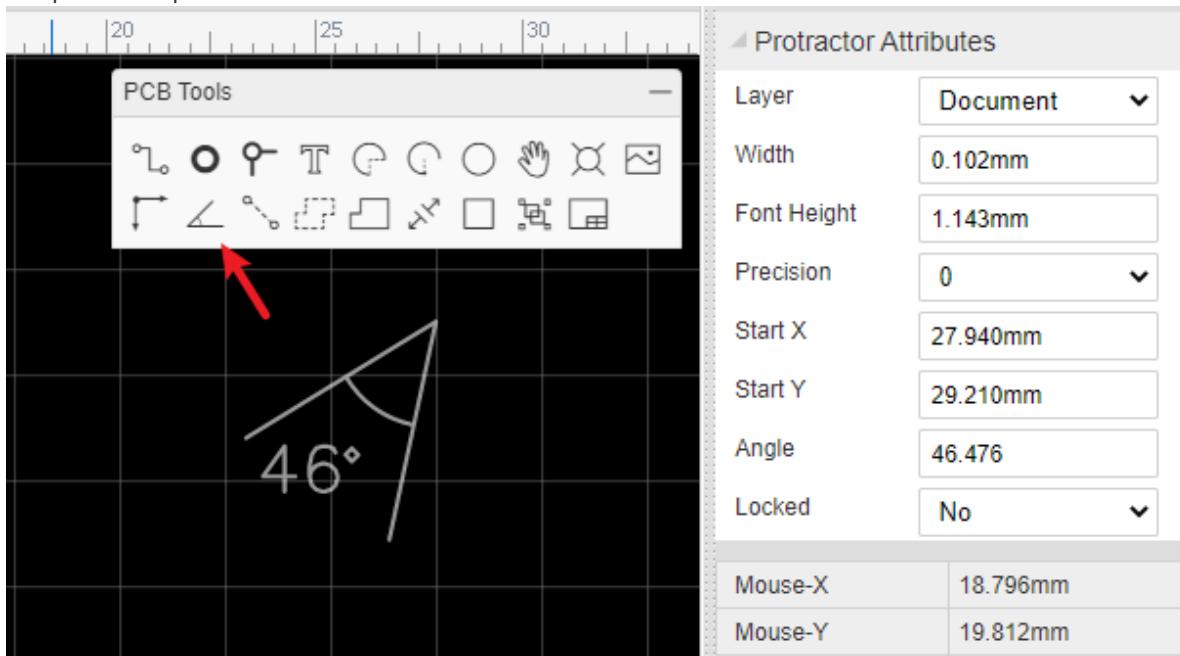
# Canvas Origin

This option is the same as schematic's Canvas Origin.



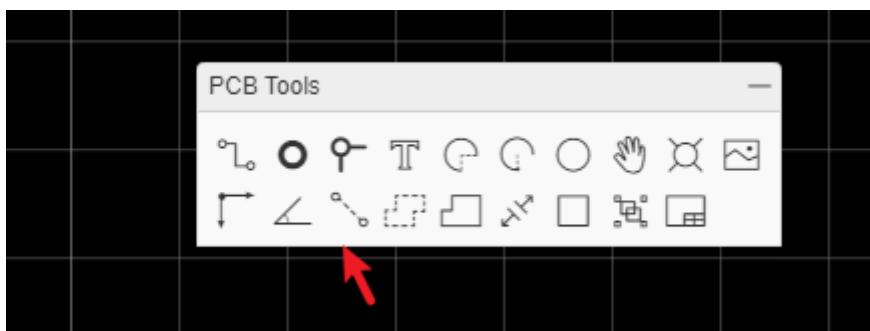
## Protractor

We provide a protractor for PCB tools.



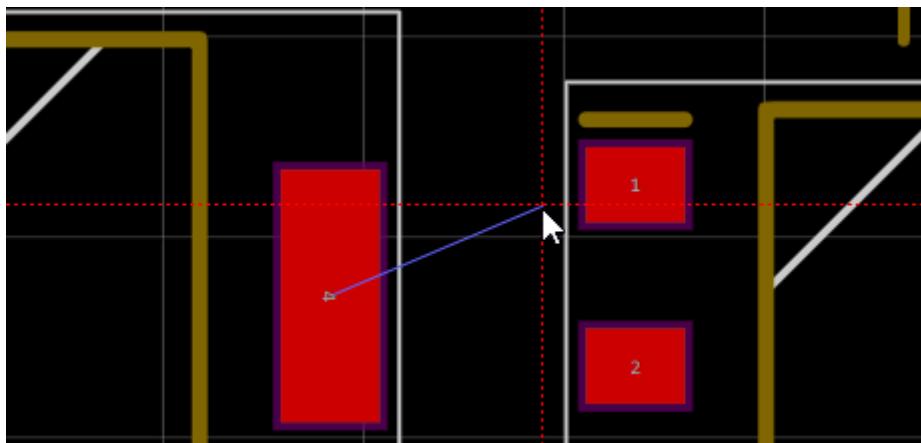
## Connect Pad to Pad

When creating a PCB without a Schematic, none of the pads on the Footprints have nets connecting them so there will be no ratlines.



Rather than try to track the pads from scratch, it is a good idea to connect them up by hand first using `Connect Pad to Pad` from the PCB Tools palette. This will help you to remember to track the pads correctly with fewer mistakes.

You could also do this by setting net names for all the pads: if the two pads are given the same net name then EasyEDA will understand that they are connected together and will automatically create a ratline between them.



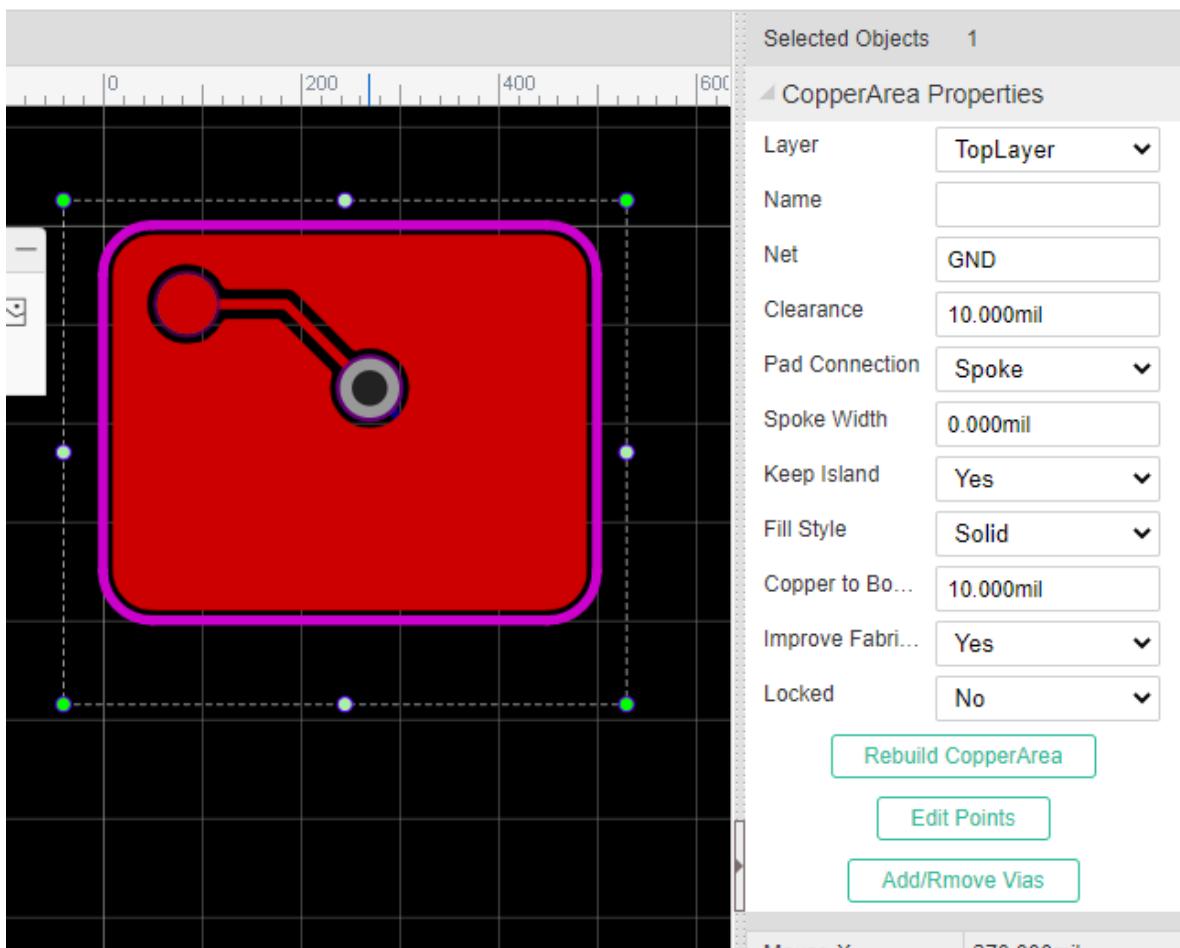
Or you can set these two pads with the same net name at the right panel Pad Properties after you click the pad.

## Copper Area

Sometimes you will want to fill in or flood an area with copper. Usually this copper area will be connected to a net such as **GND** or a supply rail. You can draw the outline of a flood using the **Copper Area** button from the PCB Tools palette.



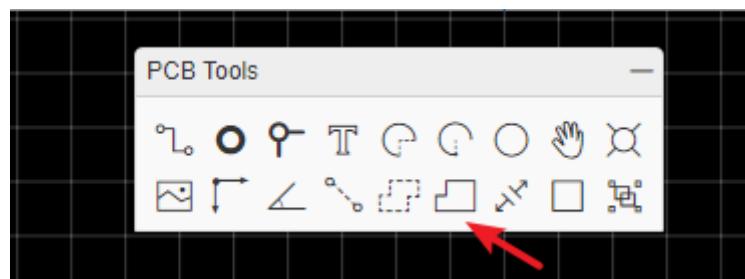
When selecting a copper area, you can find its attributes from the right hand **Properties** panels.



The more information please refer at [PCB: Copper Pour](#)

## Solid Region

EasyEDA has added a new tool Solid Region for PCB design



This is a very useful, quick way to connect Pads. You can draw a Solid Region to include all of these pads with same net name, then set the region to the same net name as the pads. It is like Copper Area but easier to use for small areas. To use Solid Region like this, set the Type attribute (in the right hand Properties panel) to Solid.



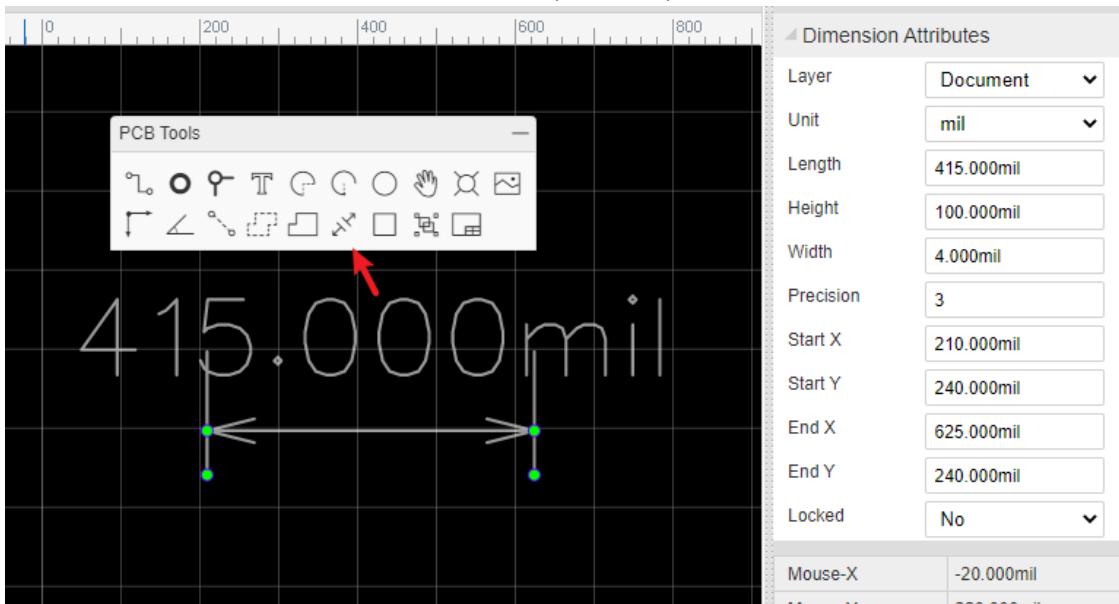
The more information please refer at [PCB: Solid Region](#)

## Measure/Dimension

Making and adding measurements is useful in PCB design. EasyEDA provides two methods to do this.

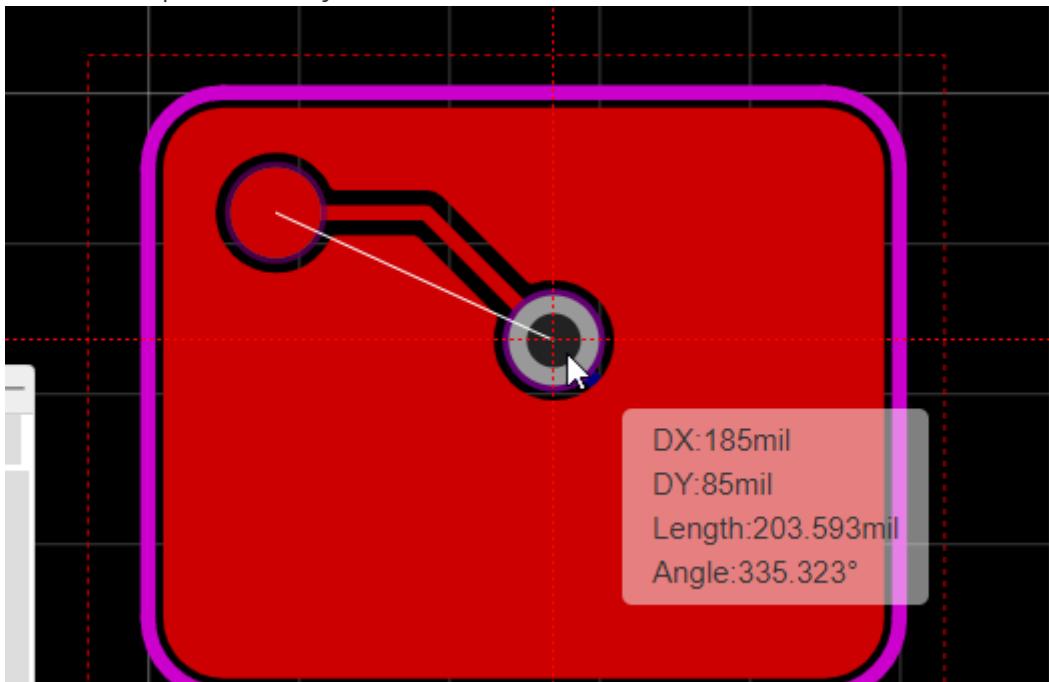
### 1. Dimension tool in the PCB Tools palette:

This tool can show three units on the canvas, milliliter, inch and millimeter.



When you click one side of the dimension on the PCB, you can drag it for any directions or change its length.

### 2. Measure a distance using Hotkey M, Or Via: Top Menu > Edit > Measure Distance, then click the two points which you would like to measure.

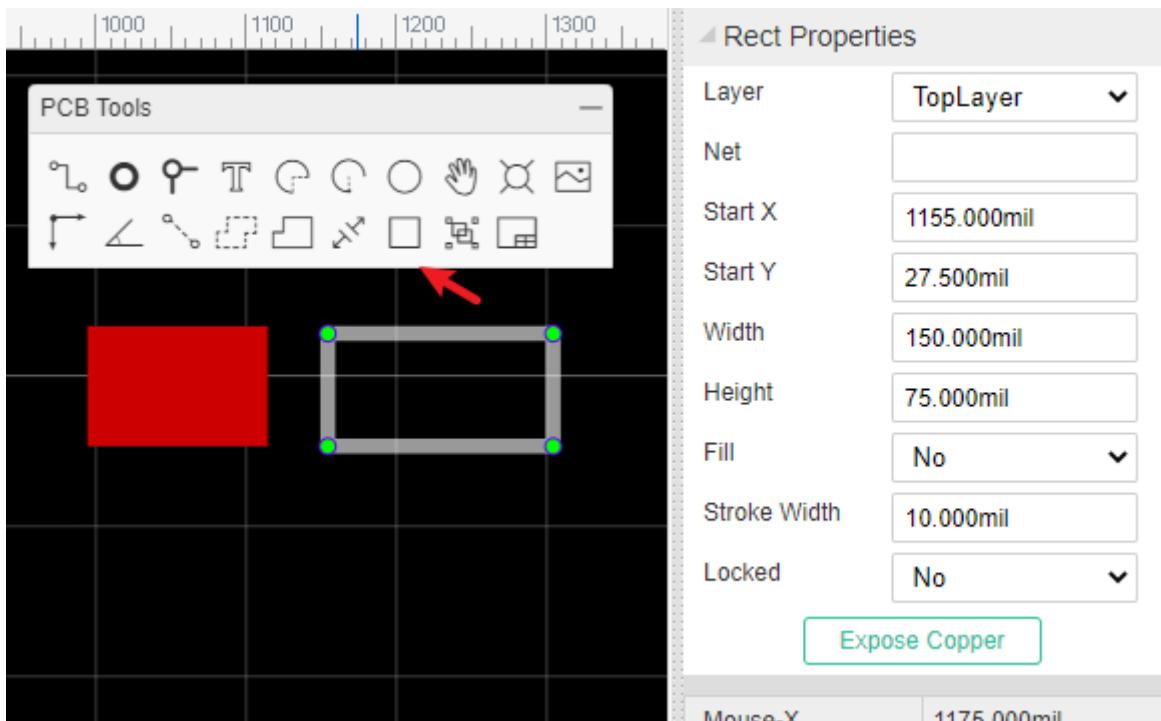


#### Tips:

- It's unit follows canvas's units.
- You can disable the snap option to measure at the canvas property panel.

## Rect

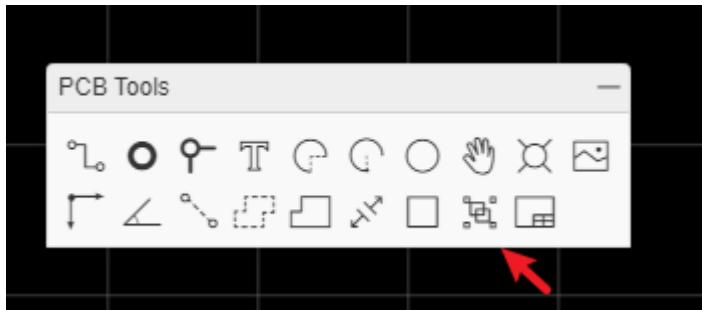
It looks like a Solid Region, but it can't be set Nets and you can't set the Layer as NTPH.



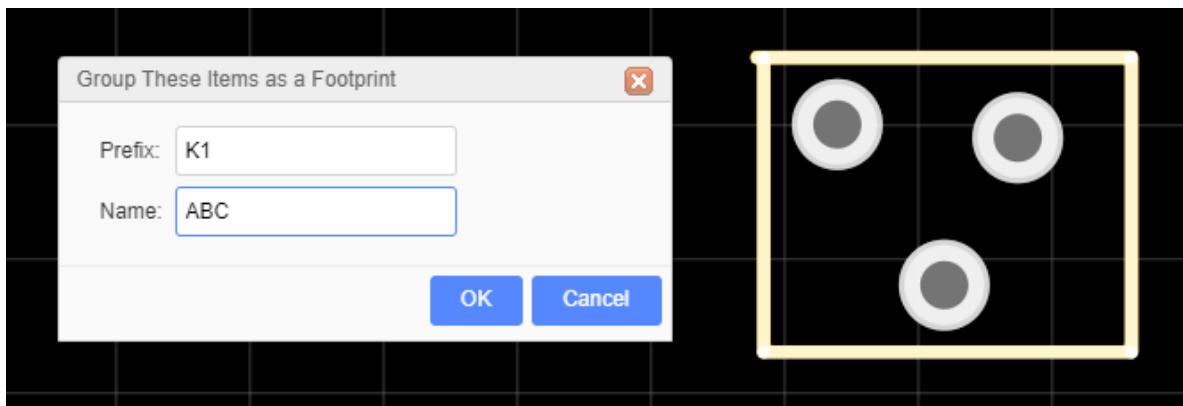
The rect doesn't rotate, you can change its width and height.

## Group/Ungroup

Just like Group/Ungroup in the Schematic Editor can be used to create a schematic lib symbol, you can use Group/Ungroup from the PCB Tools palette to create a Footprint footprint in the PCB editor.



For example, place Tracks and Pads on the canvas, then select all of them and click **Group/Ungroup** to group them like as a footprint in the image below:



Notice:

- Before ungroup the footprint, please change it's layer to top layer first, because of the footprint after grouping will at top layer.

- The grouped footprint doesn't support Import Changes, it will be removed if you Import Changes.

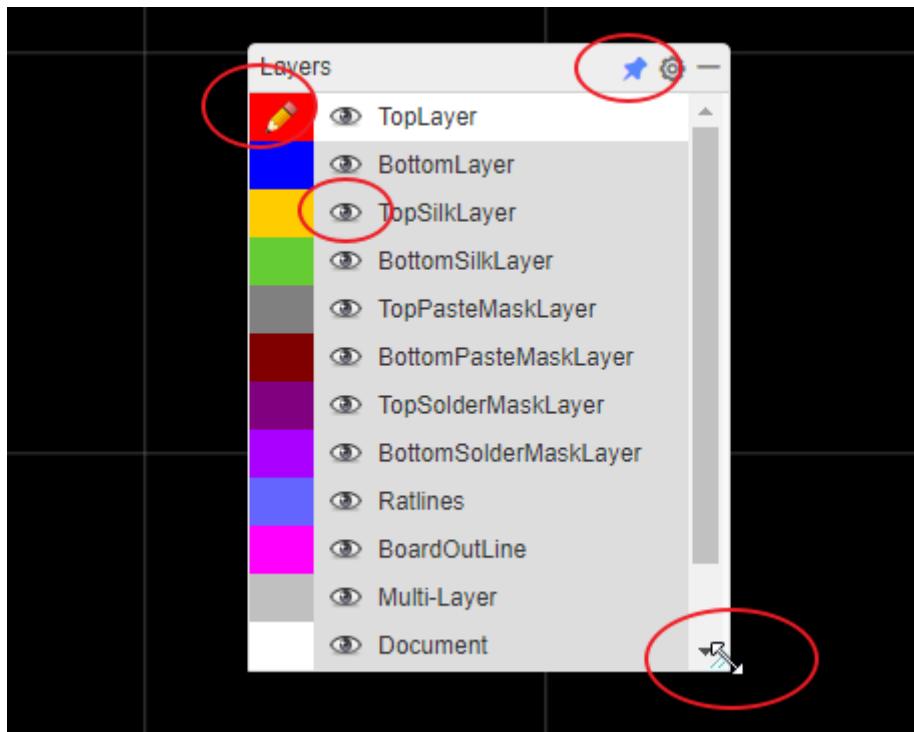
# Layers Tool and Objects

## Layers Tool

Active Layer: The colours of the layers in the **Layers Tool** are defined in the Layer Options Settings. To work on a layer then you must make it the Active layer.

To do this,

- Click on the eye icons to show/hide layers.
- The pencil icon in the coloured rectangle indicates that this is the active layer.
- Click the pin icon to fix the layering tool without automatically closing it.
- The height and width of the layer tool can be adjusted when dragging the lower right corner of the Layers Tool.



HotKeys for layer activation:

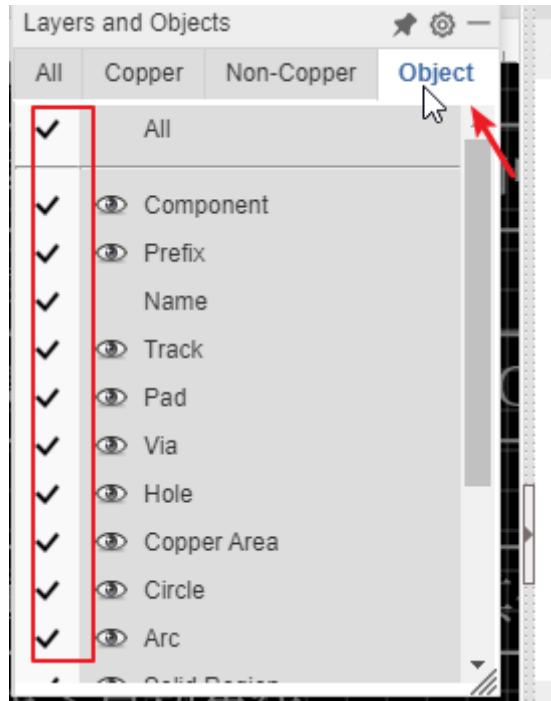
- T:** Top Layer is active
- B:** Bottom Layer
- 1:** Inner1 Layer
- 2:** Inner2 Layer
- 3:** Inner3 Layer
- 4:** Inner4 Layer

The more information for the PCB layers please refer at [PCB Layout - Layer Manager](#)

**note:** the hidden PCB layer is only visually hidden. The corresponding layer will still be exported during photo preview, 3D preview and Gerber export.

## Objects Filter Tool

Click "Object" to switch to object filtering.



**Select:** When the tick in front of the object is checked, the corresponding object in the canvas can be manipulated with the mouse. Uncheck will not allow mouse operation. Including click selection, box selection, drag and other operations.

**Eye:** Click eyes to modify the display and hiding of corresponding objects in batches.

- Component: Displays or hides the entire components, excluding the component's name and prefix
- Prefix: Displays or hides the entire components' prefix
- Name: Displays or hides the entire components' name
- Track: Displays or hides the entire tracks, for all layers
- Pad: Displays or hides the entire free pads, excluding the pads in the component
- Copper Area: Displays or hides the entire copper areas' fill area, excluding copper outline
- Text: Displays or hides the entire normal texts, excluding the text of the component

**Note:**

- The layer and object invisible and visible will not go into Undo and Redo.

# Layer Manager

## Layer Manager

You can set the PCB layer's parameters at the Layer Manager.

Via **Top Menu> Tools > Layer Manager...**, Or Click **Layers Tool** gear icon. Or right-click the canvas - Layer Manager menu.

The Layer Manager dialog:

Layer Manager

Copper Layer: 4

| No. | Display                             | Name                  | Type       | Color   | Transparency(%) |
|-----|-------------------------------------|-----------------------|------------|---------|-----------------|
| 1   | <input checked="" type="checkbox"/> | TopLayer              | Signal     | #FF0000 | 0               |
| 2   | <input checked="" type="checkbox"/> | Inner1                | Signal     | #800000 | 0               |
| 3   | <input checked="" type="checkbox"/> | Inner2                | Signal     | #008000 | 0               |
| 4   | <input checked="" type="checkbox"/> | BottomLayer           | Signal     | #0000FF | 0               |
| 5   | <input checked="" type="checkbox"/> | TopSilkLayer          | Plane      | #FFCC00 | 0               |
| 6   | <input checked="" type="checkbox"/> | BottomSilkLayer       | Non-Signal | #66CC33 | 0               |
| 7   | <input checked="" type="checkbox"/> | TopPasteMaskLayer     | Non-Signal | #808080 | 0               |
| 8   | <input checked="" type="checkbox"/> | BottomPasteMaskLayer  | Non-Signal | #800000 | 0               |
| 9   | <input checked="" type="checkbox"/> | TopSolderMaskLayer    | Non-Signal | #800080 | 0               |
| 10  | <input checked="" type="checkbox"/> | BottomSolderMaskLayer | Non-Signal | #AA00FF | 0               |
| 11  | <input checked="" type="checkbox"/> | BoardOutline          | Other      | #FF00FF | 0               |
| 12  | <input checked="" type="checkbox"/> | Multi-Layer           | Signal     | #C0C0C0 | 0               |
| 13  | <input checked="" type="checkbox"/> | Document              | Other      | #FFFFFF | 0               |

✓ Setting   Cancel   ?

The Layer Manager setting only works for the current editing PCB.

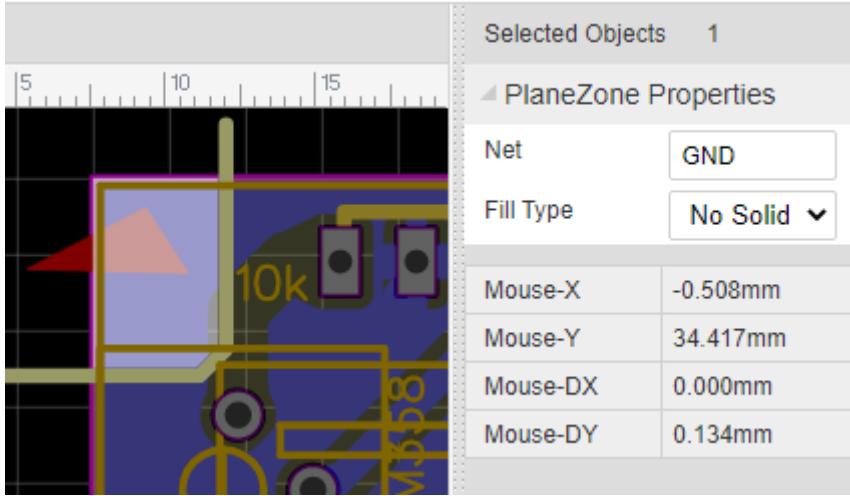
**Copper Layer:** The copper layer of your PCB. EasyEDA support 34 copper layers. The more copper layers the PCB will be more expensive. The TopLayer and BottomLayer is default layer, can not be disable. If you want change the copper layers from 4 to 2, you must delete the inner layers objects first.

**Display:** If you don't want a layer doesn't display at "Layers Tool", you can disable the checkbox. Notice: This option only hide the layer name on the "Layers Tool", the objects of the hidden layer still exist, when you generating the Gerber, they will appear.

**Name:** Layers name. For the inner layer, you can define the name.

**Type:**

- **Signal:** Which is working for the signal. Such as Top and bottom layer.
- **Plane:** When the inner layer type is "Plane", this layer will be copper poured, if you want to separate the copper area you can draw the Track or Arc. You can treat this layer is a only has the copper area, but it's easy than draw the copper area. The track you routed will generate the clearance when generating the Gerber. The "Plane" usually is using for the Power or Ground copper pour on the inner layer. You can set the net for the plane zone.



Notice:

When draw the track to separate the plane zone, the track start point and end point must over the middle line of the board outline track. Otherwise, the plane zone will not be separated; When using the plane layer, the PCB can not exist two closed board outline, only one closed board outline will generate the plane zone.

- **Non-Signal:** Such as silk screen, mechanical layer, document layer etc.

**Color:** You can define the color for each layer.

**Transparency:** You can change the layer transparency.

#### Layer Definition:

- **TopLayer/BottomLayer:** The top side and bottom side of the PCB board, copper layer.
- **InnerLayer:** Copper layer, routing track and copper pour.
- **TopSilkLayer/BottomSilkLayer:** Board silkscreen.
- **TopPasteMaskLayer/BottomPasteMaskLayer:** This layer is the layer used to make the stencil for the SMT pads, helping to solder. This layer has no effect on production if the board is not required to make the stencil.
- **TopSolderMaskLayer/BottomSolderMaskLayer:** The top and bottom cover layers of the board are typically green oil, which acts to prevent unwanted welding. This layer belongs to the negative film drawing mode. When you have wires or areas that do not need to cover green oil, draw them at the corresponding positions. After the PCB is generated, these areas will not be covered with green oil, which is convenient for operations such as tinning.
- **BoardOutline:** The board shape definition layer. To define the actual size of the board, the board factory will produce the board according to this shape.
- **TopAssemblyLayer/BottomAssemblyLayer:** Simplified outline of components for product assembly and repair. Used to export document printing, without affecting PCB production.
- **MechanicalLayer:** Record the information on the mechanical layer in the PCB design, and only use it for information recording. By default, the shape of the layer is not manufactured at the time of production. Some board manufacturers use the mechanical layer to make the frame when using Altium file to production. When using Gerber file, it is only used for text identification in JLCPCB. For example: process parameters; V cut path etc. In EasyEDA, this layer does not affect the shape of the border of the board.
- **DocumentLayer:** Similar to the mechanical layer. But this layer is only visible in the editor and will not be generated in the Gerber file.
- **RatlineLayer:** PCB network ratline display, this layer is not in the physical sense, in order to facilitate the use and set color, it is placed in the layer manager for configuration.
- **HoleLayer:** Similar to the RatlineLayer. For Hole(Non-Plated Hole) display.
- **Multi-Layer:** Similar to the RatlineLayer. For multi-layer hole(Plated hole) display. If the PAD setting layer property as multi-layer, it will connect with all copper layers.

- **DRCErrorLayer** Similar to the RatlineLayer. For DRC(Design Rule Error) marking dispaly.

## Layout Single Layer PCB

The PCB copper layers of EasyEDA are double, EasyEDA doesn't support layout a signle layer directly. if you want to layout a single layer PCB(such as only layout on the bottom layer),

There are two methods:

### Method 1:

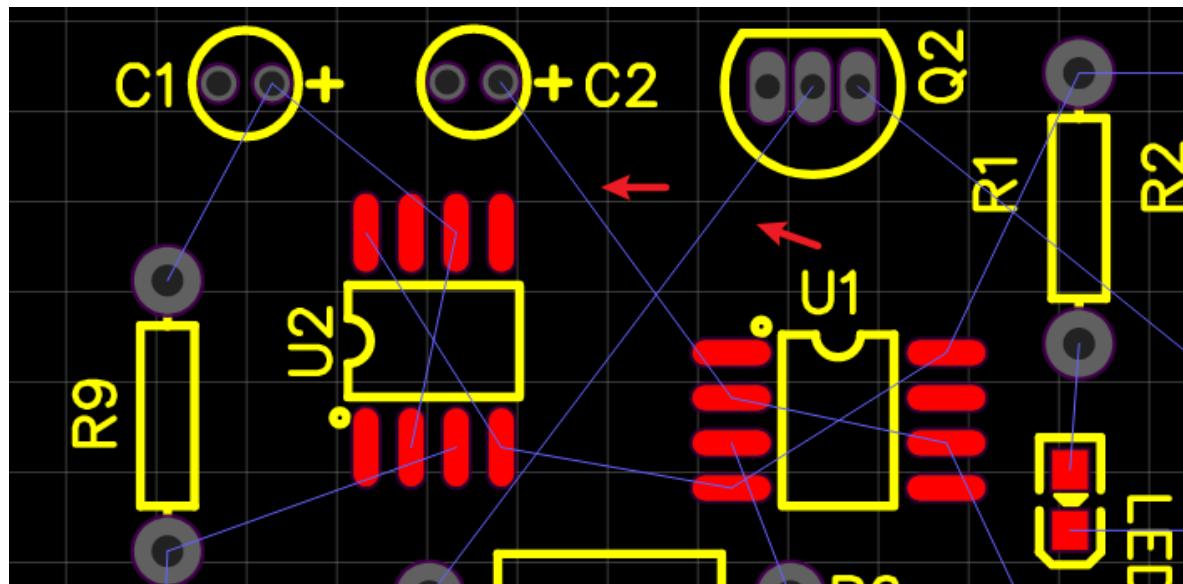
- Route the track and copper on the bottom layer, and without placing via.
- If you are using the footprints which have the multi-layer pads, that will appear on the top and bottom layer, then you need to change all multi-layer pads "Plated" as "No".
- Generate the Gerber, decompress the Gerber zip file, delete the layers which you don't need(such as Gerber\_TopLayer.GTL, Gerber\_TopSilkLayer.GTO, Gerber\_TopSolderMaskLayer.GTS, Gerber\_TopPasteMaskLayer.GTP).
- And re-compress the Gerber to a zip file, and order it.

### Method 2:

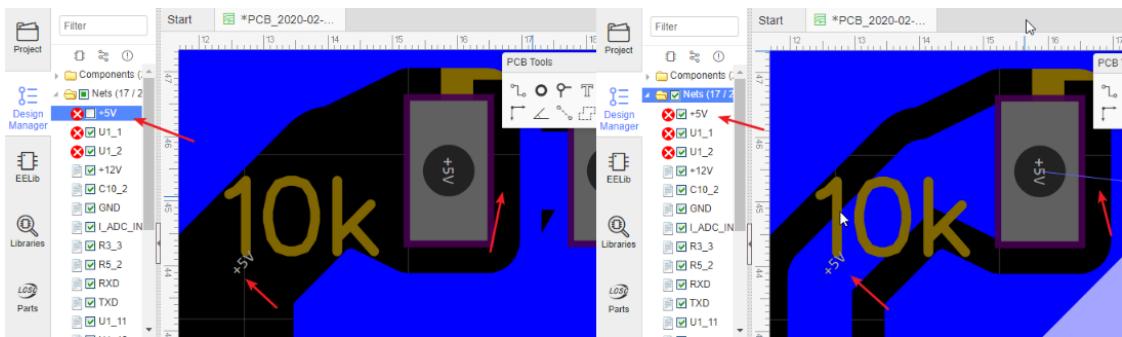
- Design your PCB at one side, if other side has pads etc, you don't need to deal with them.
- Generate the Gerber.
- Add the comment for mention that you need to order the signle layer PCB when order the PCB.

## Ratline

When you layout the track in the PCB, Between Pad and Pad as they have the same net name, a Ratline will be automatically shown among them to reveal that they can be connected with a track.



1. If you want one ratline do not show on the PCB editor, you can deselect the net in the design manager, as below deselect `+5V`:  
If you still draw a track in `+5V` after deselecting, canvas will not display this track and ratline , but it will show a net text with `+5V` as below.



Based on this skill, you don't need to lay GND net before copper area in the PCB.

2. If you want to check the ratlines with highlight, you can click the pencil on the Ratlines Layer as below, and you can change the ratline's color at Layer Manager.

| No. | Display                             | Name                  | Type       | Color   | Transparency(%) |
|-----|-------------------------------------|-----------------------|------------|---------|-----------------|
| 1   | <input checked="" type="checkbox"/> | TopLayer              | Signal     | #FF0000 | 0               |
| 2   | <input checked="" type="checkbox"/> | BottomLayer           | Signal     | #0000FF | 0               |
| 3   | <input checked="" type="checkbox"/> | TopSilkLayer          | Non-Signal | #FFCC00 | 0               |
| 4   | <input checked="" type="checkbox"/> | BottomSilkLayer       | Non-Signal | #66CC33 | 0               |
| 5   | <input checked="" type="checkbox"/> | TopPasteMaskLayer     | Non-Signal | #808080 | 0               |
| 6   | <input checked="" type="checkbox"/> | BottomPasteMaskLayer  | Non-Signal | #000000 | 0               |
| 7   | <input checked="" type="checkbox"/> | TopSolderMaskLayer    | Non-Signal | #000080 | 30              |
| 8   | <input checked="" type="checkbox"/> | BottomSolderMaskLayer | Non-Signal | #AA00FF | 30              |
| 9   | <input checked="" type="checkbox"/> | Ratlines              | Other      | #6464FF | 0               |
| 10  | <input checked="" type="checkbox"/> | BoardOutline          | Other      | #FF00FF | 0               |

3. If you want to hightlight one ratline all the time, you can click a pad, press hotkey H, press it again unhighlight.

4. If you want to change one ratline's color, you can set it at: - Tools - Net Color. After setting the color, you need to click the plus icon on the right. The color is not affected by the color of the ratline layer.

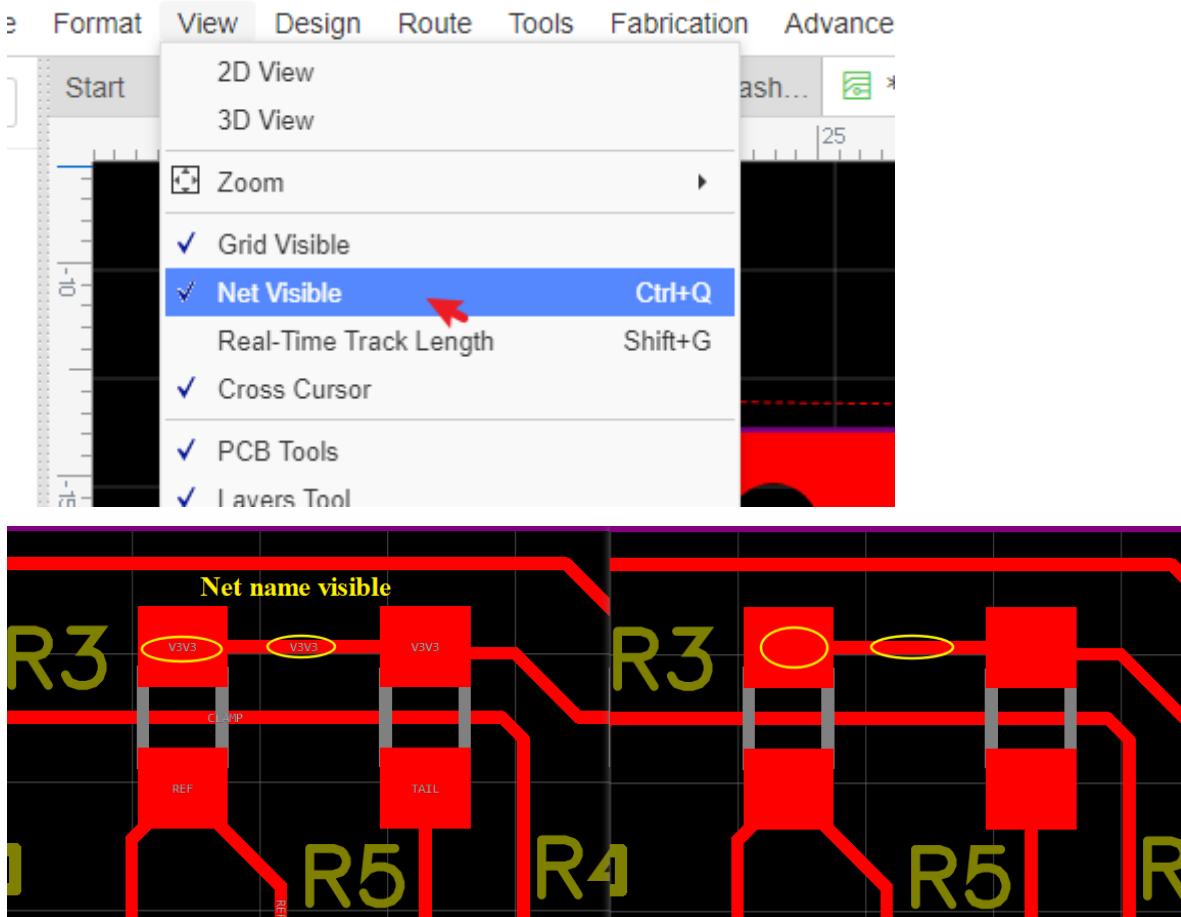
5. If you want to remove one ratline, you just need to remove objects' net. Select it and empty the net.

# PCB Net

## Net Name Visible

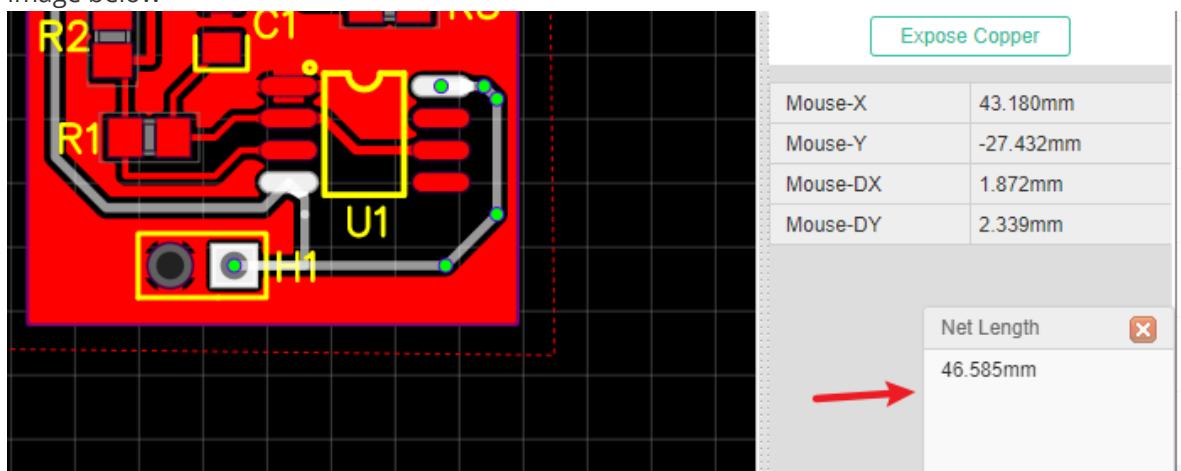
PCB editor can display net name in the track or Pads, if you don't need this feature, just need to turn it off via:

Top Menu > View > PCB Net Visible, or press hotkey **CTR+Q**.



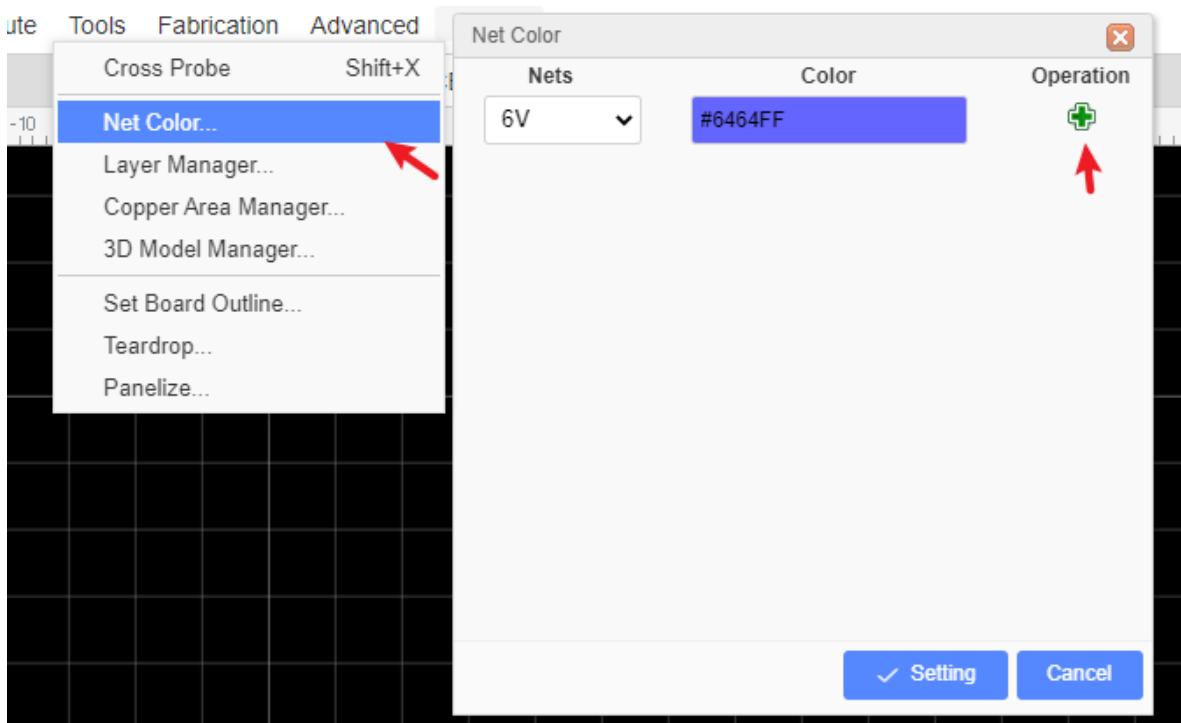
## Net Length

After selecting a track, and then pressing **H** key or click its net at Design Manager, EasyEDA will highlight the whole net and pop a message box to tell you the whole net's length. like in the image below



## Net Color

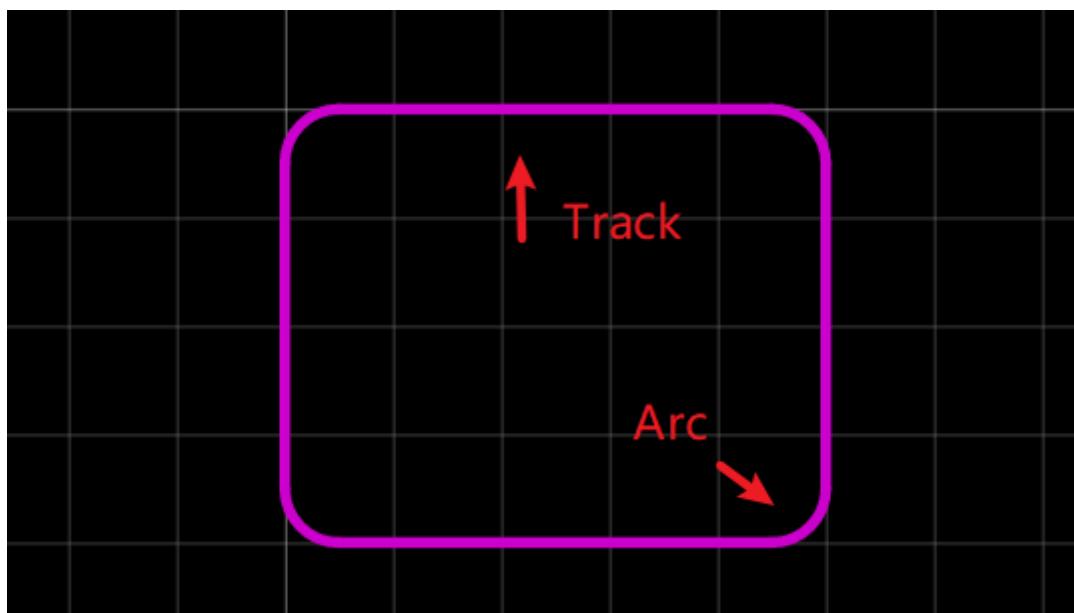
If you want to change one Ratline's or Net's color, you can set it at: **Top Menu- Tools - Net Color**. After setting the color, you need to click the plus icon on the right. The color is not affected by the color of the ratline layer.



When you set a color for a net, you need to click the + button to make it works.

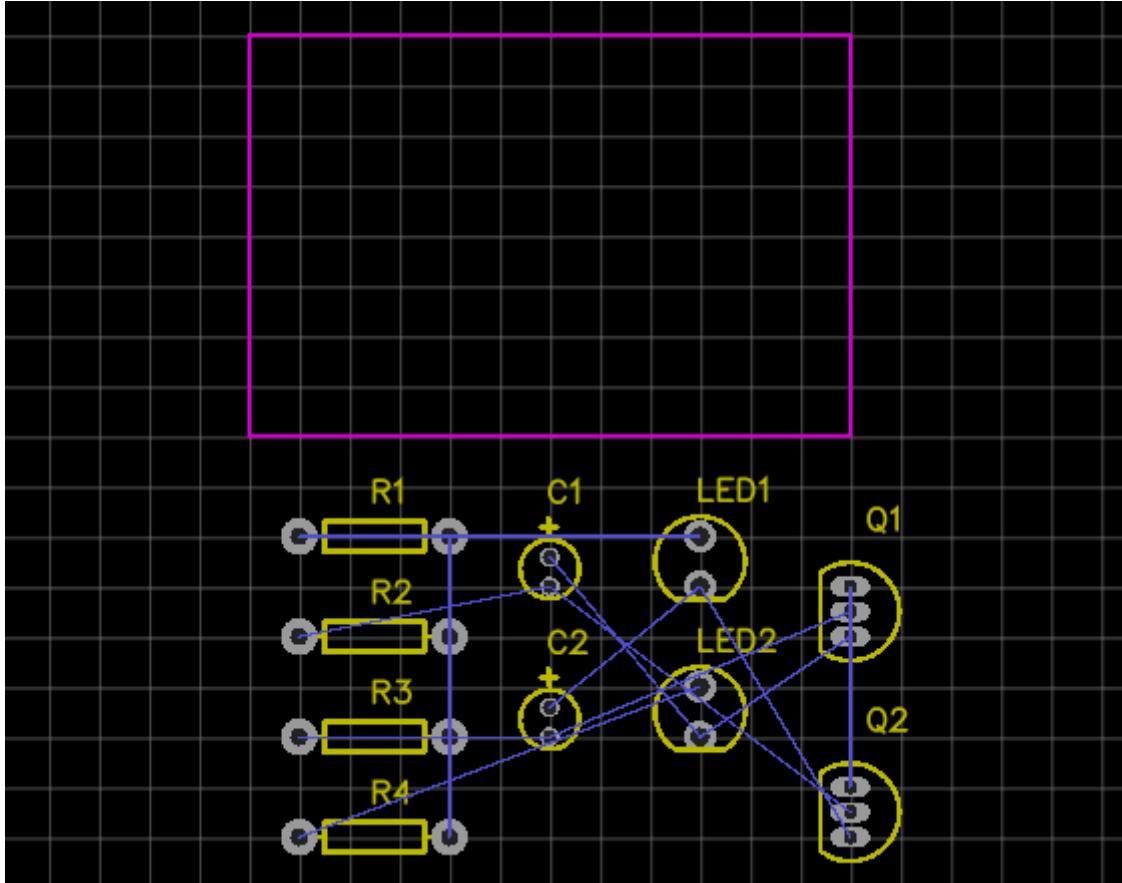
## Board Outline

Before placing footprints we need to create a board outline. The board outline must be drawn on the **Board OutLine** layer. So first, set **Board OutLine** as the active layer, then draw the board outline using **Track** and **Arc** from the PCB Tools palette.



When converting a Schematic to PCB, EasyEDA will try to create a board outline for you.

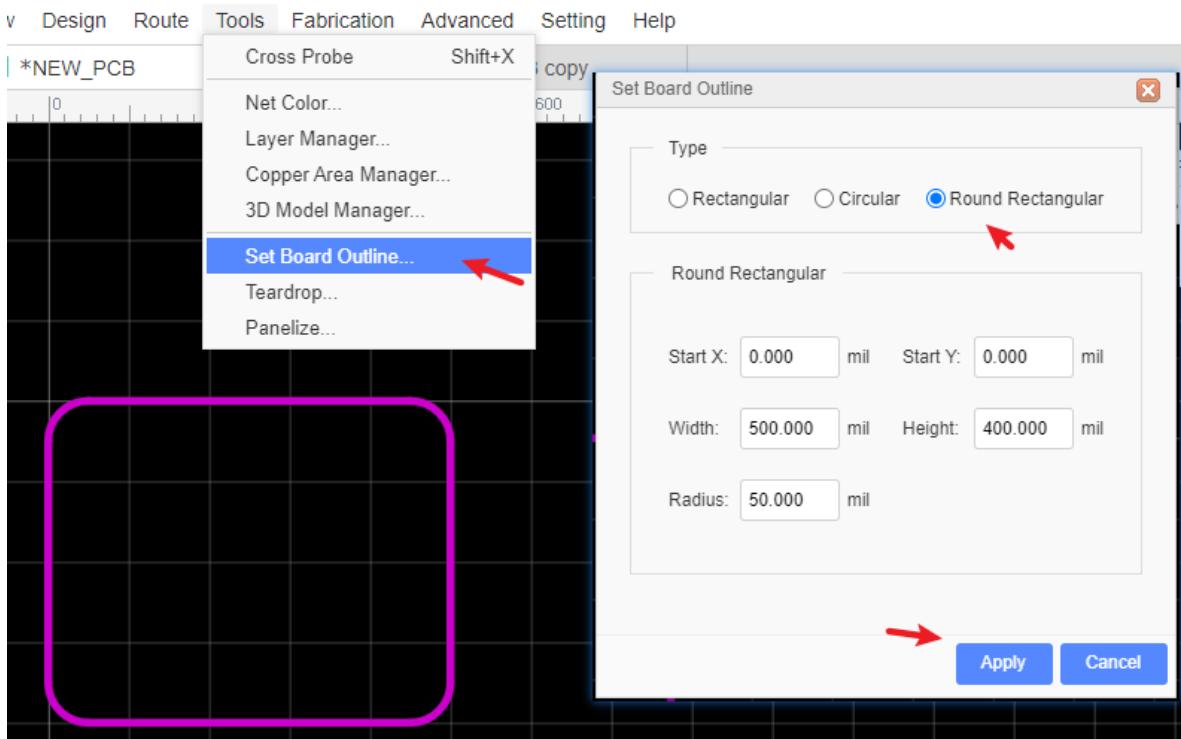
The area of the default board outline area is 1.5 times the sum of the area of all of your footprints, so you can place all of your footprints into this board outline with some allowance for tracking. If you do not like the board outline, you can remove the elements it is made up from and draw your own.



To create a simple rectangular board outline, this arc can be removed and the line X and Y end points edited - either directly in the Properties panel or by dragging the line ends - to close the rectangle.

And EasyEDA provides a **Board outline wizard**, so it is very easy to create a board outline.

Via: **Top Menu > Tools > Set Board Outline**, Or find it on the toolbar.



In this dialog, there's a choice of 3 types of board outlines, Rectangular , Circular, Round Rect. If you need a different more complex board outline, you need to import a DXF file.

#### **Notice:**

- When generate the Gerber, EasyEDA will show error if the board outline doesn't closed or the board outline tracks overlap.
- You can cutout the hole by using the board outline, or use `Hole`, or `Solid Region`(Type: `Board Cutout`) to create the hole instead of using the board outline.
- You can right-click track or circle to convert to board coutout.
- If the board outline doesn't closed, the copper pour will not show up.

## Route Tracks

### Route Tracks

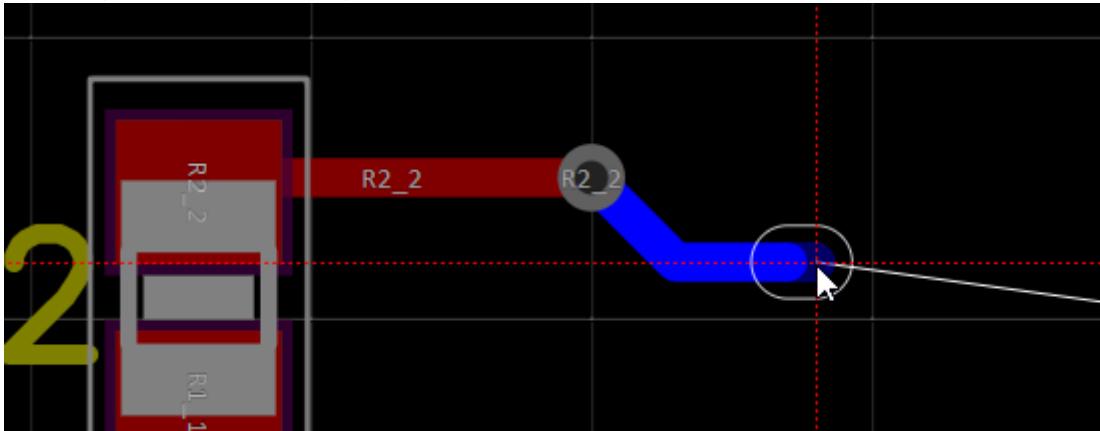
In the schematic editor, we use Wire or the `w` Hotkey to connect Pins, in a similar way in the PCB editor, we use Track to connect Pads. Track allows you to draw PCB tracks and can be found on the PCB Tools palette or using the `w` Hotkey (not T).

#### **Some Tips about Track**

1. Single click to start drawing a track. Single click again to pin the track to the canvas and continue on from that point. Right click to end a track. Double right-click to exit track mode.

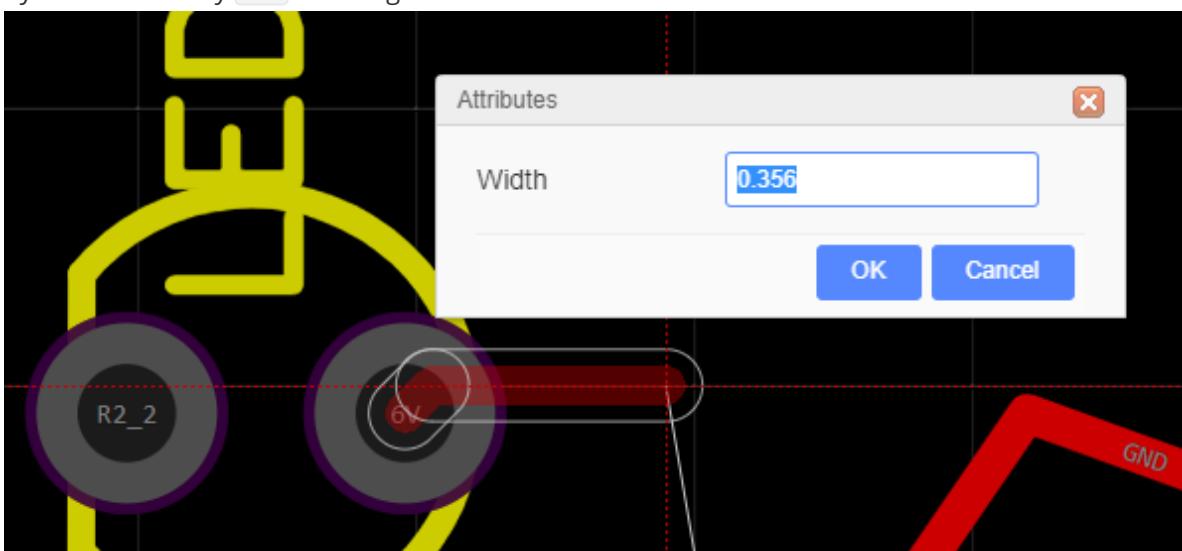


2. Drawing a track at the same time as using a hotkey(for example hotkey **B**) for changing the active layer will automatically insert a Via:



If you start drawing a track on the top layer, you will see it drawn in red, then press the B key to change to bottom layer and you will see EasyEDA insert a grey via and then the track will continue being drawn but now on the bottom layer in blue.

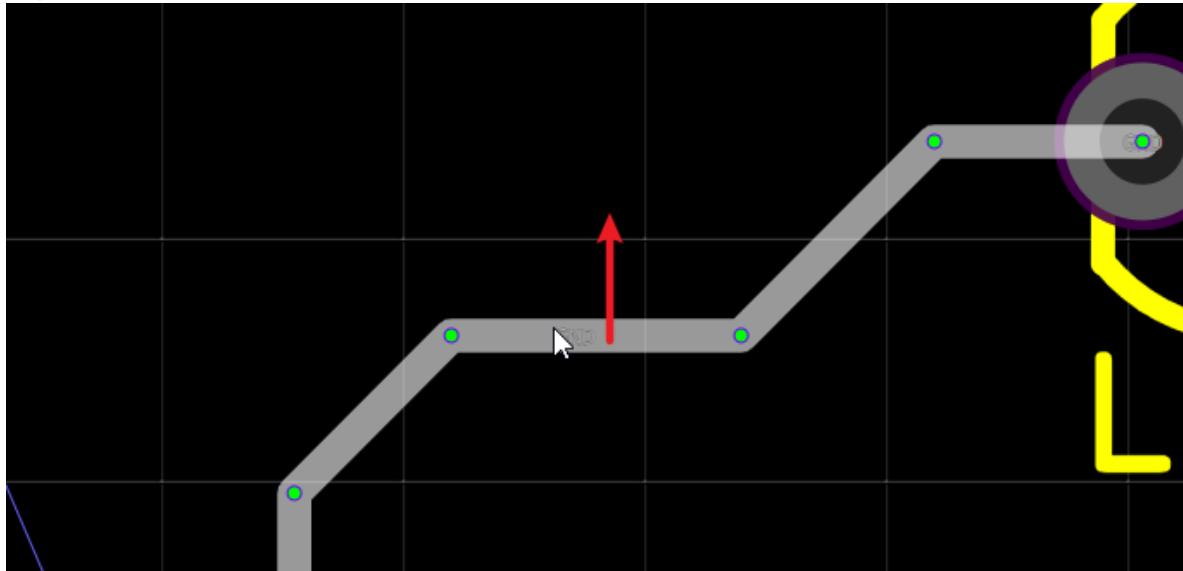
3. Pressing the **+** or **-** Hotkeys when drawing the track will change the width of the track on the fly. Use the hotkey **TAB** to change the track width.



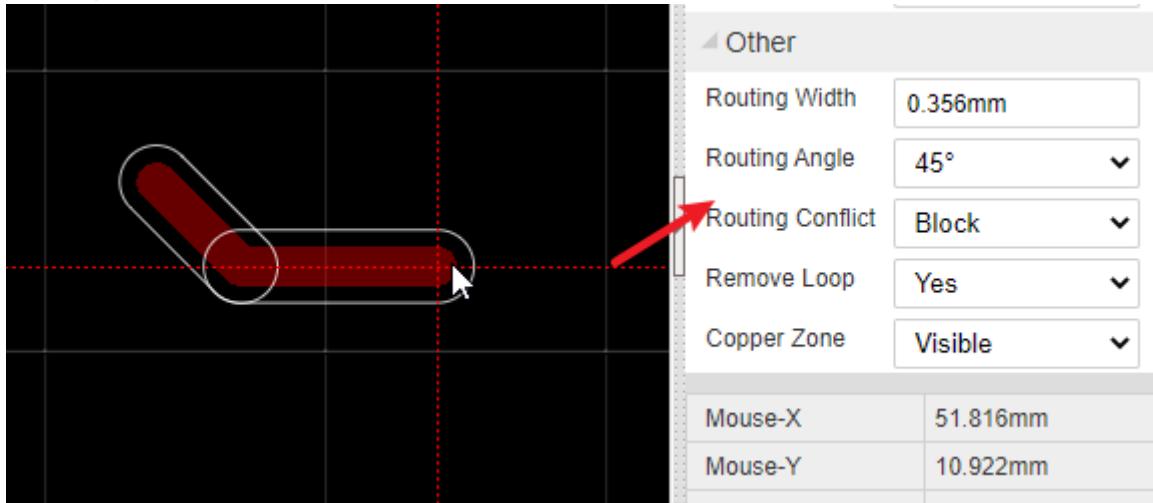
4. Double clicking on a drawn section of the track will add a new vertex at that point. You can drag the vertex to form a new corner. And you can right-click the point and delete it.



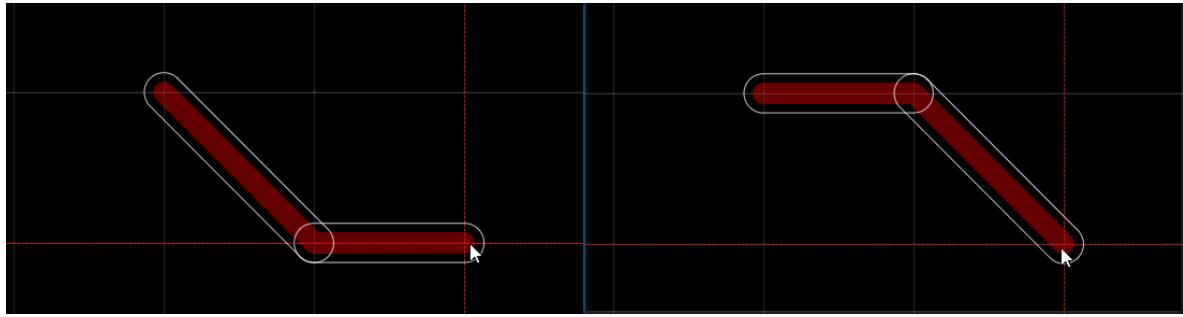
5. Click to select the track and then Click and Drag on a segment of the track to adjust the segment between vertices.



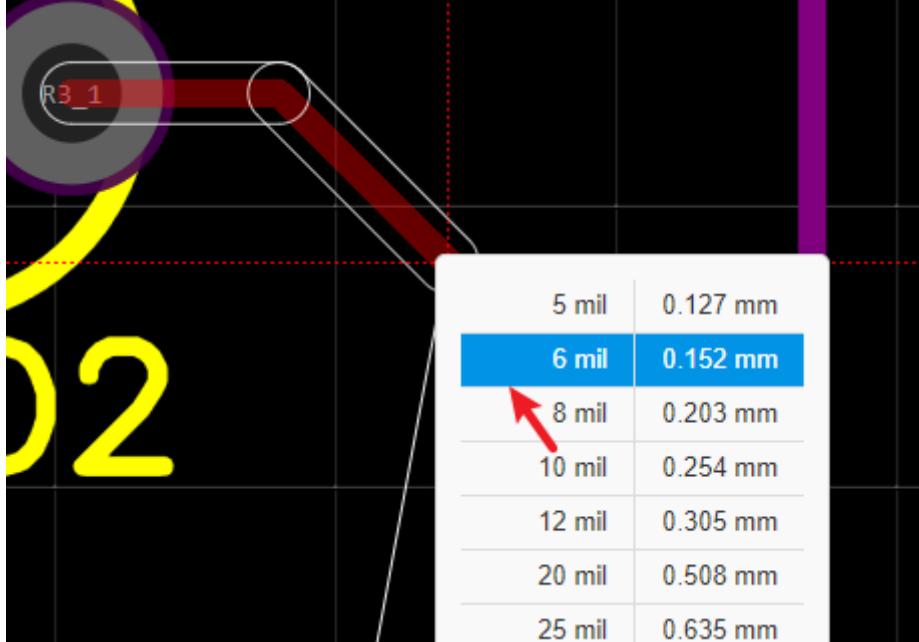
6. Pressing the **L** Hotkey when drawing the track will change the track's Route Angle on the fly. And you can change Route Angle on the Canvas Attributes of the right panel before the next drawing.



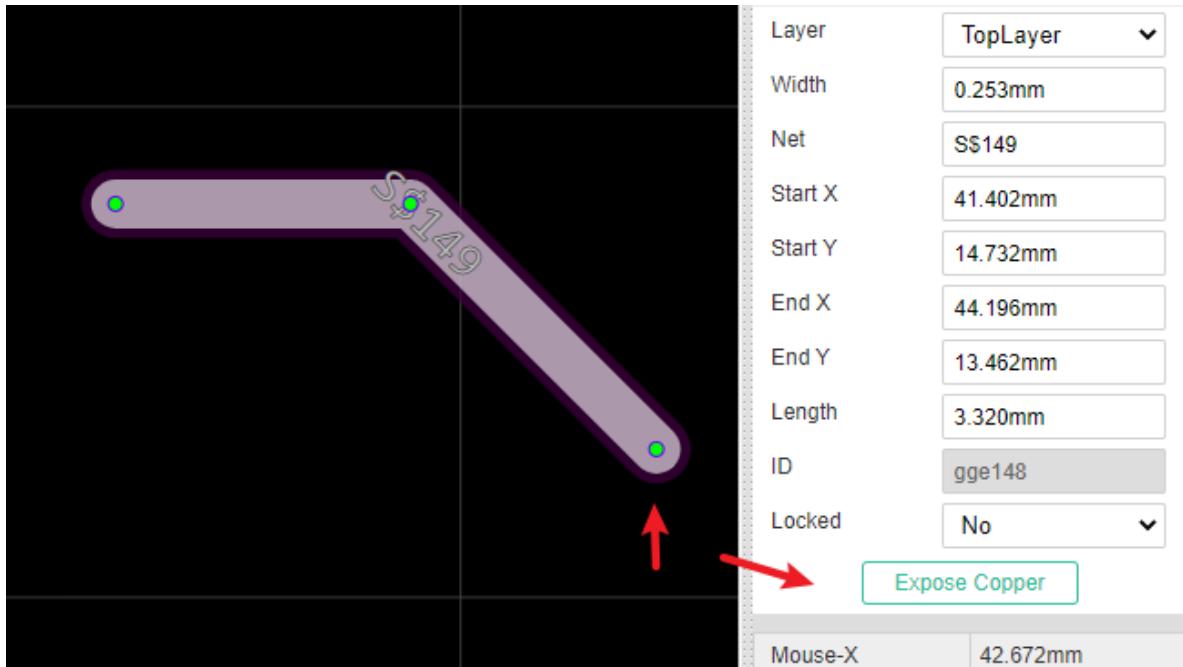
7. You can change inflection direction when routing, just press **Space** key.



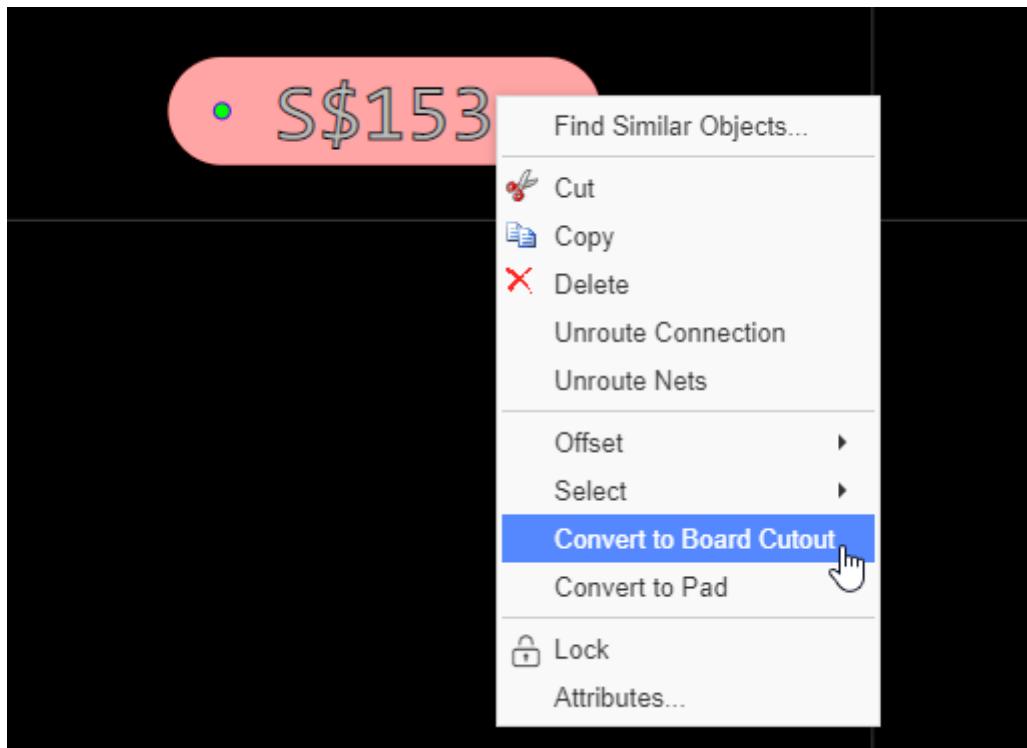
8. If you want to route a track and use "L", and then press "+", you will get two different size track segments. or press "SHIFT+W".



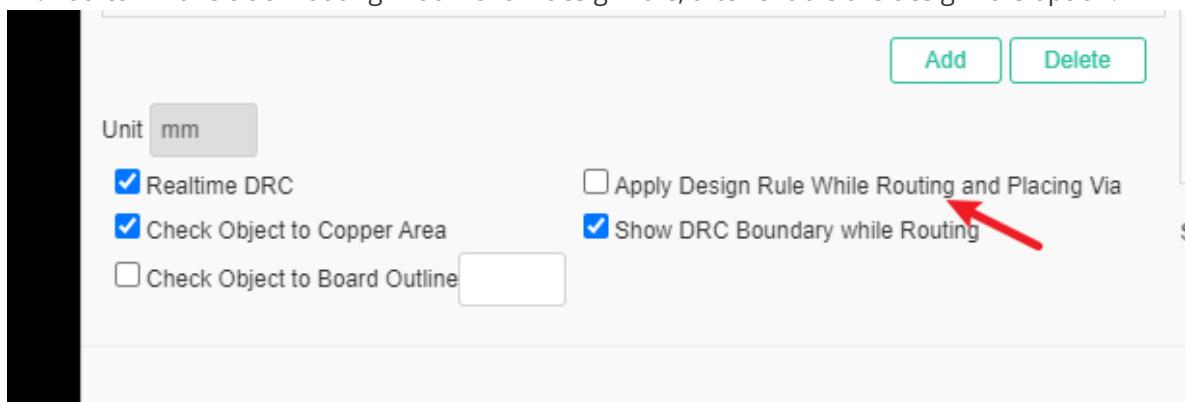
9. If you want to create the solder mask aperture for the track, you can use "Expose Copper" when you select the track on the right-hand panel. The solder mask will be bigger 4mil than the track.



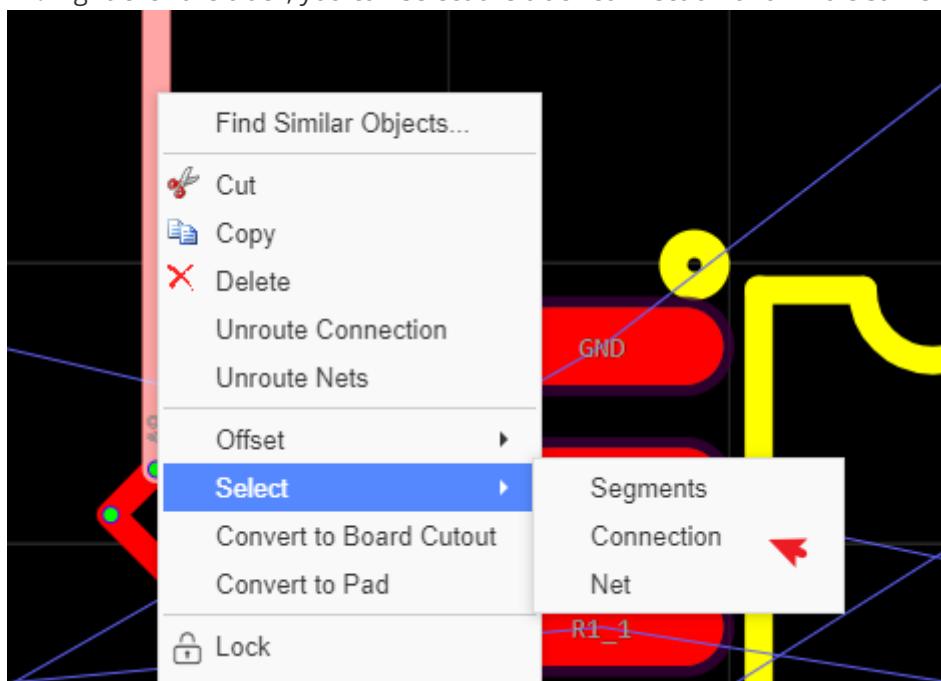
10. And if you want to create the slot hole, you can route a track, and then right-click the "Convert to Board Cutout" menu.



11. You can make track routing width follow design rule, after enable the design rule option.

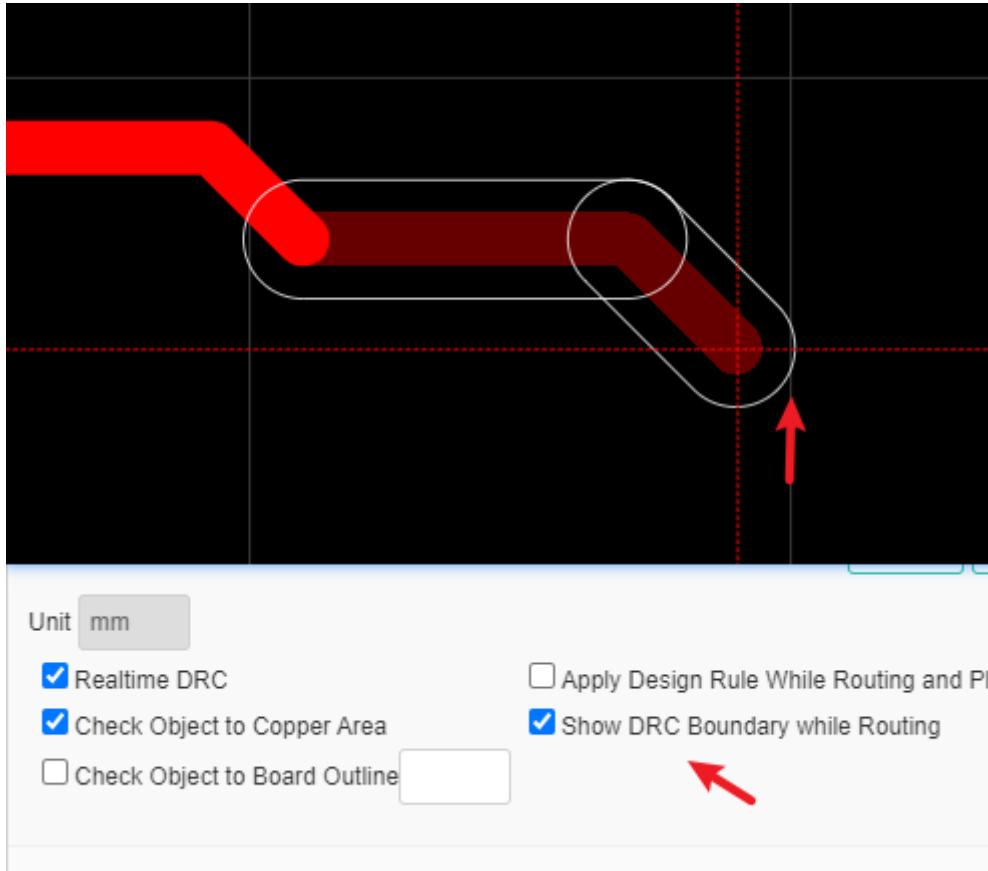


12. Right-click the track, you can select the track connection or a whole same net tracks.

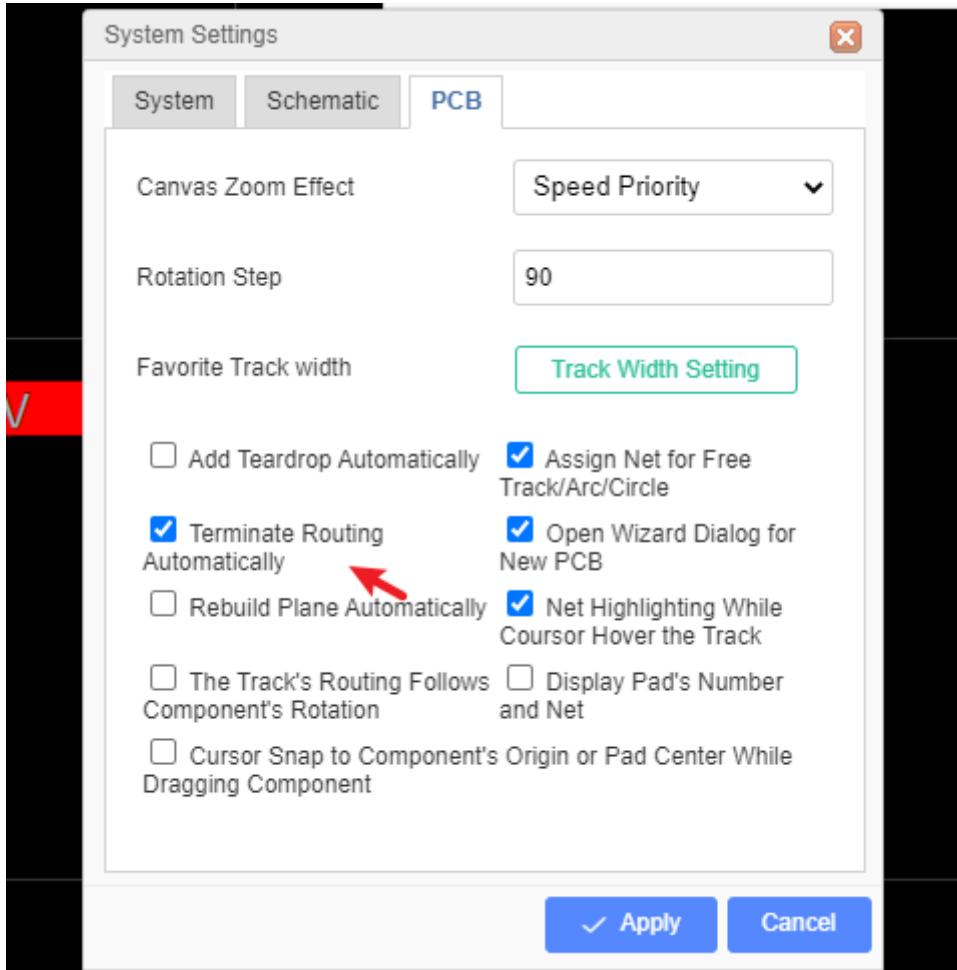


13. If you want to the whole track, you can press **SHIFT** and move it.

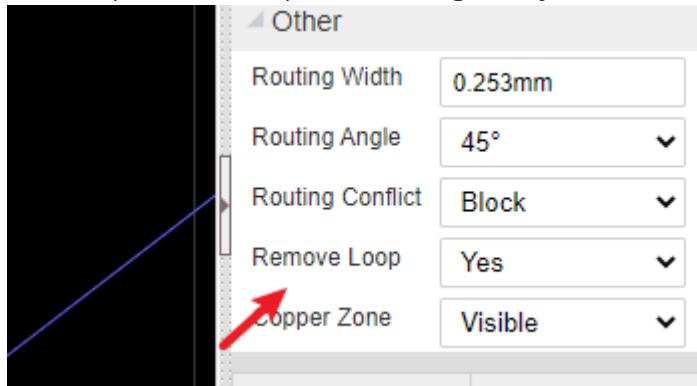
14. You can disable the DRC boundary at Desgin Rule. The size follow the rule.



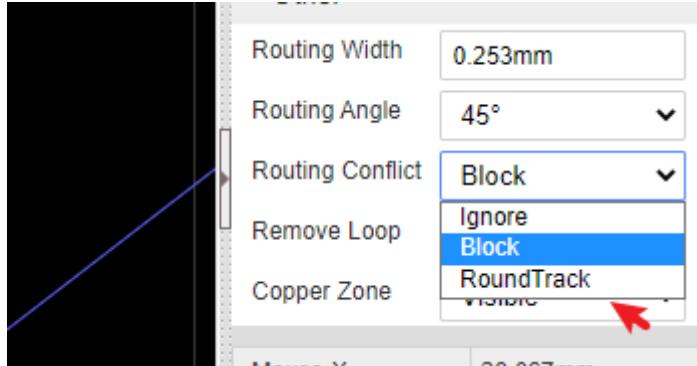
15. If you want to continue routing for a net, you can disable the "Terminate Routing Automatically" option at "Setting - System Setting - PCB".



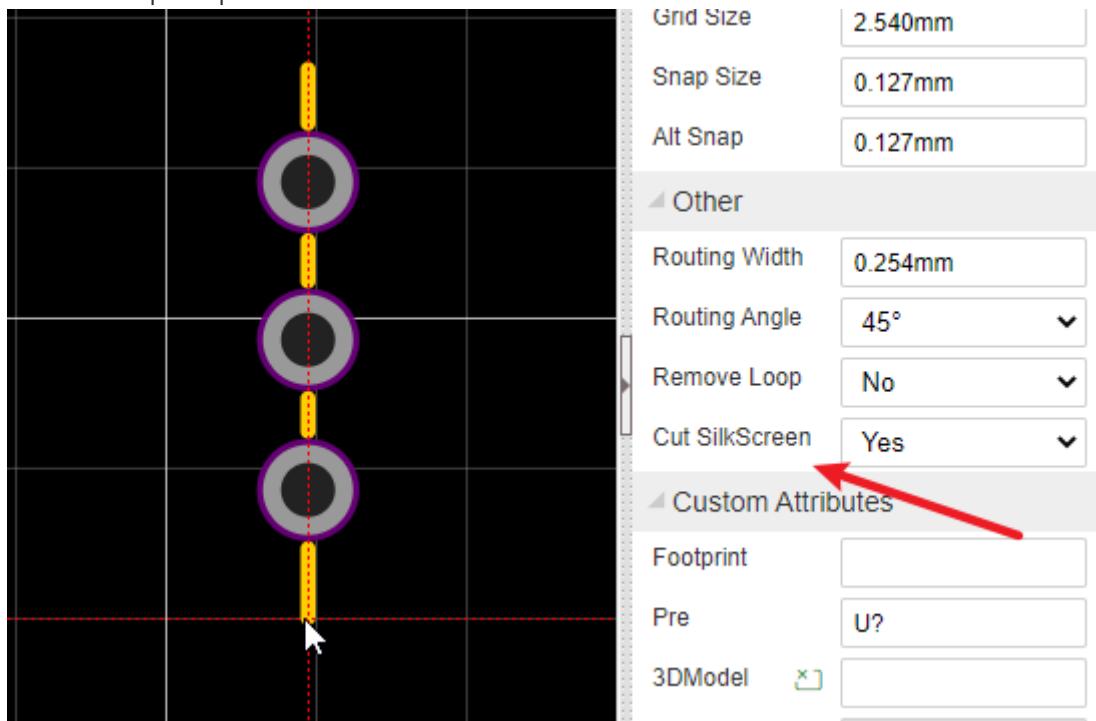
16. Set up Remove Loop while routing, it only works on copper layer.



17. Using Routing Conflict as "RoundTrack" will help you finish routing quickly.

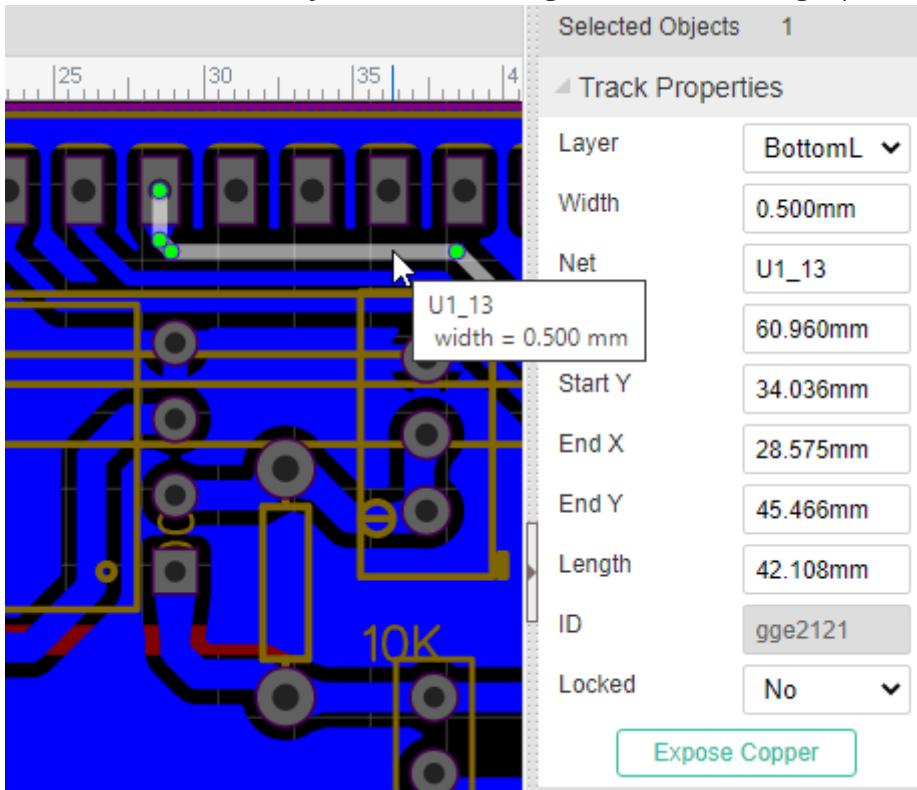


18. When edit the footprint document, you can set up the "Cut Silkscreen" to avoid the silkscreen track overlap the pad.



Track Length

- When a track is selected, you can find its Length attribute in the right panel.



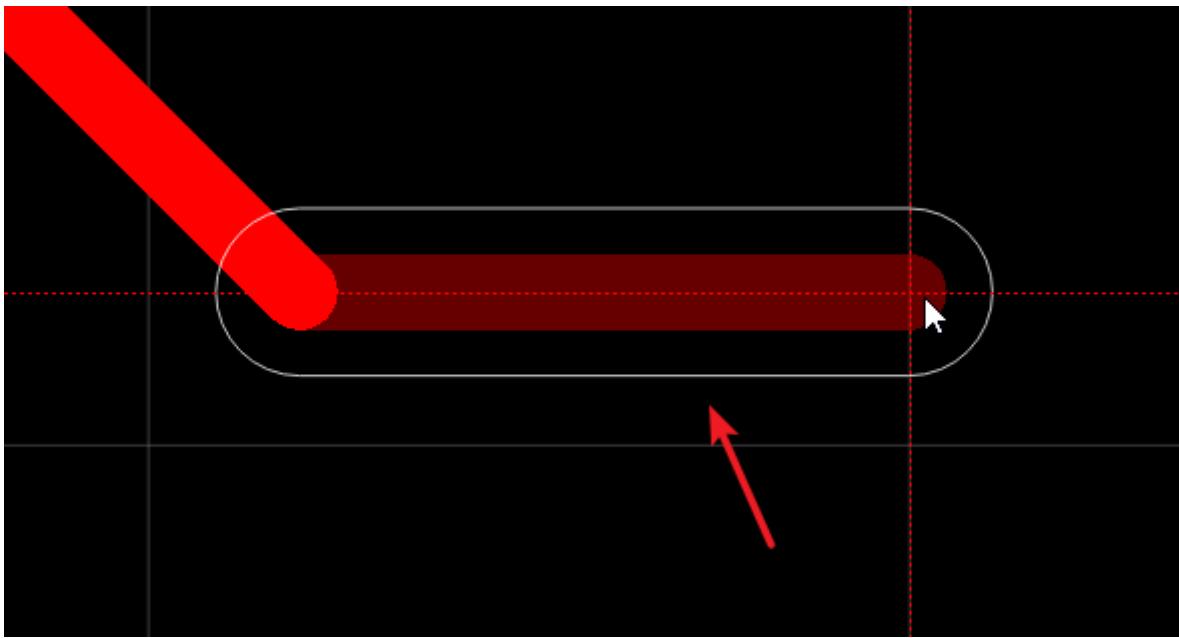
- At left-hand Design Manager, click a net, will pop up a dialog to show you this net track length.
- Click a track, press hotkey H will keep highlight this track and net, and show this net's length.

### Delete a Segment from a Track

- While routing, if you want to undo previous track path, you can press key "Delete" or "Backspace".
- Move your mouse to the segment which you want to delete, click it, then hold **SHIFT** and **double click it**. the segment will be removed. Or right-click delete the node.
- Right-click the track node to delete the track
- Click the track, right-click delete it, or press "Delete" key directly.

### DRC outline

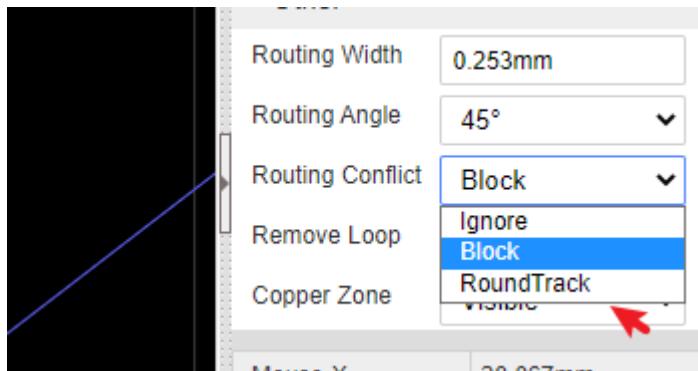
When you routing a track on the signal layer, you will see an outline around the first track, it is the DRC outline, the clearance from outline to the track edge depends on your Design Rule(DRC) clearance setting.



## Routing Conflict

When the PCB comes from the schematic converted, the "Routing Conflict - Block" will be opened automatically.

At the right-hand attributes panel - others, you can find a "Routing Conflict" option:



- Ignore: You can route the track overlap the different net name objects.
- Block: If the track net name different with other objects, this track will be blocked when routing.
- RoundTrack: The track while routing will walk arround the different net objects.
- Push: Doesn't develop yet.

## Differential Pair Routing

EasyEDA provide a easy experience for the differential pair routing.

Via: Top Menu - Route - Differential Pair Routing

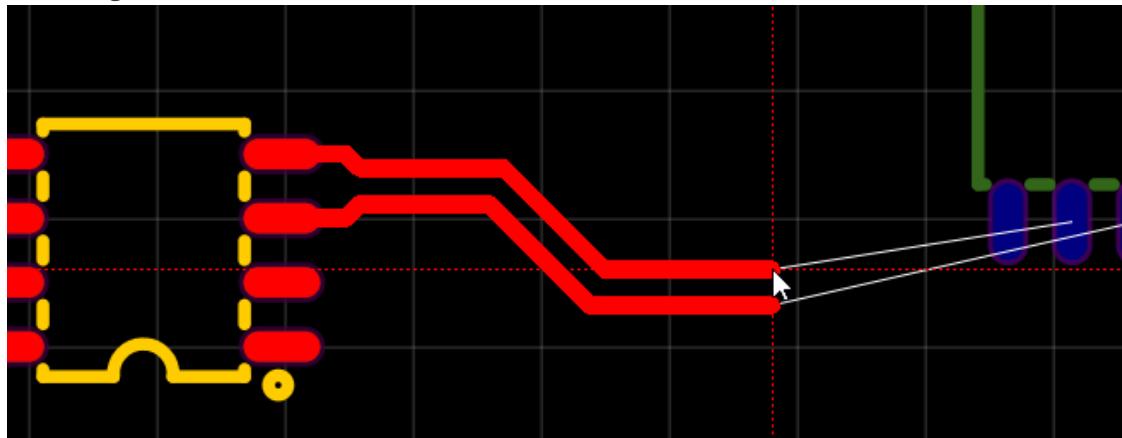
You must make sure the Differential Pair net names must be `xxx_N`, `XXX_P` or `xxx+,xxx-`.

and you need to set Differential Pair net rule at the "Top Menu - Tool - Design Rule" first.

How to route Differential Pair:

- 1.Set the Differential Pair net name as `xxx_N`, `XXX_P` or `xxx+,xxx-`, and set the rule for the Differential Pair net at the "Design Rule"
- 2.Click the menu `Top Menu - Route - Differential Pair Routing`
- 3.Click the one pad of the Differential Pair pads

- 4.Routing

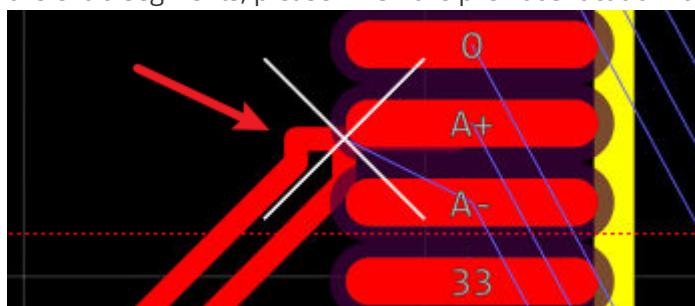


Notice:

- Only for 45 degrees routing, doesn't support hotkey L and Space key.
- Doesn't support the fanout routing.
- Doesn't support the DRC blocking.

Known Issue:

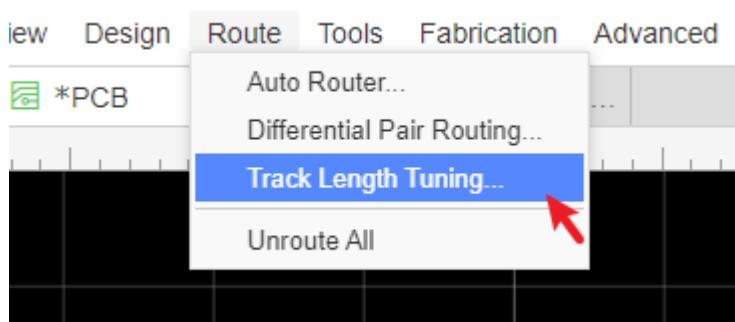
- When finish previous routing location too close with the finish pads, the track will generate the extra segments, please finish the previous location far away from finish pads.

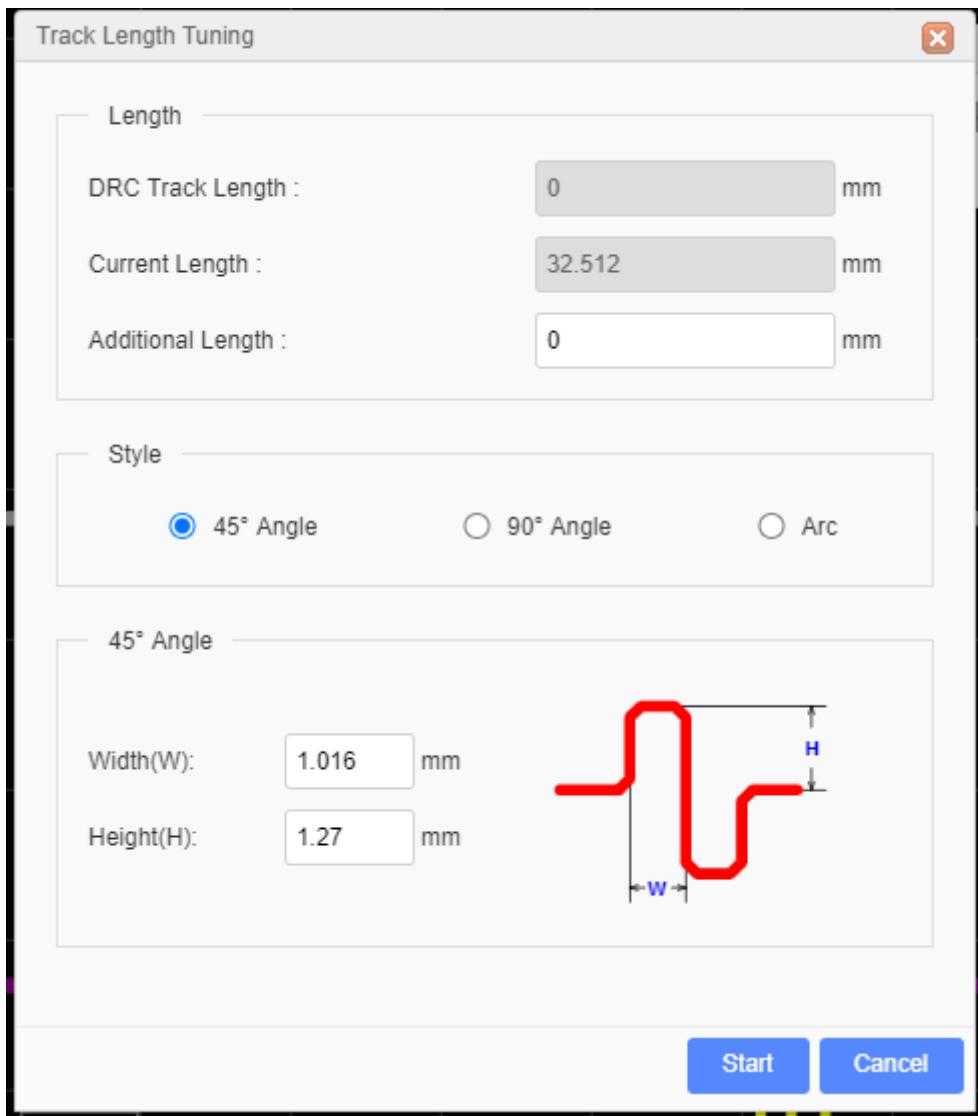


## Track length Tuning

You can tuning your track very easy on the editor.

Via: Top Menu - Route - Track length Tuning





#### How to use:

- 1. Select the track which is you want to tune
- 2. Click the menu: Top Menu - Route - Track length Tuning
- 3. Set the parameter, start
- 4. Left-Click the track where is you want to start, and then move the mouse
- 5. When the track length close your setting, it will stop tuning.



#### Notice:

- Doesn't support one side tuning for a track yet
- Doesn't support auto push or avoid the nearby tracks yet

## Cloud Auto Router

For some simple or prototype PCBs, you may want to use the auto router function to save time. Layout is a time costly and dull job. EasyEDA spends lots of time to provide such a feature and it is loved by our users.

Before using the auto router, you need to set the board outline for the PCB.

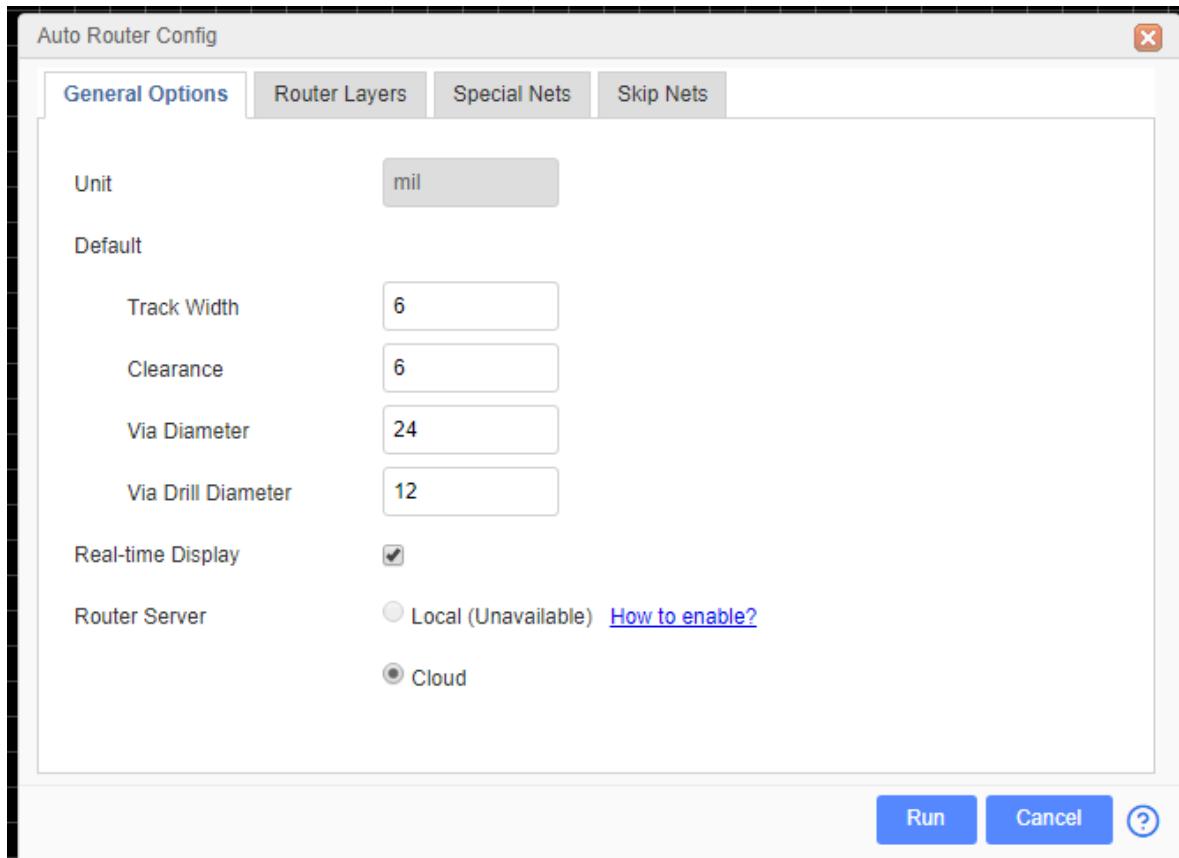
**Auto router is not good enough! Suggest routing manually! You can use "RoundTrack(Walk Arround)" option to route tracks, via right-hand panel - Routing Conflict.**

Steps:

**1 Click the the auto router button from the Top Menu"Top Menu> Route > Auto Router"**

**2 Config the auto router**

After you click that button, you will get a config dialog like in the image below.



In the config dialog, you can set some rules to make the auto router result professional. These rule must equalize or more than DRC setting.

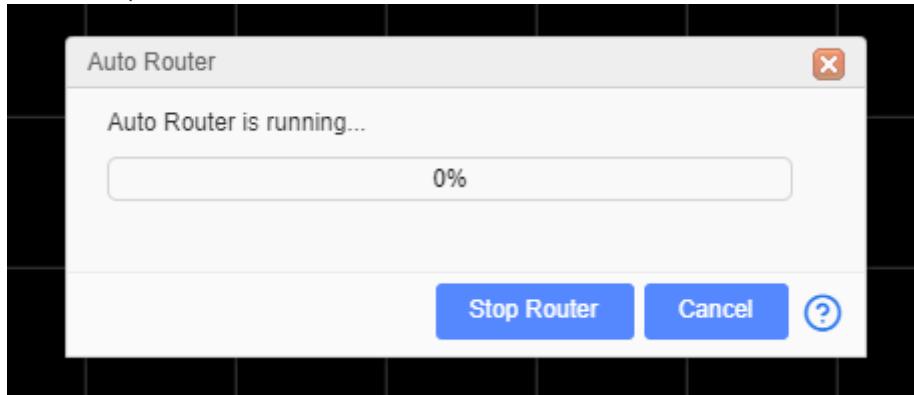
### General Options

- **Unit:** The unit follows PCB canvas unit.
- **Track width:** The auto-route track width.
- **Clearance:** The clearance of the objects.
- **Via Diameter/Via Drill Diameter:** The via placing by auto-router.
- **Realtime Display:** when you select it , the real time routing status will show on.
- **Router Server:**
  - **Cloud:** Using EasyEDA online server.
  - **Local:** Using the local auto router server, when you click the Auto Router icon, the editor will check the local router server available or not automatically. How to use please see as below.

- **Router Layers:** If you want to route inner layer, you have to enable the inner layer first.
- **Special Nets:** For the power supply track, you may want it to be bigger, so you can add some special rules.
- **Skip Nets:** If you like to keep the a net with no route, you can skip it. For example, if you want to use copper area to connect `GND` net, you can skip the `GND` net. If you want to reserve the routed track, you need to select the `skip Routed Nets`.

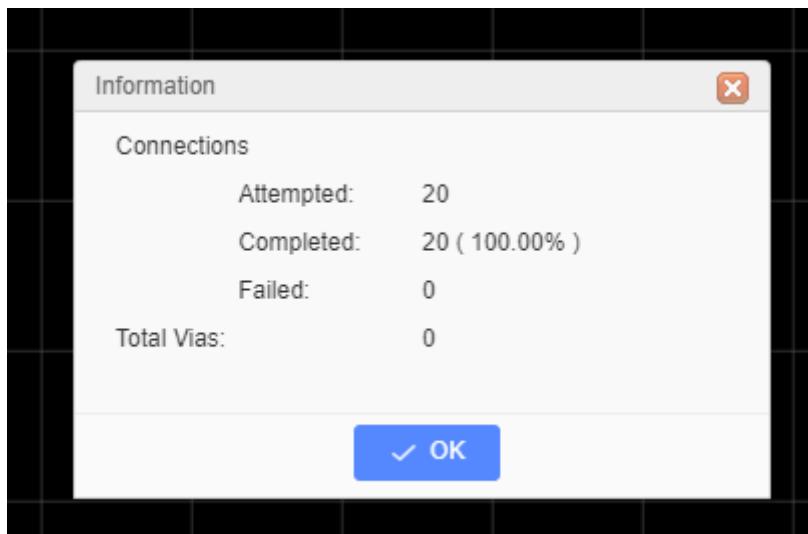
### 3 Run it

After click the "Run" button , The real time check box will let you see how it is going, but it will make the process a little bit slow.



Waiting for a few minutes, after adding bottom and top copper area, you will get a finished PCB board.

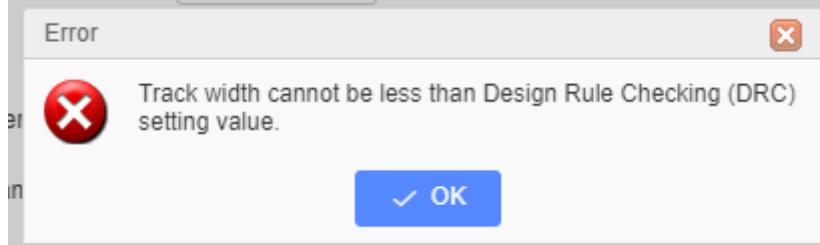
When finish, will pop up a window.



The connection means the track connect times.

Notice:

- The parameter can't less than DRC rule, otherwise will report error.



# Local Auto Router

EasyEDA suggest that using local auto router rather than using the cloud server, because when many users using cloud server, the cloud auto router will fail. Only support 64bit system.

For the local auto router, please follow the steps as below:

- **1.Download the local auto router server.**

EasyEDA: [EasyEDA Router.zip\(134MB\)](#)

Supported OS:

- Windows7(x64) or later 64bit Windows
- Ubuntu17.04(x64) or other 64bit Linux, Linux recommend [Deepin](#)
- macOS(x64)

- **2.Unzip it to the User folder, such as driver D.**

- **3.Configure the browser.**

**Notice:** Please use the latest Chrome or Firefox !!!

- **1)Chrome**

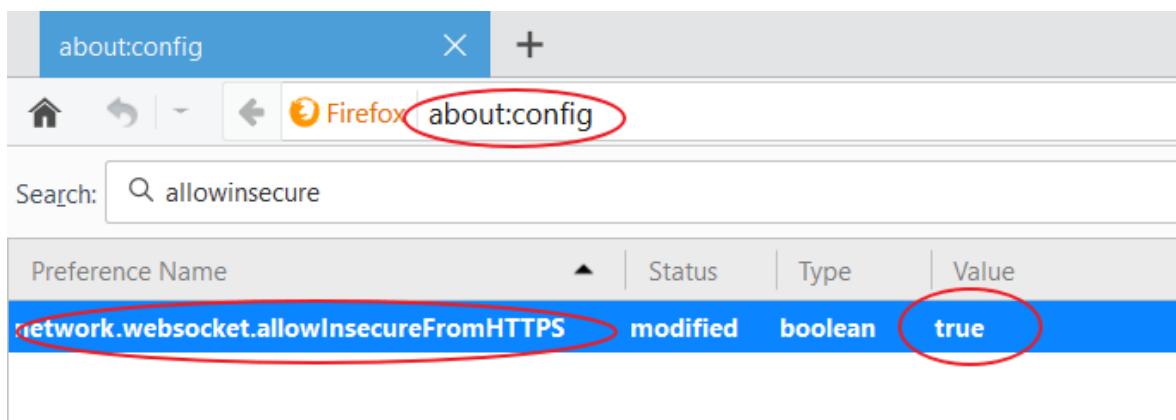
The Chrome Browser don't need to be configure, If the local auto router is unavailable, you have to upgrade Chrome to version 60.0.3112.78 or later.

- **2)Firefox**

- 1.Type "about:config" into the address bar then press enter.
- 2.Search and double click the options as below (change the values to "true"):

```
network.websocket.allowInsecureFromHTTPS
```

```
security.mixed_content.block_active_content
```



- 3.Re-open Firefox.

- **4.Open the decompress folder, Start local Auto Router(don't need to install, just run it and keep the command window open):**

- Double click `win64.bat` in Windows.
- Run `sh lin64.sh` on command terminal in Linux. Open the terminal, use the `cd` command to change the directory to the `lin64.sh` location, and type `sh lin64.sh`, then enter.
- Run `sh mac64.sh` on command prompt in MacOS. Open the terminal, use the `cd` command to change the directory to the `mac64.sh` location, and type `sh mac64.sh`, then enter.

- **5.Open the editor, open the PCB, Click the Auto Router\*\* icon at editor to start auto-router.\*\***  
If the local router server is available, the dialog will tell you. Click the **Run** button, the dialog will show the process.

## Tips

Sometimes, if you can't get it done, try the tips below.

- Make sure the net of PCB doesn't contain the special characters, such as ; ~ \ / [ ] = etc. the characters - and \_ are supported.
- Make sure the board outline is closed, doesn't have board outline overlap situation.
- Make sure there are no DRC clearance errors (short circuit issue), such as two different network pads overlapping, or different net pads in the same location within the footprint.
- Make sure no footprint outside the board outline.
- Make sure PCB rule doesn't have 3 decimal places, EasyEDA auto router only supports 2 decimal places.
- Skip the GND nets, add copper area to GND net.
- Use small tracks and small clearance, but make sure the value is more than 6mil.
- Route some key tracks manually before auto routing and ignore them when auto routing.
- Add more layers, 4 layers or 6 layers, but that will make the PCB more expensive.
- Change the component layout, make them have more space between each other.
- Don't make any via/pad/solid region overlap the different net objects.
- Use local auto router rather than cloud server.
- Tell the error detail to us and download and send your PCB file as EasyEDA Source json file:  
<https://docs.easyeda.com/en/Export/Export-EasyEDA-Source-File/index.html>  
via email.

[support@easyeda.com](mailto:support@easyeda.com)

Some professional people don't like the auto router, because they think auto router is not professional, but you can use the auto router to check your placement to check the density of your PCB.

At present, the auto router is not good enough, suggest routing manually, we will improve it in the future.

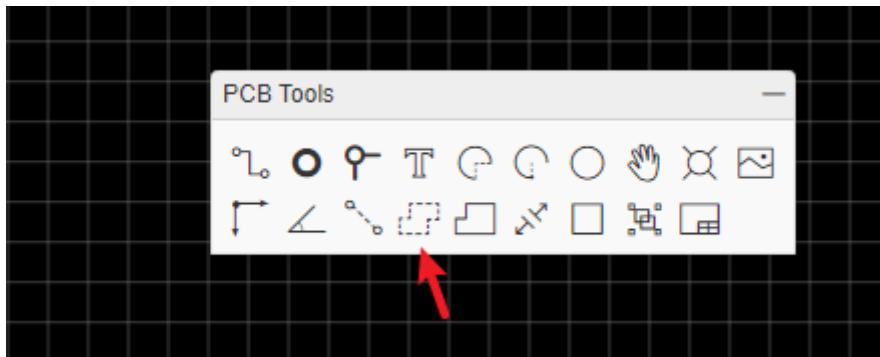
# Copper Area

---

## Copper Area

---

Sometimes you will want to fill in or flood an area with copper(Copper Pour). Normally after drawing the copper area, set the net it is to be connected to (floating copper areas are not recommended because they can cause EMC and Signal Integrity (SI) problems).

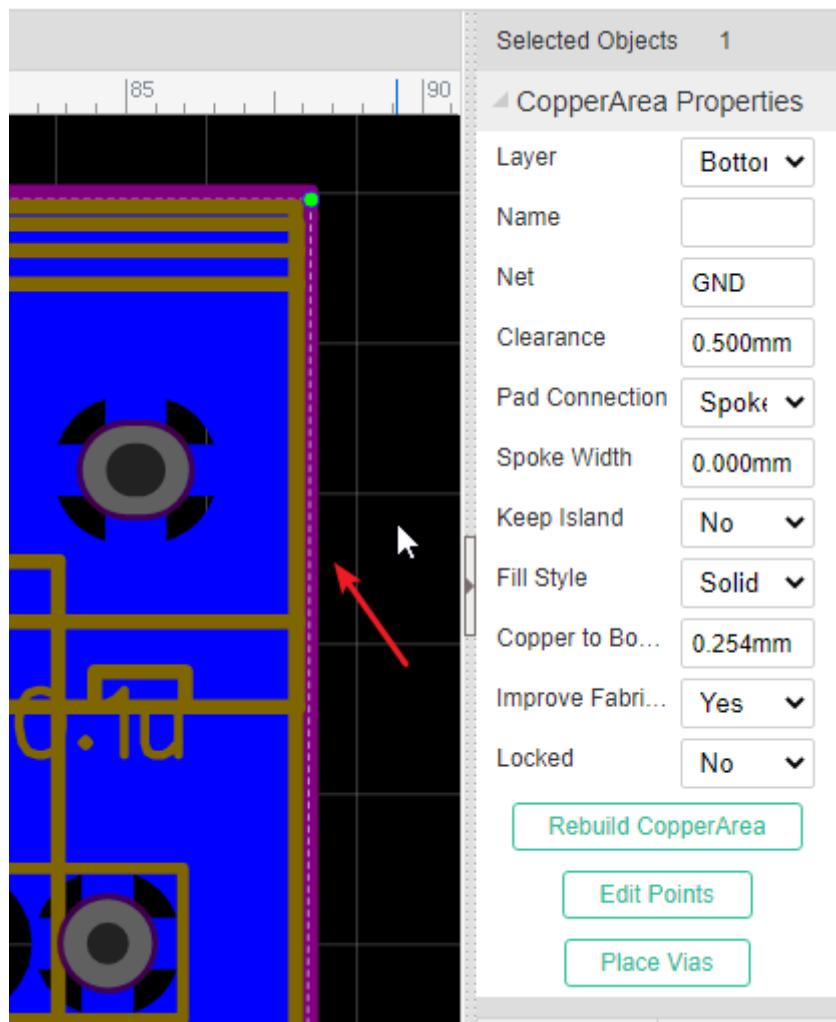


**Before using Copper Area, please make sure your PCB has a closed board outline!**

Usually this copper area will be connected to a net such as **GND** or a supply rail. You can draw the outline of a flood using the **Copper Area** button from the PCB Tools palette.

## Copper Area Attributes

When selecting the copper area outline, you can find its attributes from the right hand **Properties** panels.



**Layer:** Bottom, Top, Inner1, Inner2, Inner3, Inner4 etc.;

**Net:** the net that the copper area is connected to;

**Name:** set a name for it.

**Clearance:** clearance of the copper area from other nets and floods;

**Pad Connection:** direct or spoke (i.e. a cross shaped heat shunt);

**Spoke Width:** When Pad Connection is Spoke, you can set the Spoke width, which is copper area fill connect with Pads.

**Keep Island:** Yes/No. This keeps or removes any isolated areas of copper created as part of the flooding process. It is usually good practice to remove these unless you really need them to maintain a more even spread of copper (copper balance) on your PCB.

**Fill Style:** Solid/No Solid/Grid. Selecting **No Solid** will remove the fill so that you can see the tracks more clearly; when select Grid, you can set the grid spacing and grid width.

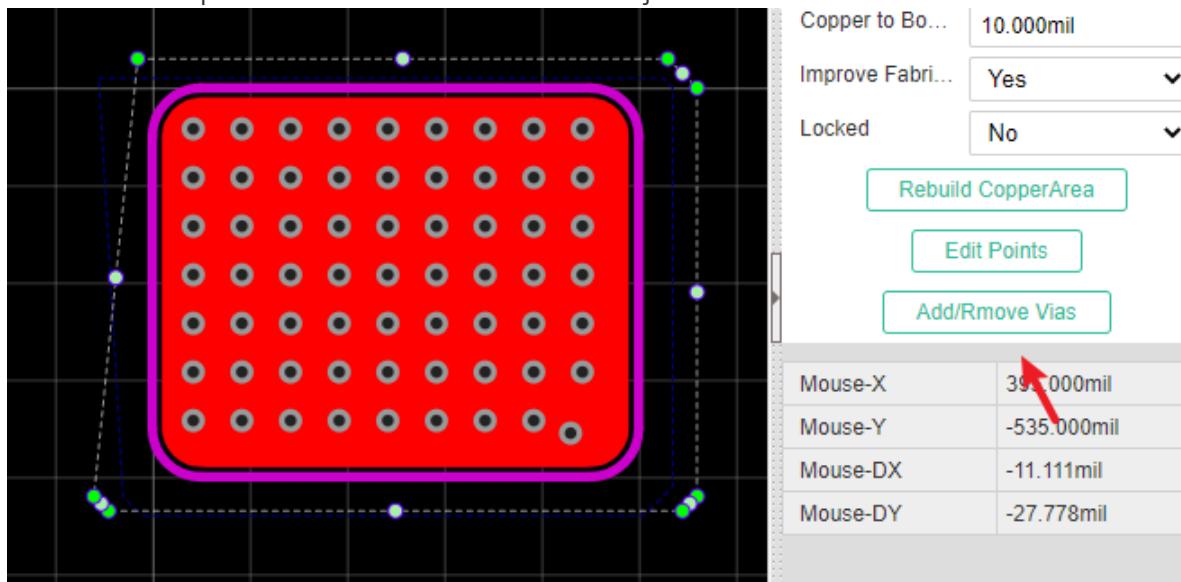
**Copper to BoardOutline:** Setting the clearance between copper with board outline.

**Improve Fabrication:** Yes/No. If you set as No, you will see much sharp copper corners, that is not good for PCB fabrication.

**Rebuild CopperArea:** Click the button to Rebuild Copper Area if you make any changes.

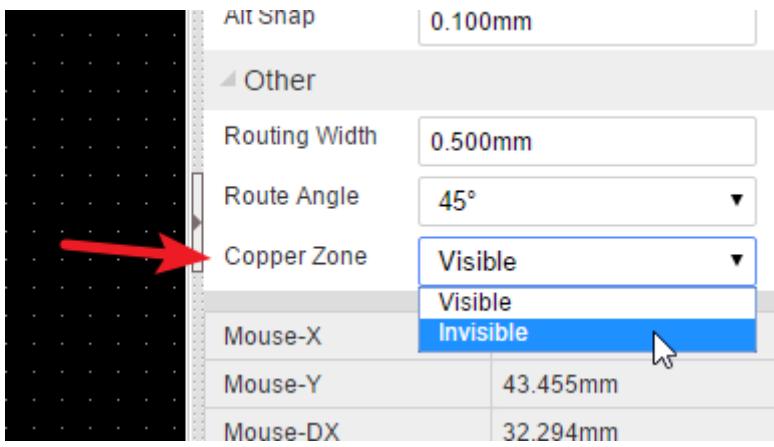
**Edit Points:** You can edit the copper area shape manually, any shape as you want.

**Add/Remove Vias:** When you add copper areas at two and more layers which are having same net, you can add multiple vias for the copper fill area, just click the "Add/Remove Via" button, then set the via parameter. The vias will avoid the objects if the via conflict the DRC.

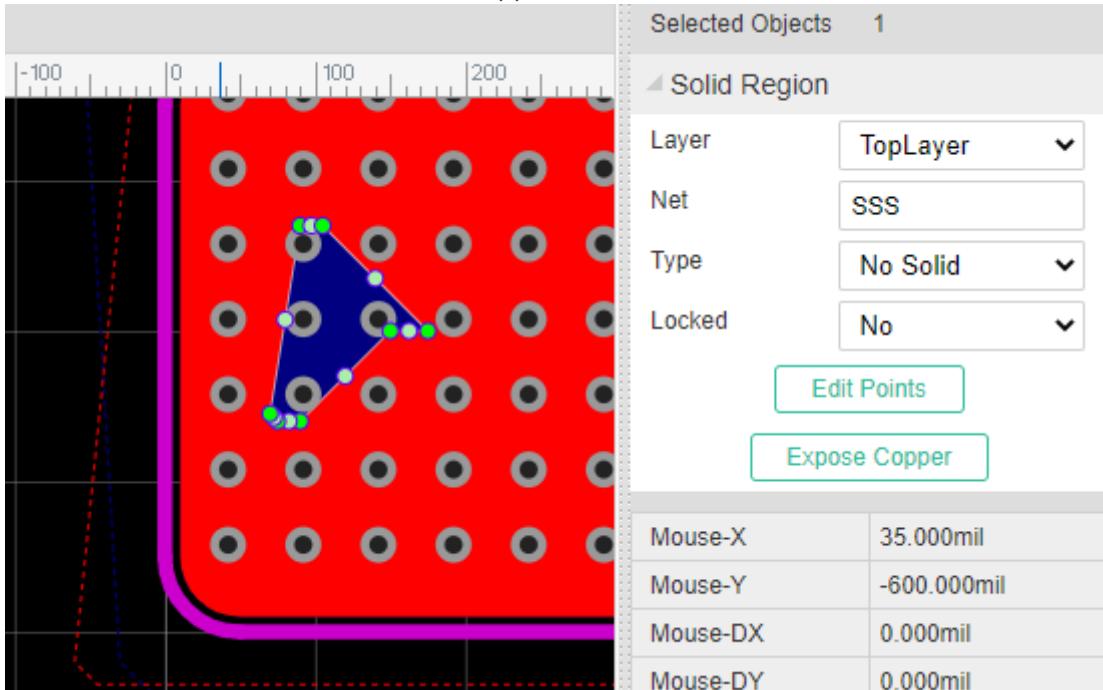


## Tips

- Hotkey **E** to start draw copper area.
- Hotkey **L** to change drawing type(90 degrees or 45 degrees or Arc)
- Hotkey **shift+B** to build all of the copper areas.
- Hotkey **shift+M** to hide copper areas fill zone, just show the copper outline.
- Hotkey **Delete** or **Backspace** to redo previous steps.
- If you after copper pours but no copper fills show up, you need to set it a net same one of the PCB nets, or keep the island as YES, and the rebuild the copper area via "Rebuild Copper Area" button or "SHIFT+B".
- If you want to hide the copper area and keep routing tracks, you can set the copper zone invisible at the right-hand panel.



- If you want to cutout some copper corners, you can use "Solid Region - No Solid", and then set different net for it, and rebuild the copper area.



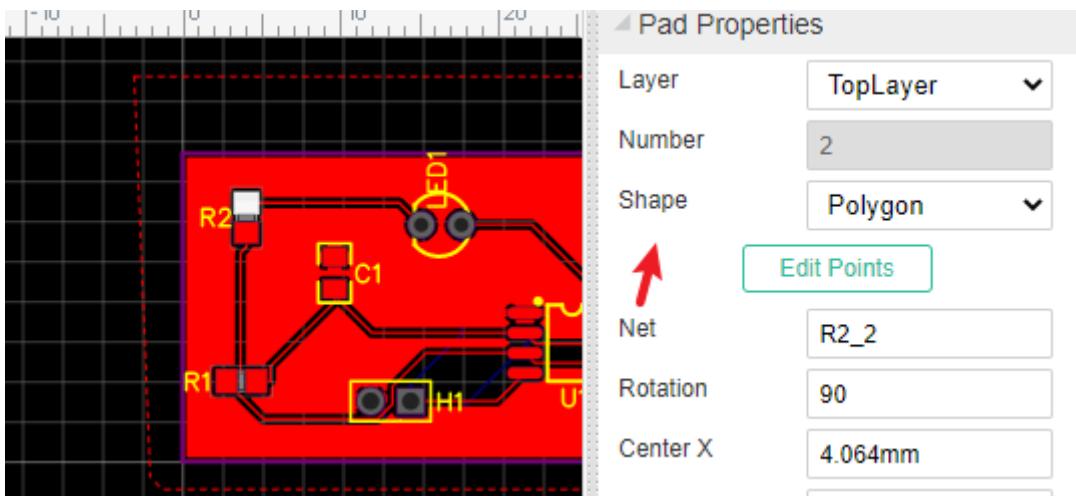
## Notice

- Because of the browser's performance issue, EasyEDA doesn't support the real-time copper pour, after PCB modifying, please rebuild copper area via Hotkey `Shift+B`.*
- EasyEDA doesn't support click the copper zone, you need to click the copper outline to select it.*
- The copper filled data is stored in the client or browser(that is because some copper filled data is too large to save at server), and the copper area outline data is stored in the file. Therefore, when the PCB is opened for the first time, the copper area filled data will be automatically pouring and saving at local, and the second time the PCB is opened, the filled data will be automatically loaded from the local storage. When you need to draw the forbidden copper-laying area, please use the "No Solid" property of "Fill Type" to cutout the copper area and rebuild it, do not use the operation of drawing the area with wires or circles and then removing the wires or circles to create the forbidden copper pour area.**

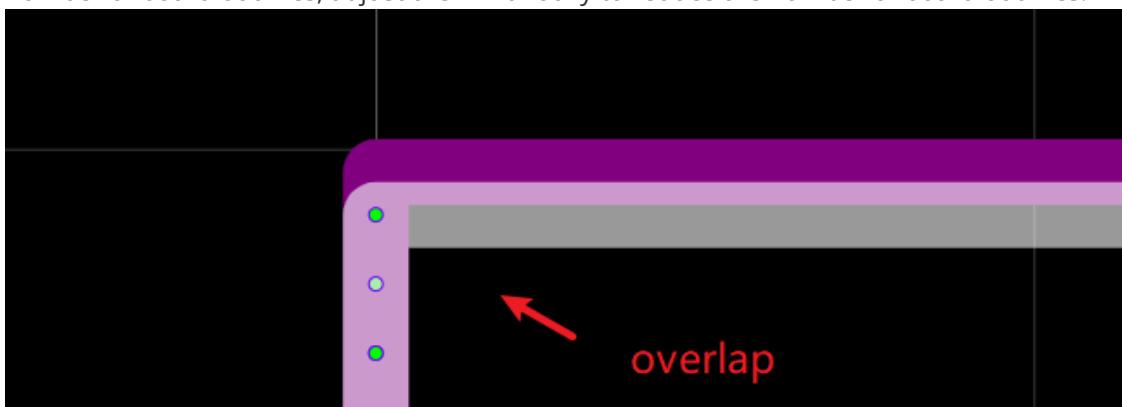
## FAQ

### Why sometimes it takes a long time to copper pour

- Check that the PCB has a large number of polygon pads, which generally appear in the PCB imported Altium Design files, and if so, manually modify them to Round or Rectangle pads.

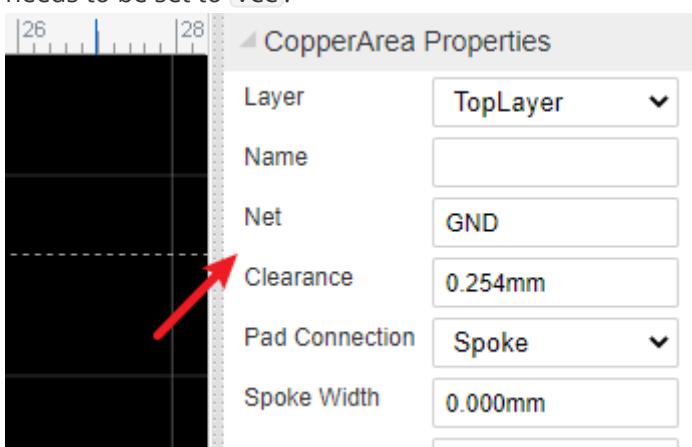


2. Check if there are a large number of wire arcs, generally appear in the imported Altium Design PCB, Altium Design picture is a large number of track segments combined, need to be manually removed.
3. Check that the board outline is complicated, with overlapping board outlines, or a large number of board outlines, adjust them manually to reduce the number of board outlines.



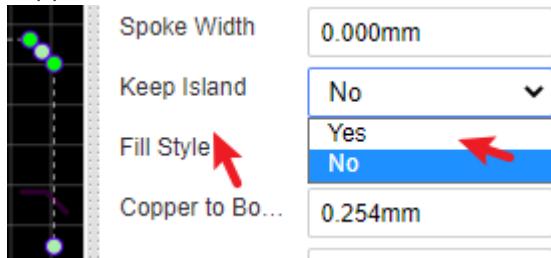
#### Why did I not show the copper fill after copper poured

1. Your copper area net must have the same pad or via semen as the current layer, otherwise it will be considered an island to be removed. Click on the copper wire frame to modify the net in the property panel on the right. For example, your pad net is `vcc`, you lay copper net needs to be set to `vcc`.

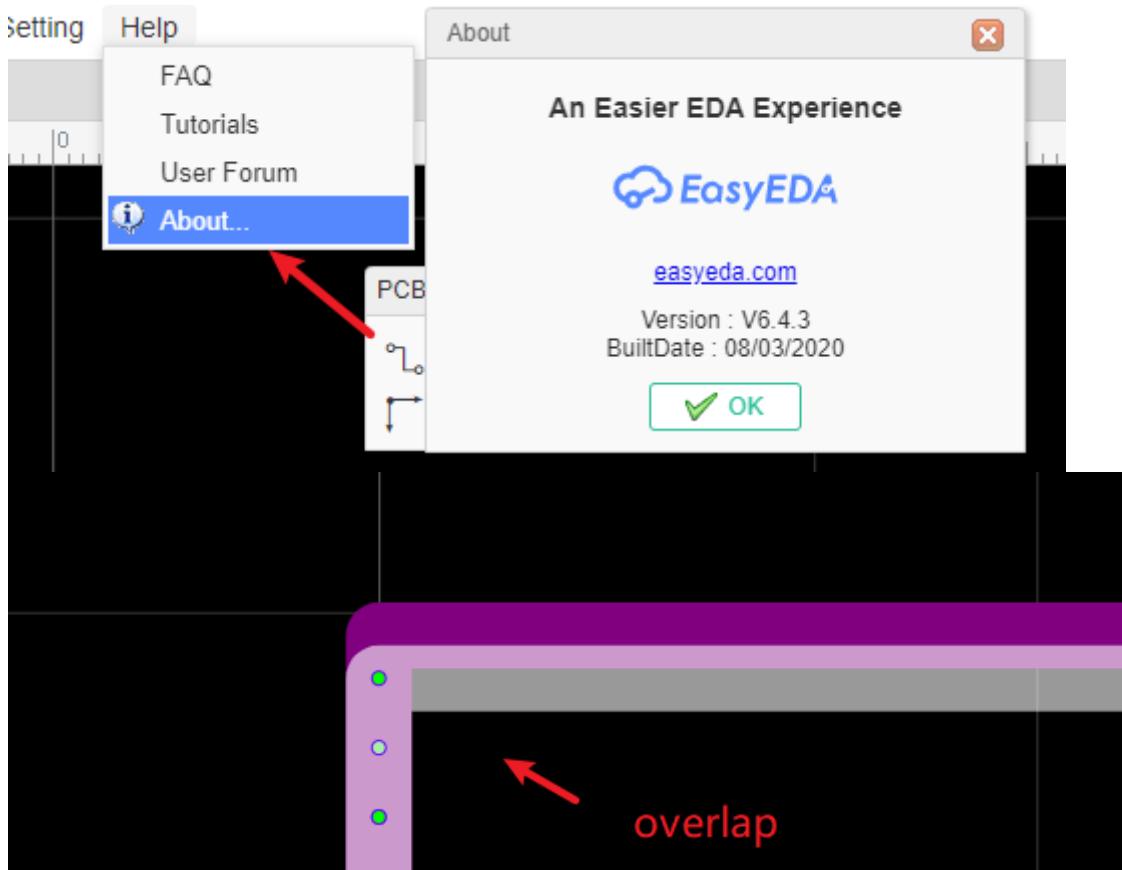


2. If you don't change the copper area ne, you can click on the copper outline and modify the property "Keep Island" to Yes in the right property panel.  
The copper area logic of the EasyEAD is based on whether there is a connection or not to decide whether it is an island, and if there is no element connection to the same net, the

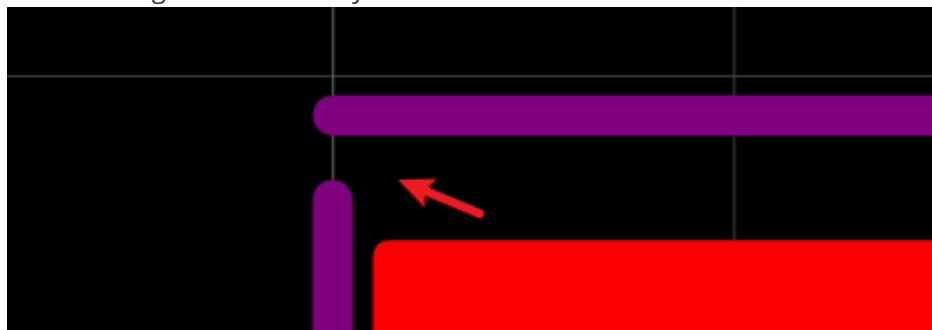
copper area will be considered an island.



3. Check that the editor version is 6.3 above, 6.3 PCB board open in version 6.2 can not properly copper pour. Please CTRL+F5 refresh editor page upgrade to 6.3, if it is true that can not upgrade to 6.3, you must remove the copper area and redraw.



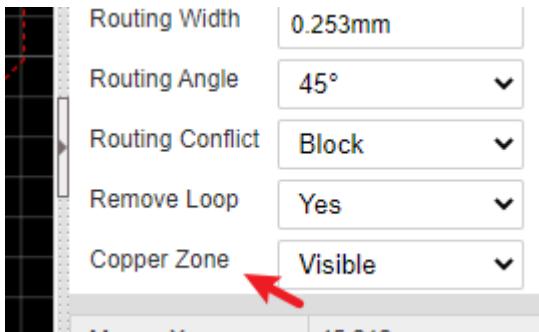
4. Check that the board outline is closed and that endpoints need to be closed between the tracks, and that there are overlapping segments of the board outline (usually inside the imported PCB). Once you can hide all layers, only the board outline layer view is displayed, and each segment is carefully examined.



5. Check that the copper area property is set to type No Solid and needs to be set to Solid or Grid.



6. Whether to make the copper area invisible, on the right side of the canvas, set the copper zone to Visible.

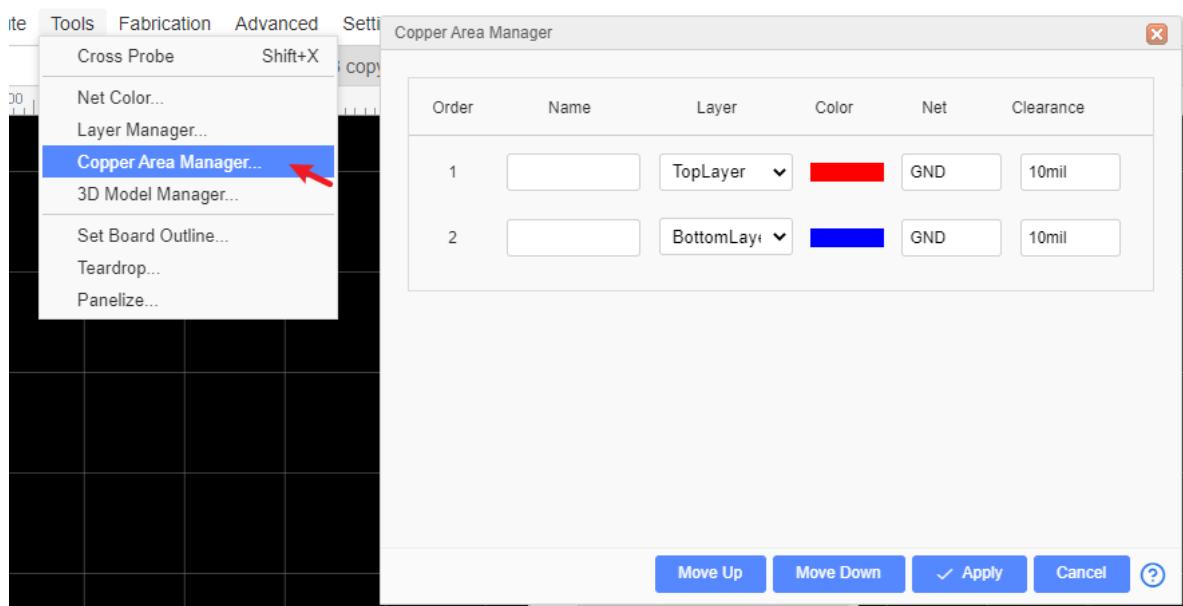


7. Still unable to copper pour may be an editor bug, please contact us.

## Copper Area Manager

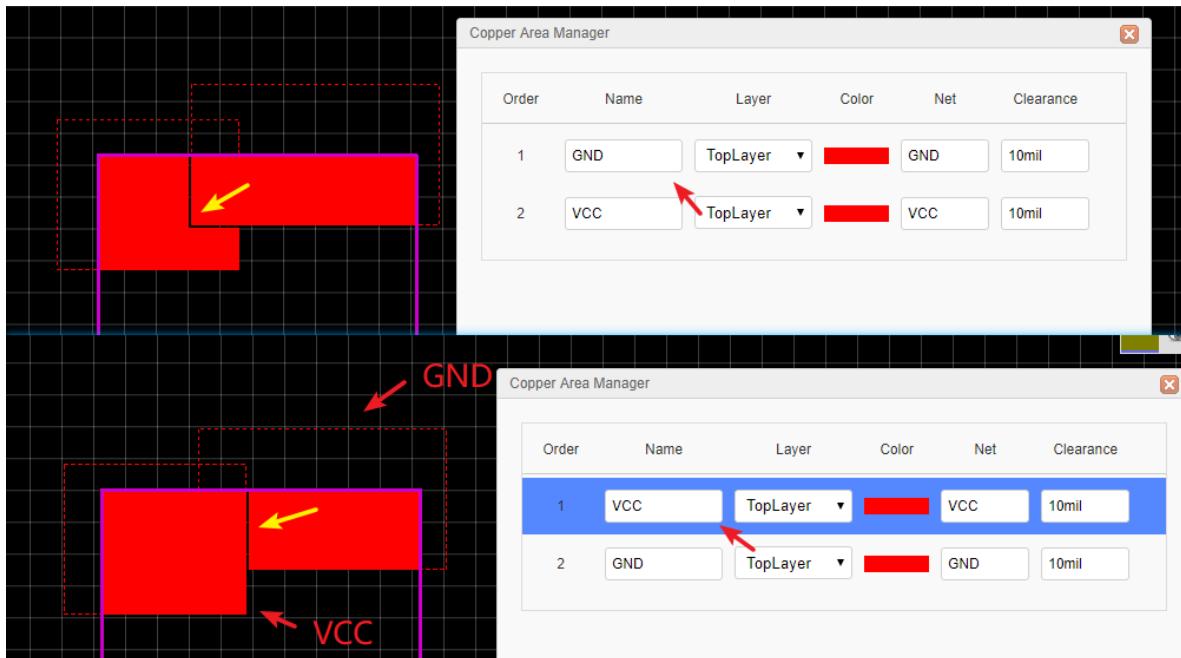
EasyEDA support copper area manager now, you can set the copper order and apply, the forward copper area will be poured first.

Via: Top Menu - Tools - Copper Area Manager



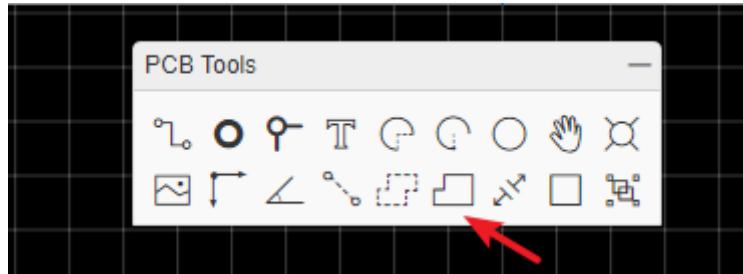
For example:

The GND on the top and VCC on the top, you can see the clearance is different.



## Solid Region

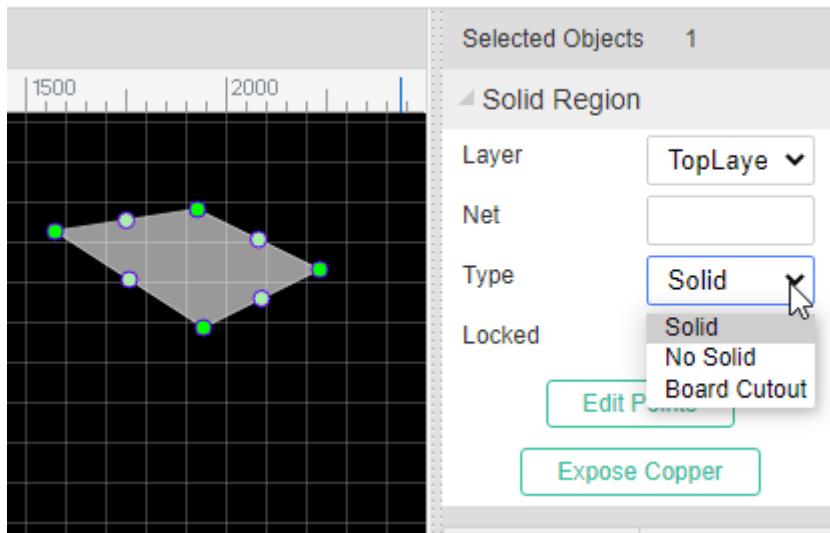
EasyEDA has added a new tool Solid Region for PCB design



This is a very useful, quick way to connect Pads. You can draw a Solid Region to include all of these pads with same net name, then set the region to the same net name as the pads. It is like Copper Area but easier to use for small areas. To use Solid Region like this, set the Type attribute (in the right hand Properties panel) to Solid.

When you drawing the solid region, you can use the hotkey `L` and `space` to change the route type(Arc, 90 degrees, 45 degrees, Free Angle), just like the track routing.

When you finish drawing, you can click the solid region and change its attributes at the right-hand panel.



- **Layer:** Solid Region support many layers, you need to enable the layer at the Layer Manager first.
- **Net:** When change to top or bottom or other inner signal layer, the solid region can be set a net to connect other objects. Sometimes, you can use solid region to make the copper instead of "Copper Area".
- **Type:** Solid,Board Cutout,No Solid ,
  - **Solid:** It will fill the solid area.
  - **No Solid:** It will cutout the area such as copper area. **Notice, if you cutout a copper area, the solid region's net must different than copper area's net.** After setting to this option, you need to rebuild the copper area with SHIFT+B.
  - **Board Cutout:** you can use this feature to create a slot hole(Non Plated Through Hole).



The outline of the solid region can not be self-intersection, when it happens, please delete the self-interaction point at "Edit Points".

## Distribute Array

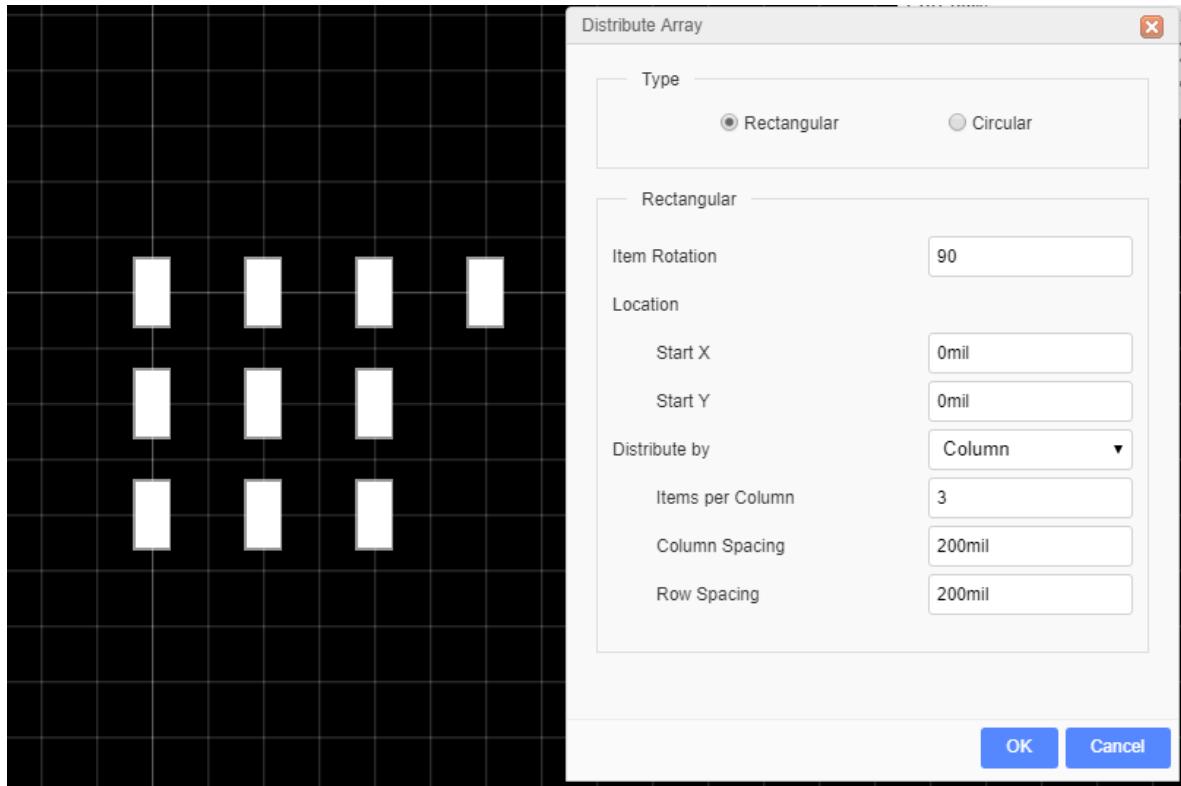
EasyEDA doesn't support the paste array, but EasyEDA provide a powerful function - Distribute Array. It works at PCB, Footprint, PCB module.

Via: Top Menu - Align - Distribute Array

How to use: Selected the objects - Click the Distribute Array - Set the parameters, and apply.

**Rectangular:**

- Item Rotation: The rotation of the item, if you set 30 degrees, all selected item will rotate 30 degrees.
- Location: The location for the first item to place. on the left-top corner of the items.
- Distribute By:
  - Column: From top to bottom, and then from left to right, like word N.
  - Row: From left to right, and then from top to bottom, like work Z.



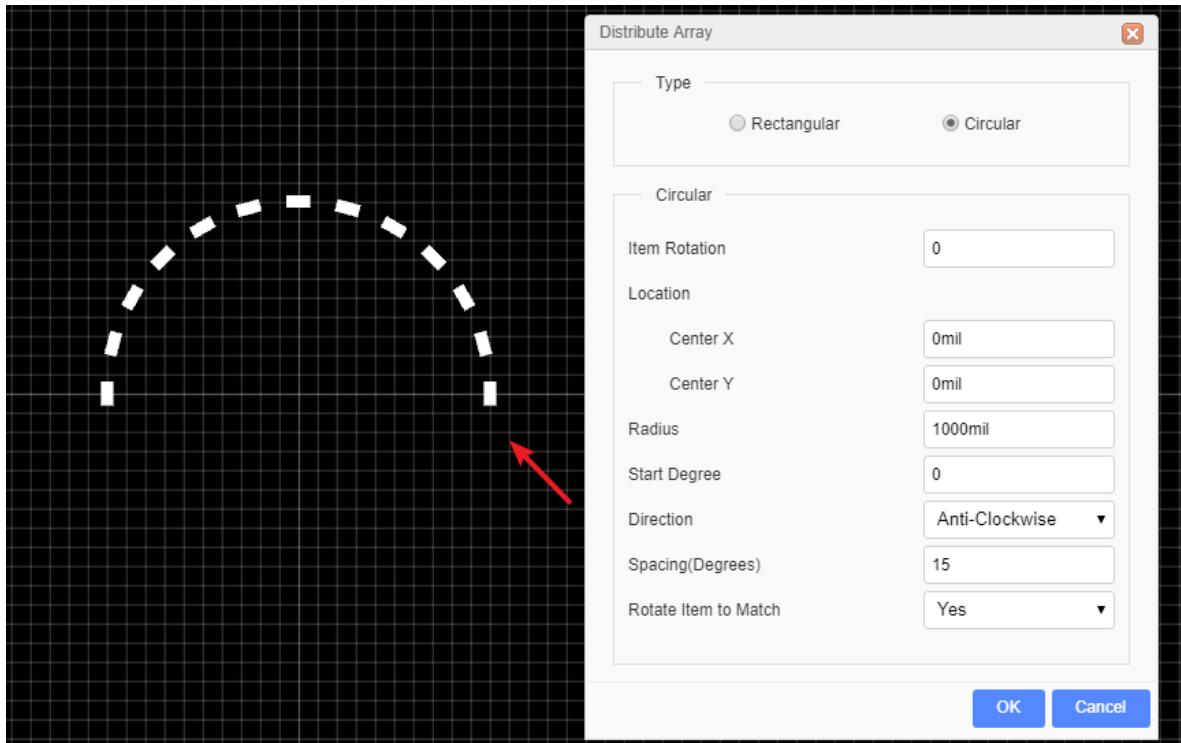
### Circular:

- Item Rotation: The rotation of the item.
- Location: The center location of the circle.
- Radius: The radius of the circle.
- Start Degree: The start degree of the first item. 0 degree is on the middle of the right side.
- Direction: The forward direction of the items. Anti-Clockwise or Clockwise
- Spacing(Degrees): The spacing between each item.
- Rotate Item to Match: If choose Yes, the item will rotate to match the circle. When setting Yes, the item actually rotation will be "Item Rotation + Spacing".

Before Rotate Item to Match:



After Rotate Item to Match:

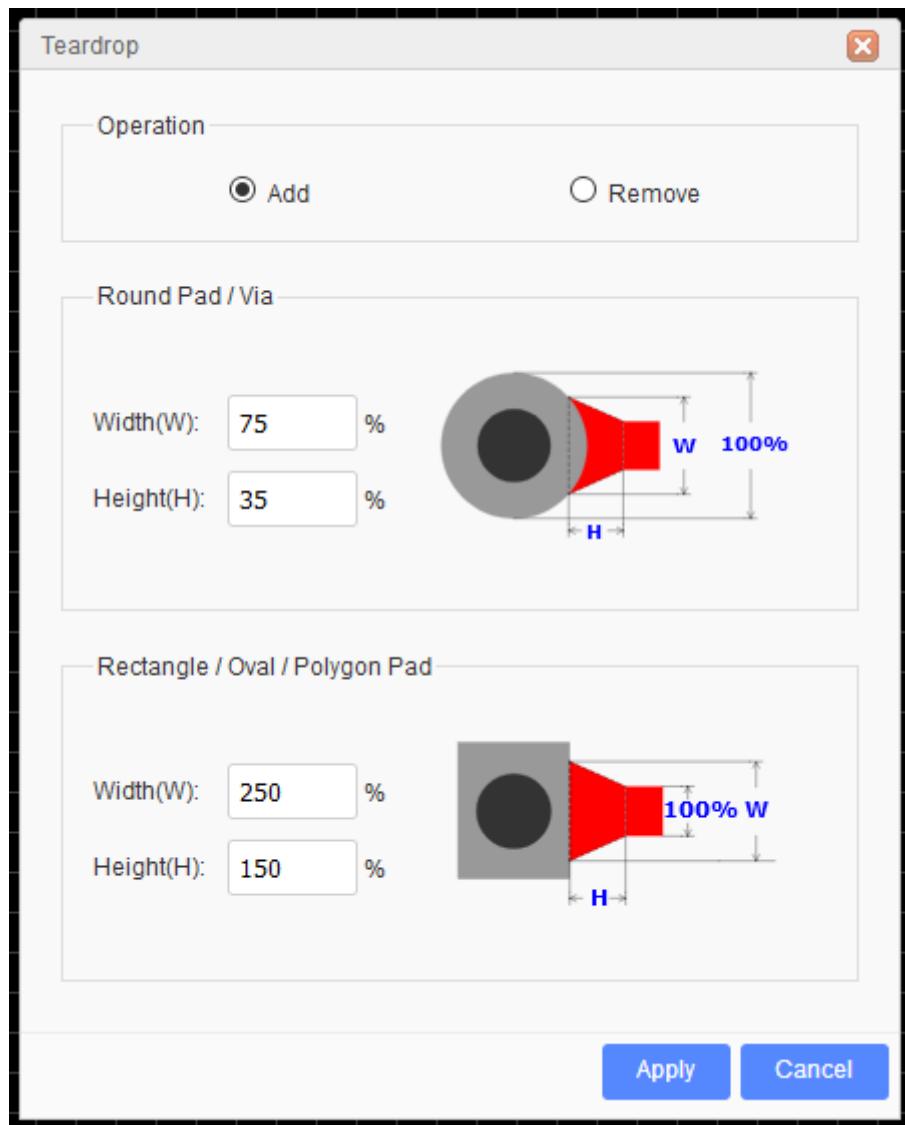


## Teardrop

---

Via: Top Menu - Tools - Teardrop

You need to set the parameter first, and then Apply.



When delete the track, the teardrop will be deleted too.

If the teardrop detect the DRC errors while generating, this teardrop will not generate.

At present, doesn't support add teardrops for one part.

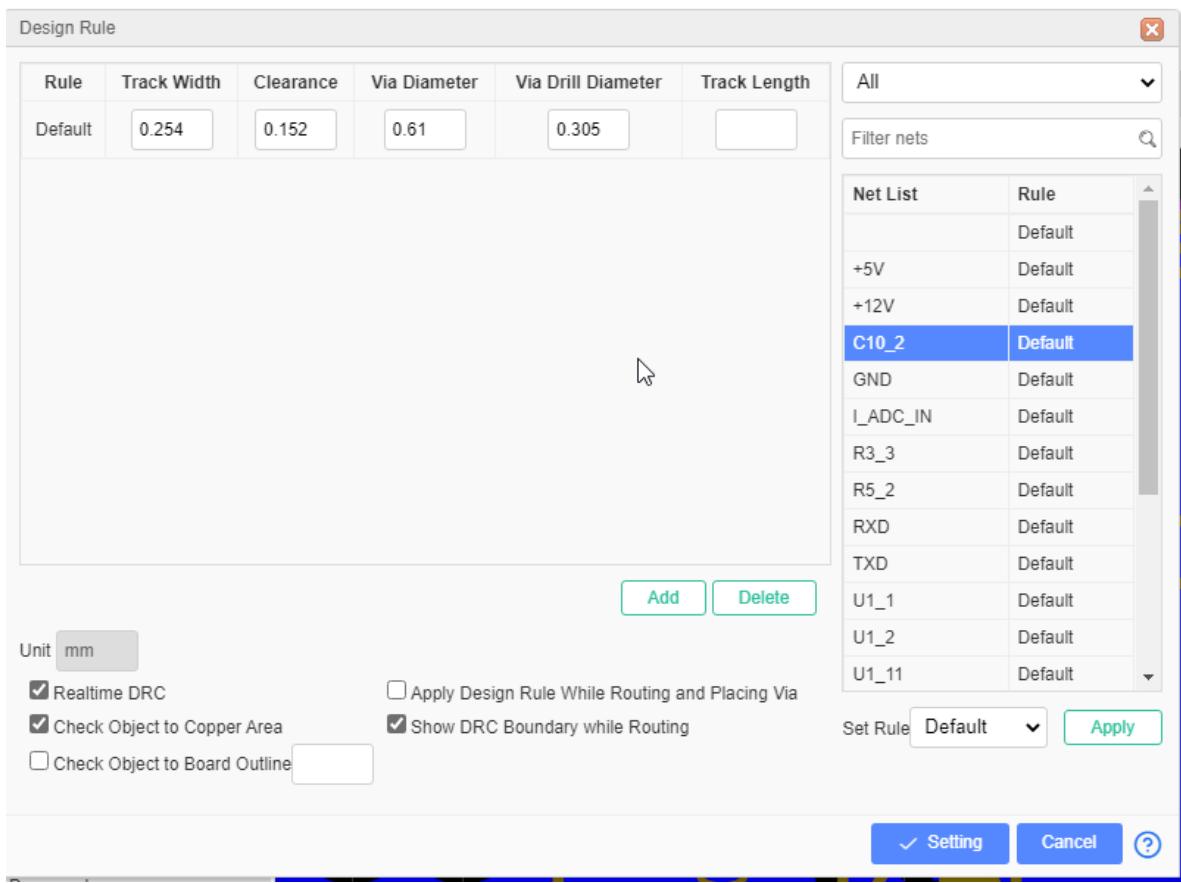
In fact, the teardrop is a Solid Region, when you select it, you can modify its attributes.

## Design Rule Check(DRC)

EasyEDA provides a real time DRC(Design Rule Check) function. This is a big feature of EasyEDA. It is hard to fix DRC errors after laying out the PCB. Now EasyEDA will let you know the error in routing. You will find an flag to mark the error.

## Design Rule Setting

Via at: **Tools > Design Rule...**, or Via: **right-click the canvas - Design Rule...** to open the **Design Rule** setting dialog:



The unit follow the canvas unit.

**Rule:** The default rule named "Default", you can add the new rule you can rename and set parameters for it. Each net can be set a rule.

**Track Width:** Current rule's track width. The PCB track width can not less than this value.

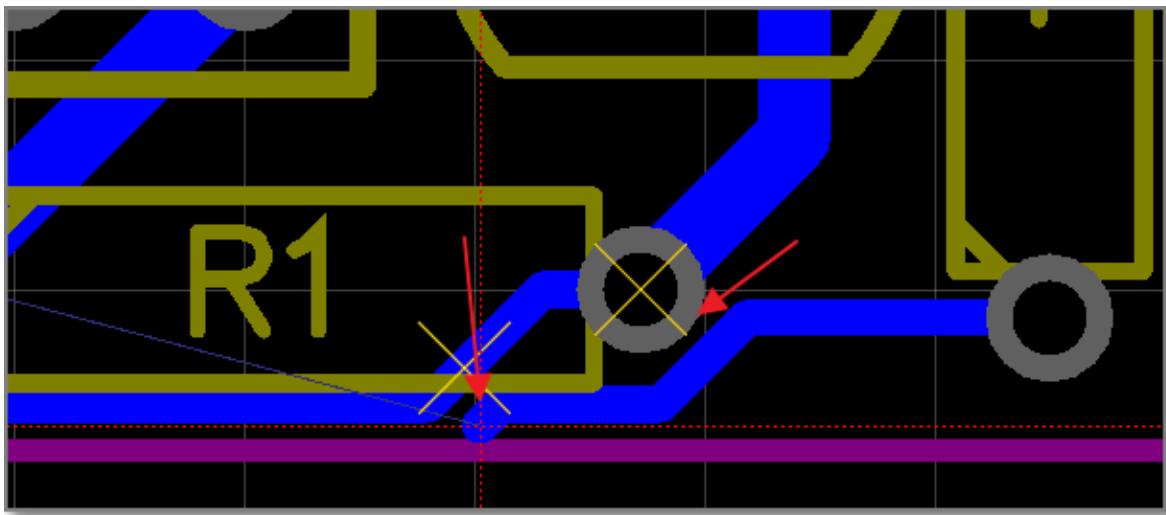
**Clearance:** The clearance of different objects which have different net. The clearance of the PCB can not less than this value.

**Via Diameter:** The via diameter of current rule. The via diameter of the PCB can not less than this value. Such as the Hole/Multi-layer Pad's diameter.

**Via Drill Diameter:** The via drill diameter of current rule. The via drill diameter of the PCB can not less than this value.

**Track Length:** All track length of current rule. The length of tracks belong to a same net should not be longer than this value. Including the arc length. When the input box is empty the length will be unlimited.

**Realtime DRC:** After enable, when you routing the DRC will checking all the time, when appear the error the canvas will show the "X" marking.



**Check Object to Copper Area:** Check the clearance of the objects to copper area. If you disable this option, you must rebuild the copper area before generating the Gerber with SHIFT+B.

**Check Object to Board Outline:** When you enable, you can set a value to check the clearance of the objects to board outline.

**Apply Design Rule while Routing and Placing Via:** When you routing and placing a new via, them will follow the design rule to set them width and size.

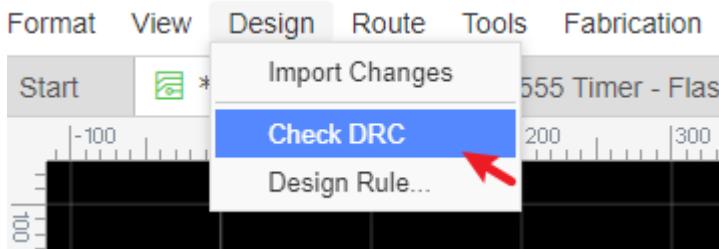
**Show DRC Boundary while Routing:** When routing you will see a outline around the track. Its diameter depends on desgin rule.

## Set Rule for a Net

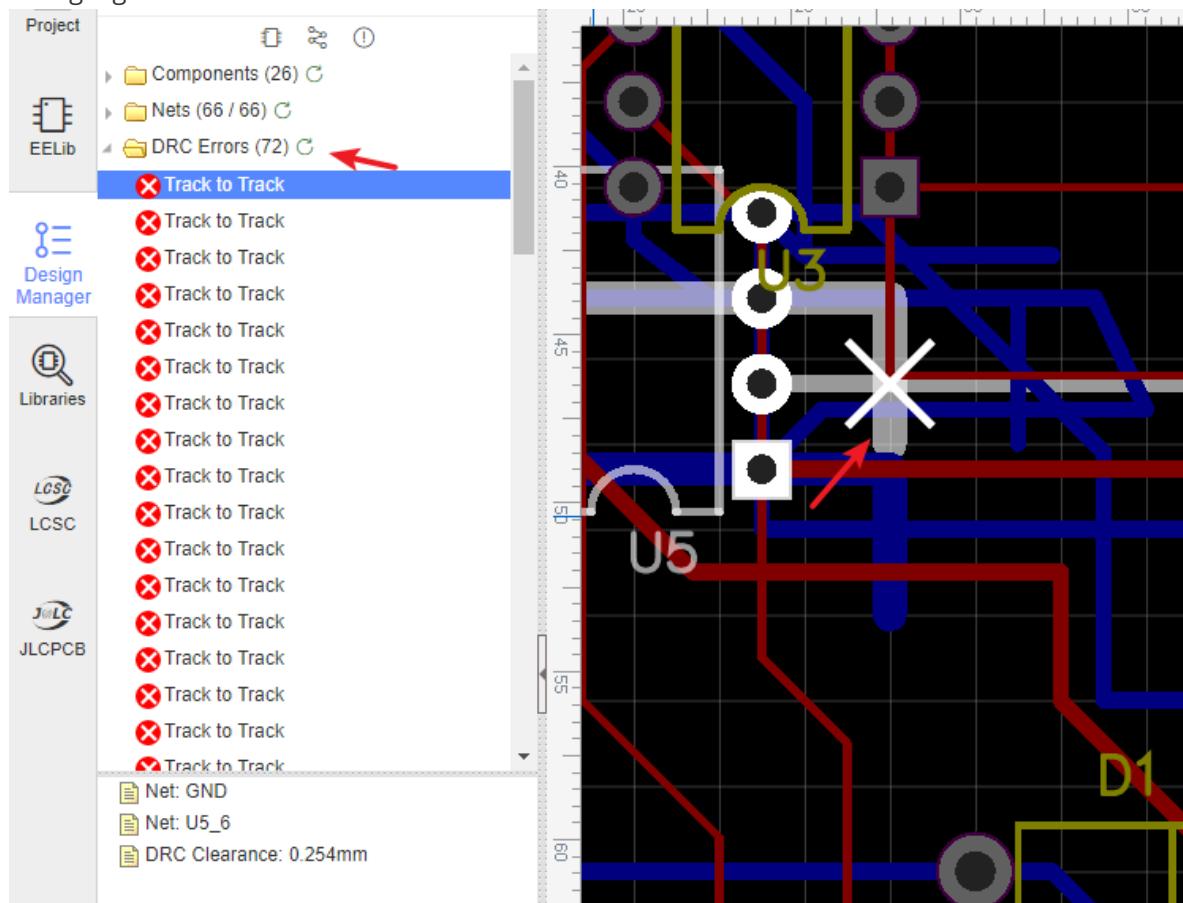
1. Click the "new" button to create a rule, or use the default rule
2. Select one or more networks on the right, support holding down the CTRL key for multiple selection, and also can perform keyword filtering and rule classification filtering
3. Then select the rule you want to set in the "set rules" section below and click the "apply" button. The network applies the rule.
4. Click the "Settings" button to apply the rule.

## Check the DRC Error

Via "Design Manager - DRC Error" or "Top Menu - Design - Check DRC", click the refresh icon to run the DRC. If your PCB is a big file, and have the copper area that will take some times to check the DRC, please wait a while.



After checking, you can view all the error at the "DRC Error", click the error the related objects will be highlighted.

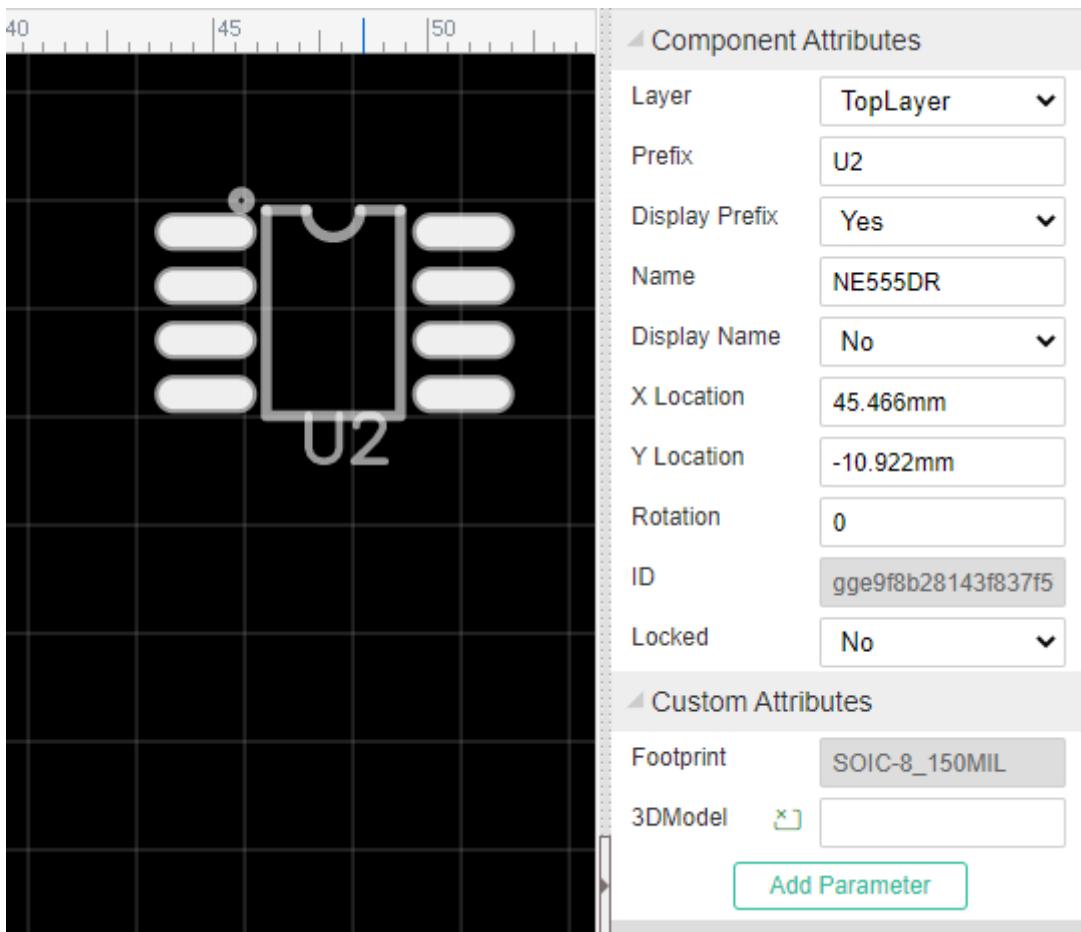


#### Note:

- When you convert a schematic to PCB, the real time DRC is enable. But in the old PCB, the real time DRC is disable. you can enable it in the image as above.
- Design rule checking can only help you find some obvious errors.
- The color of the DRC error can be set in the layer manager.

## Footprint Attributes

When selecting a Footprint, you can find its attributes at the right hand Properties panel.



**Prefix:** It is same as the schematic. If you move the prefix too far away from the footprint, it will be dragged back to the footprint when you open the PCB again, if you don't need the prefix please set the prefix display as No.

**Layer:** You can set a footprint to be on the TopLayer or BottomLayer, it same as board side.  
 \*Note: The footprint mirrors when it swapping layers. it doesn't support to mirror at current layer.\*\*

**X-Location and Y-Location:** Moves the origin of the footprint to a precise position.

**Rotation:** Rotates the footprint about its origin over the range from 0o to any angle in 10 steps (visually of course multiples of 360o will appear identical).

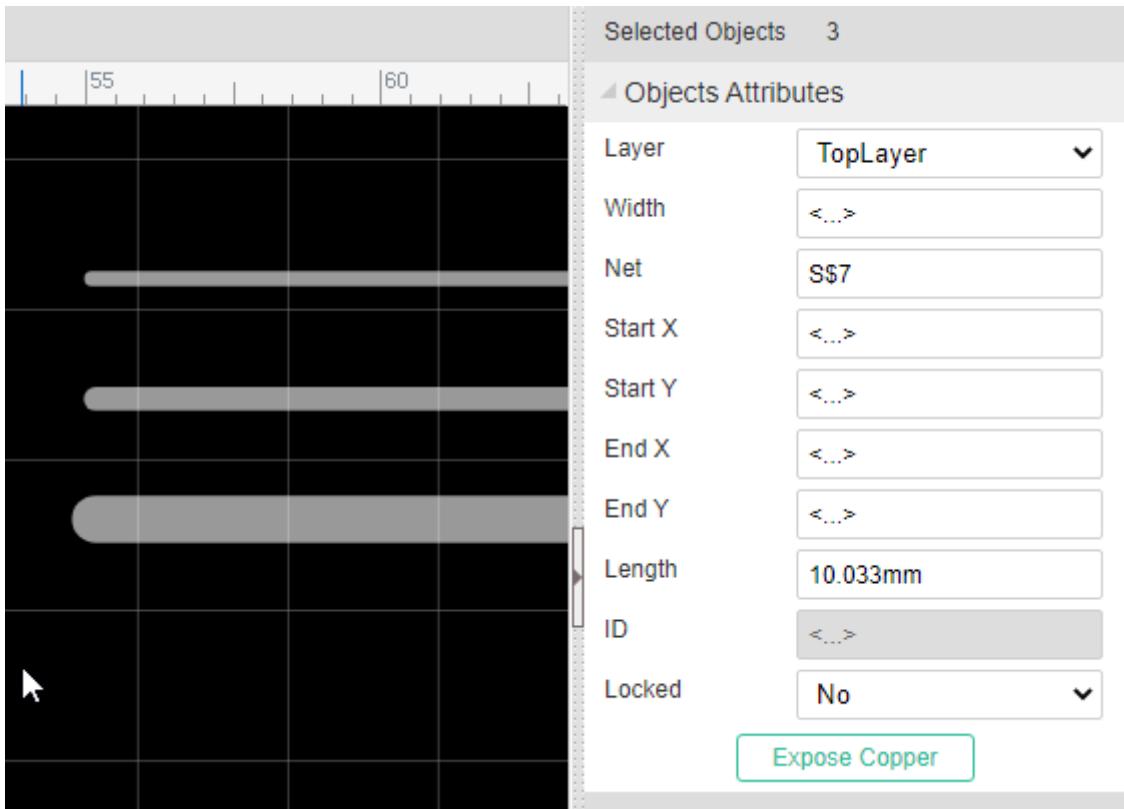
**ID:** EasyEDA will assign a unique ID for each footprint automatically, you can't modify it.

### Change Attributes in Batch on PCB Editor

Sometimes, we need to change some attributes of multiple objects together, such as the track width, hole size and font size.

Now, you can select them and do some changes.

Taking the track for an example. If you select 3 tracks, now you can change their `width`, `Layer`, `Net` together. The difference property values will combine as `<...>`, change it directly will apply to all selected objects.



# Design Manager

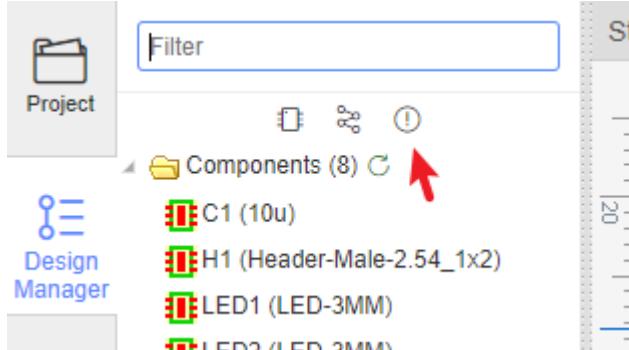
Just like Schematic's Design Manager, PCB's Design Manager can be found via:

**Left Navigation panel > Design**

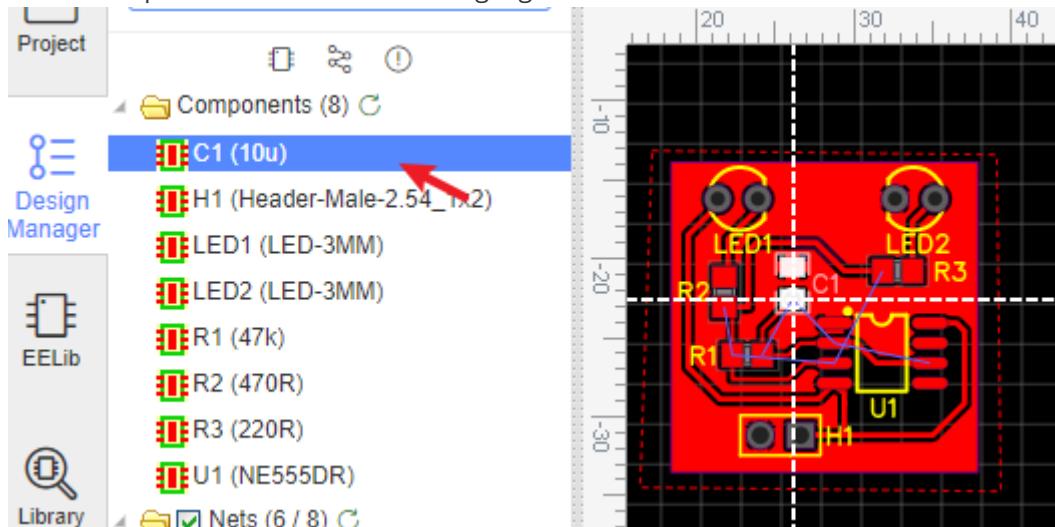
or just press the **CTRL+D** hotkey to open the Design Manager dialog.

In this dialog, you can:

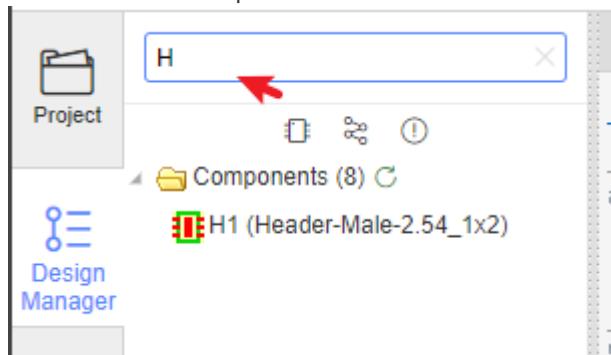
- Click the icon to jump to folder.



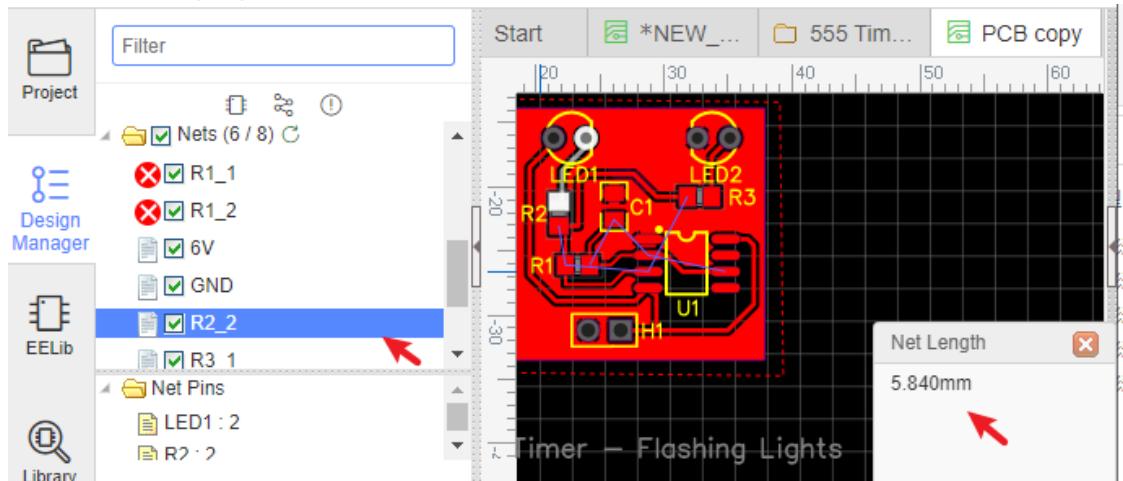
- Click a component/Net/DRC Error to highlight it.



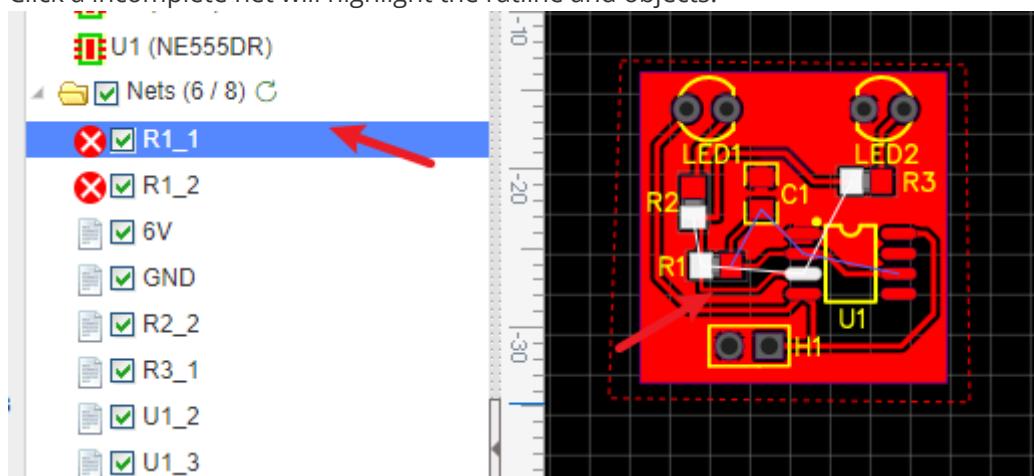
- Filter to find a component or net.



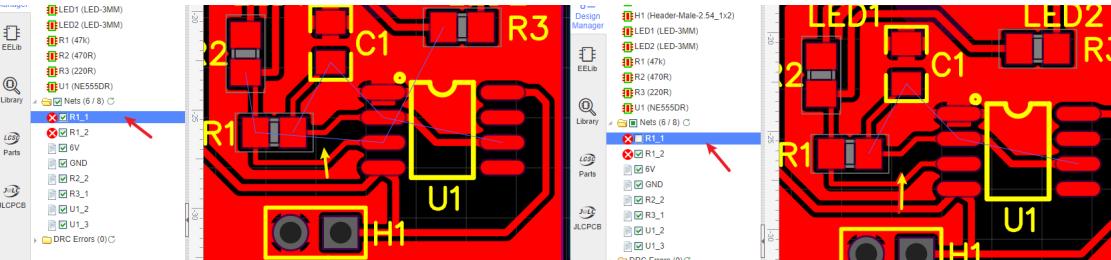
- Click a net to highlight the tracks/vias with the same net.



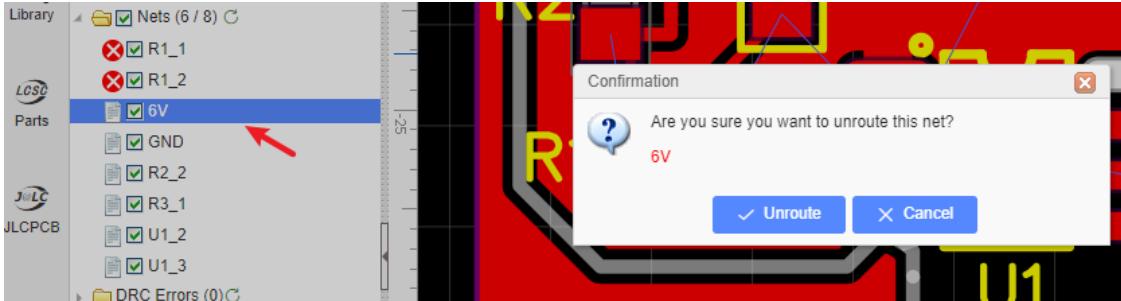
- Click an incomplete net will highlight the ratline and objects.



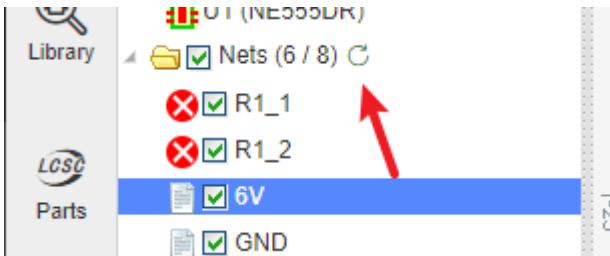
- Check/uncheck the net to show/hide the ratline of the net.



- Double click the net to remove all of the tracks and vias with the net name. If you want to reroute a net, this is the recommended method to use to un-route it first.



- Click the refresh icon the refresh the list.



Notice:

- Design Manager list doesn't support to refresh automatically, you must click the refresh icon manually.

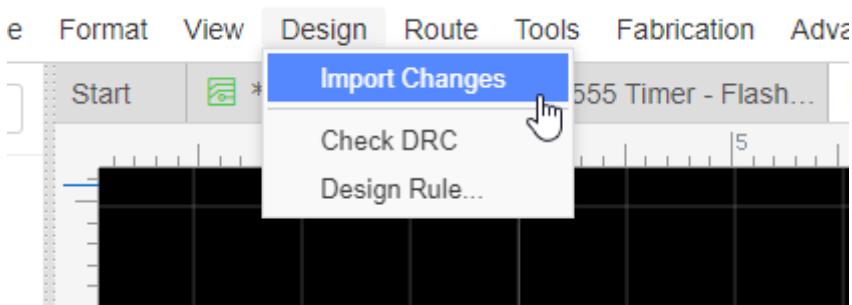
## Import Changes

### Import Changes

Sometimes, while working on a project, you need to make changes to the schematic and then update your board, to incorporate them.

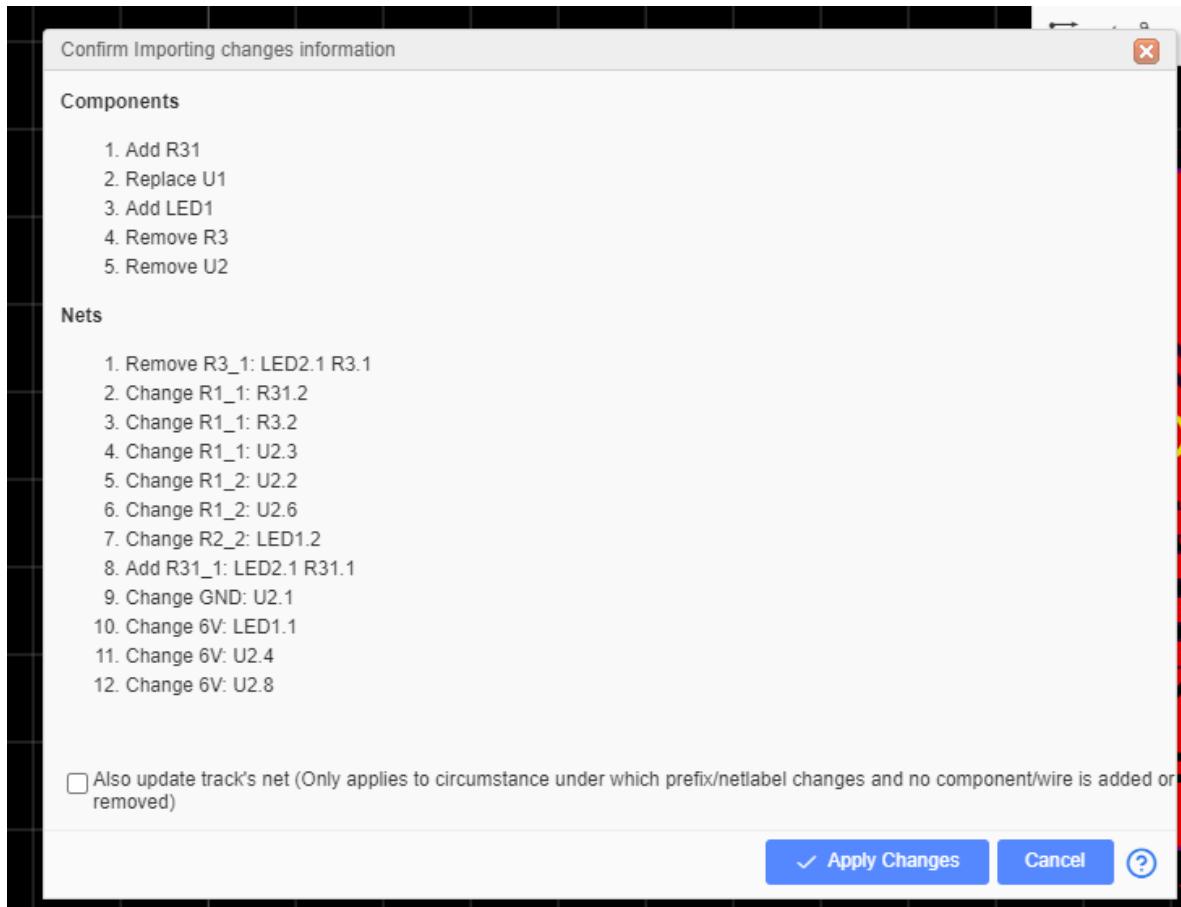
It's easy to do this with EasyEDA.

Go to the **PCB Editor**, via: **Top Meun > Design > Import Changes**



If there are some errors at schematic, such as prefix duplicated, no footprint, it will pop up notice dialog, the more information please refer: [Schematic - Convert to PCB](#)

If no errors, you will get a "Confirm Importing changes information" dialog:



If you are happy with your changes, just click the Apply Change button.

If you want to update the PCB tracks net same as the schemtiac, you need to enable "Also update track's net" option. The editor will update the related track's net depends on the pad's net.

The changes will then be passed into the PCB layout and you can then adjust the tracking to suit.

Notice:

- Because of the net of the schematic is generated after calculating, when you change some netlabel, after Import Changes, the PCB track will not be deleted.
- When enable the "Also update track's net" option, after Import Changes, the related tracks vias will update the net from the pads, there will be some nets changed isn't you want, you need to change them manually, such change prefix, modify the parts connection, delete or add part at the schematic, you can change the tracks net via: right-click the track - click Select menu - Connection, and them all connection will be selected, you can change them net at the right-hand property panel.
- After Import Changes, there are some action can not be undo.

## Panelize

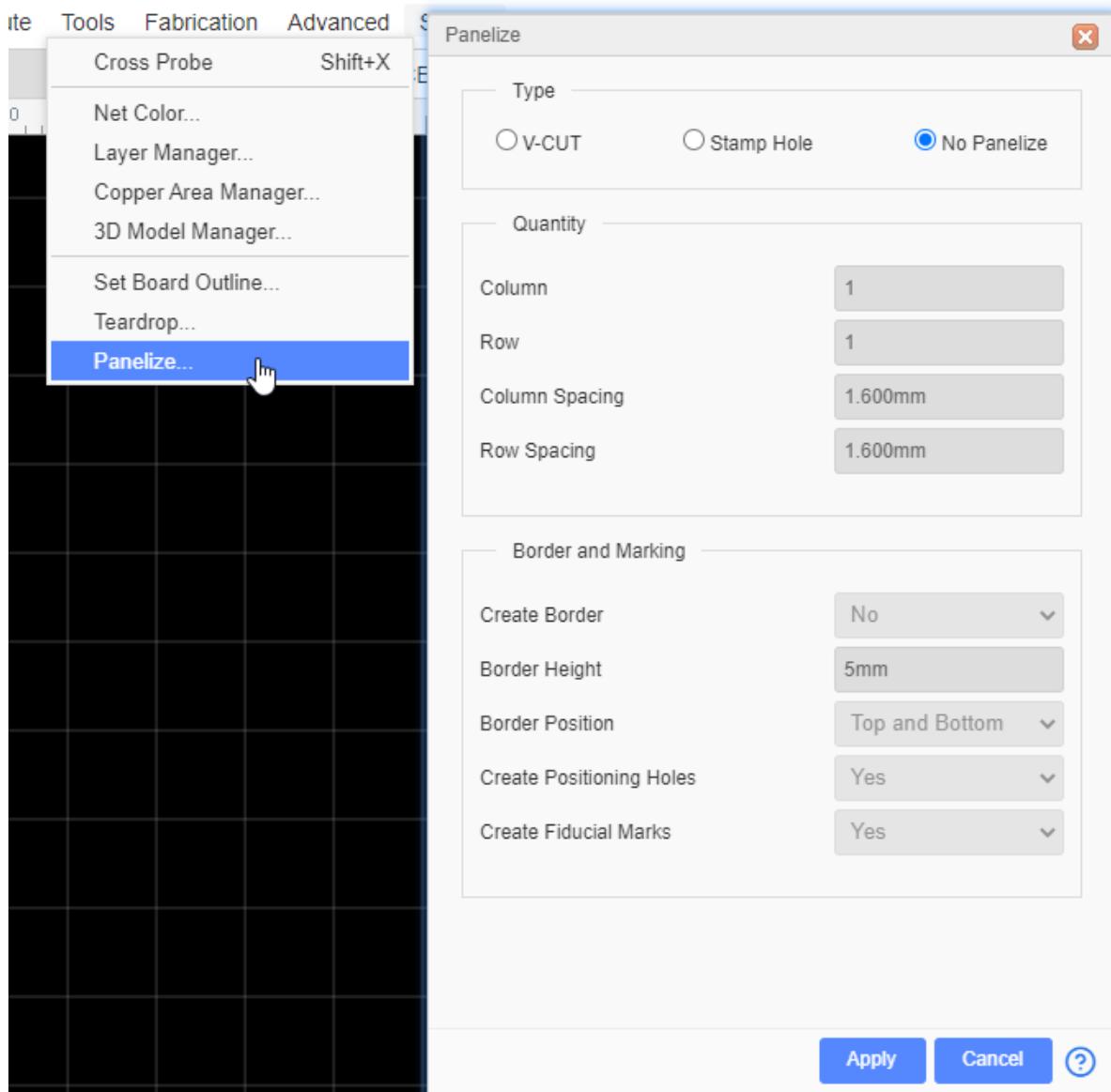
via: Top Menu - Tools - Panelize

## Panelize by Editor

At present, EasyEDA only support to panelize PCB itself, in order to decrease the file size, the panelized file only panelize the board outline.

Normally, all the PCB factory will support this panelized file, if you not sure, you need to contact your PCB factory support.

via: **Top Menu - Tools - Panelize**

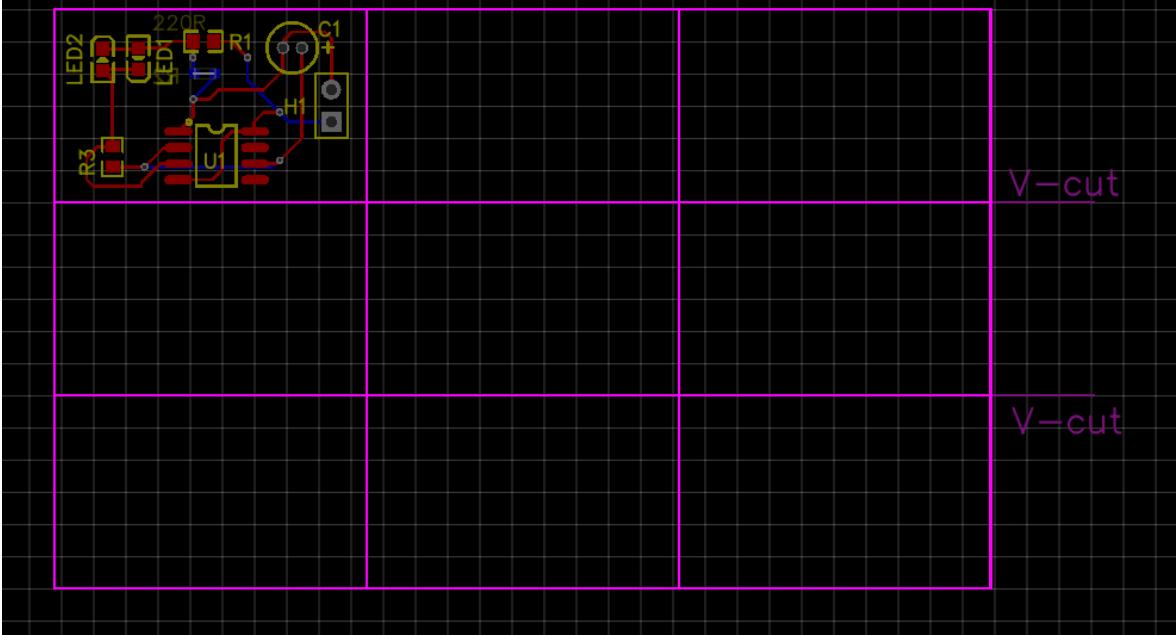


The Border height can not less then 3mm.

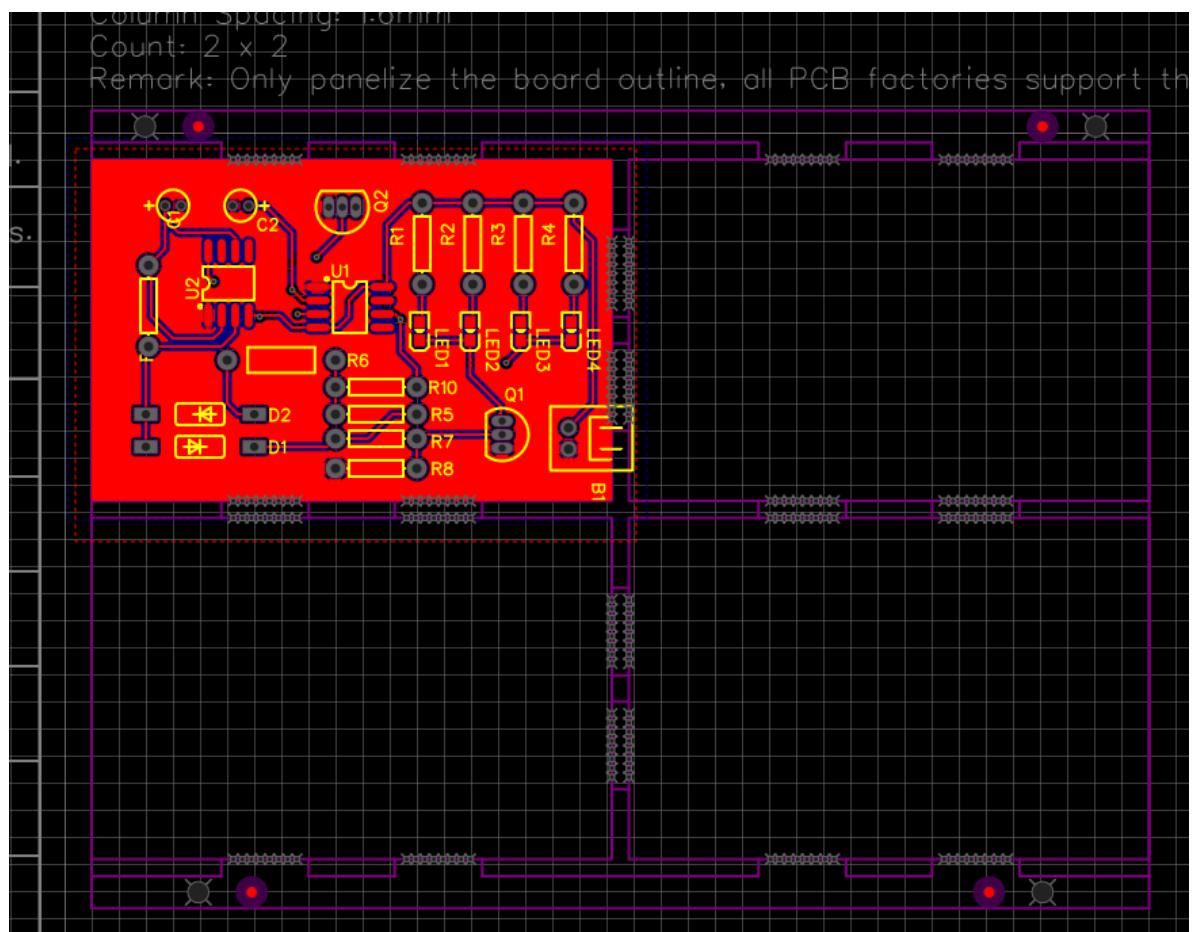
#### V-cut:

If you choose V-Cut, the editor will add the v-cut indication track on mechanical layer.

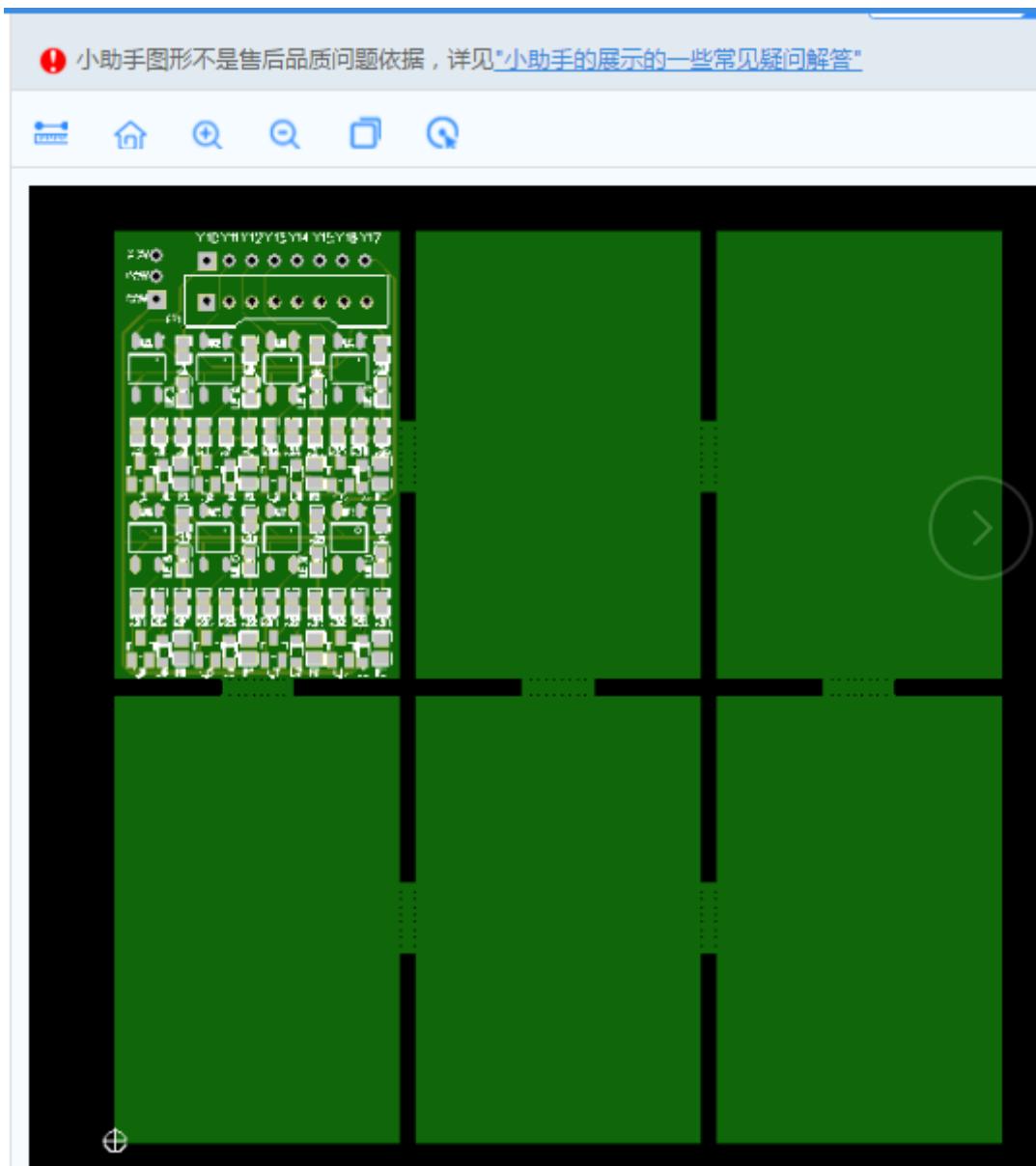
Row spacing: 0.000mm  
Column spacing: 0.000mm  
Count: 3 x 3



#### Stamp Hole:



When you preview the Panelize Gerber at [JLCPCB.com](https://JLCPCB.com), you will get the image like below:



JLCPCB will take care of your design, they know how to do.

## Panelize by Manually

Process:

1. Select the whole board, hotkey `CTRL+A`.
2. Copy the whole board by reference point, hotkey `CTRL+SHIFT+C` or `CTRL+C`.
3. Paste the board via hotkey `CTRL+SHIFT+V`, this hotkey will keep the prefix and hide the ratline layer.
4. Paste repeatedly, after finish, rebuild the copper area with `SHIFT+B`, recommend draw copper area at the end.

### Notice

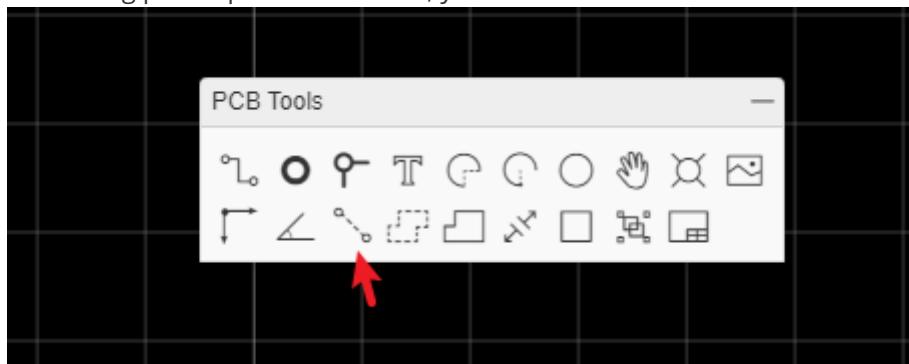
- If the board contains plane layer, it can not be panelized by manually, it will not generate the plane zone as you want.

For some small PCB projects, maybe you don't need a schematic. EasyEDA allows you to lay the PCB directly from the PCB Editor.

1. Start a new PCB
2. add footprints directly from the Footprints from Left Navigation Panel **Library - Footprint**
3. and then just route track for them.

The PCB created by New PCB menu directly, it will hide the ratline layer defaultly.

For setting pad to pad connections, you can check the above **Connect Pad to Pad** section.

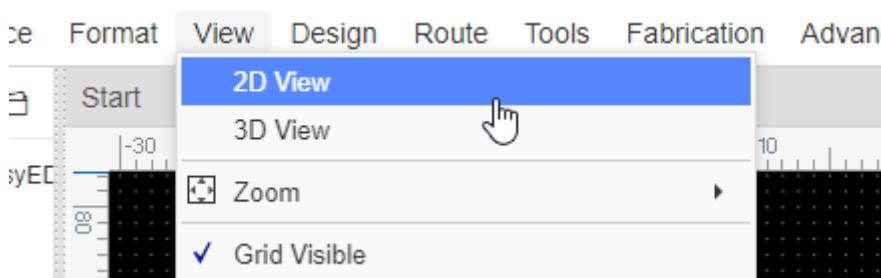


## PCB Preview

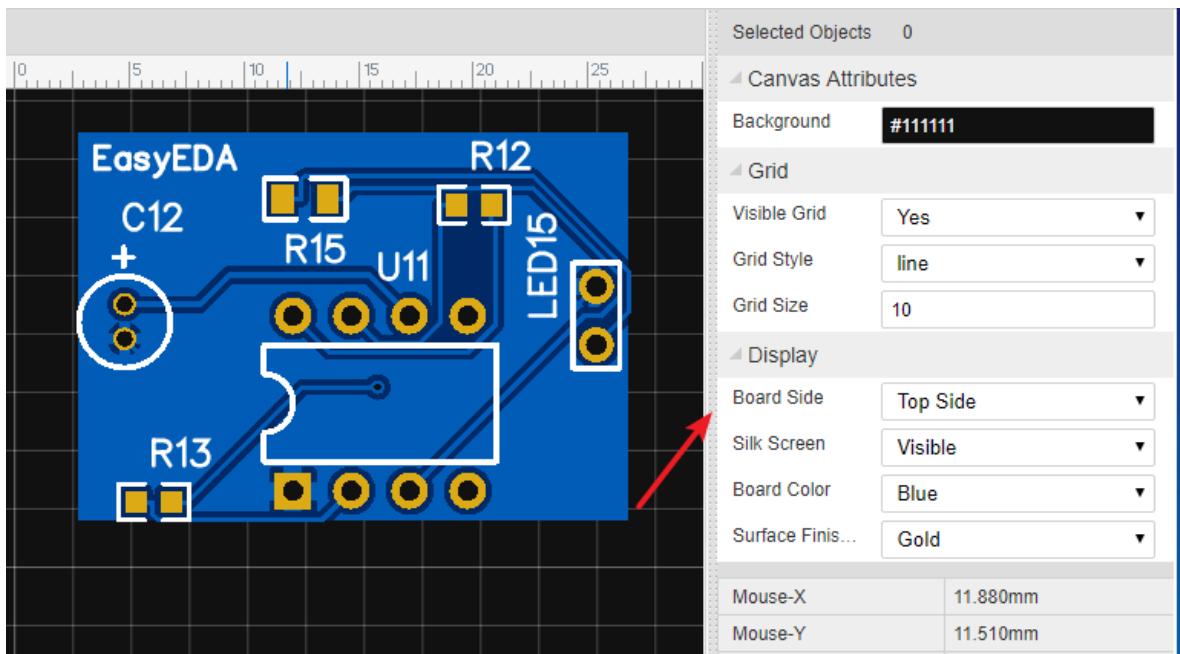
### 2D View

EasyEDA provide a nice Photo View to help you to check the PCB.

Via: Top Menu - View - 2D View.

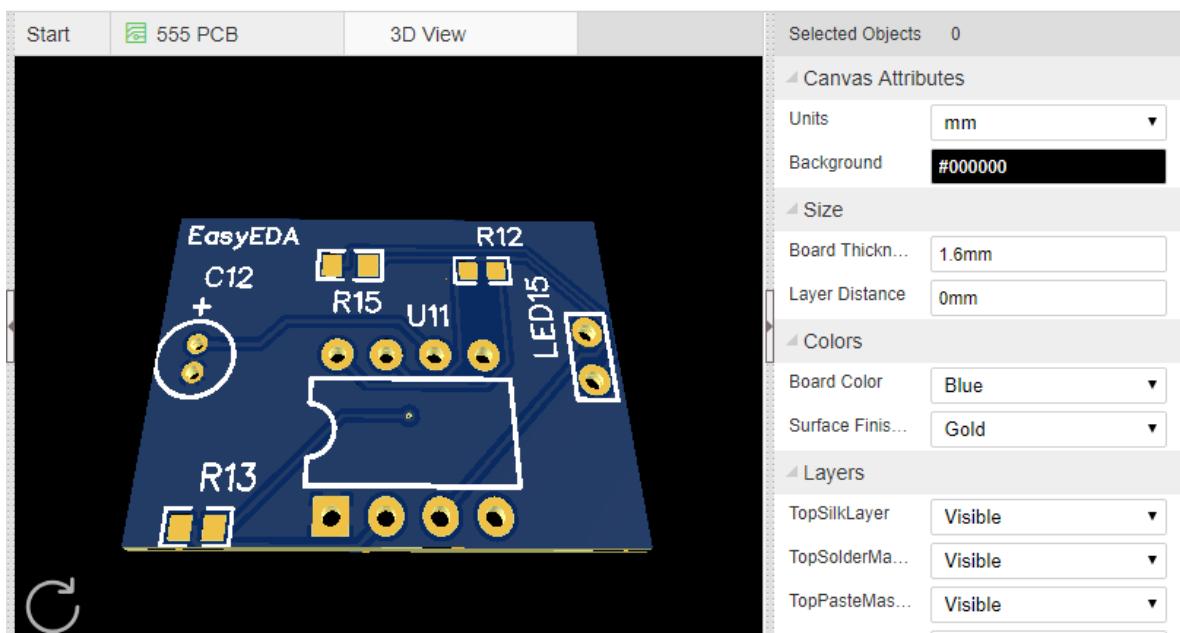


After converting the PCB to Photo View, you can see the result as in the image below.



## 3D View

After click 3D view menu, the server will generate the 3D view file, when the editor loading finish, you will see a pretty cool 3D view.



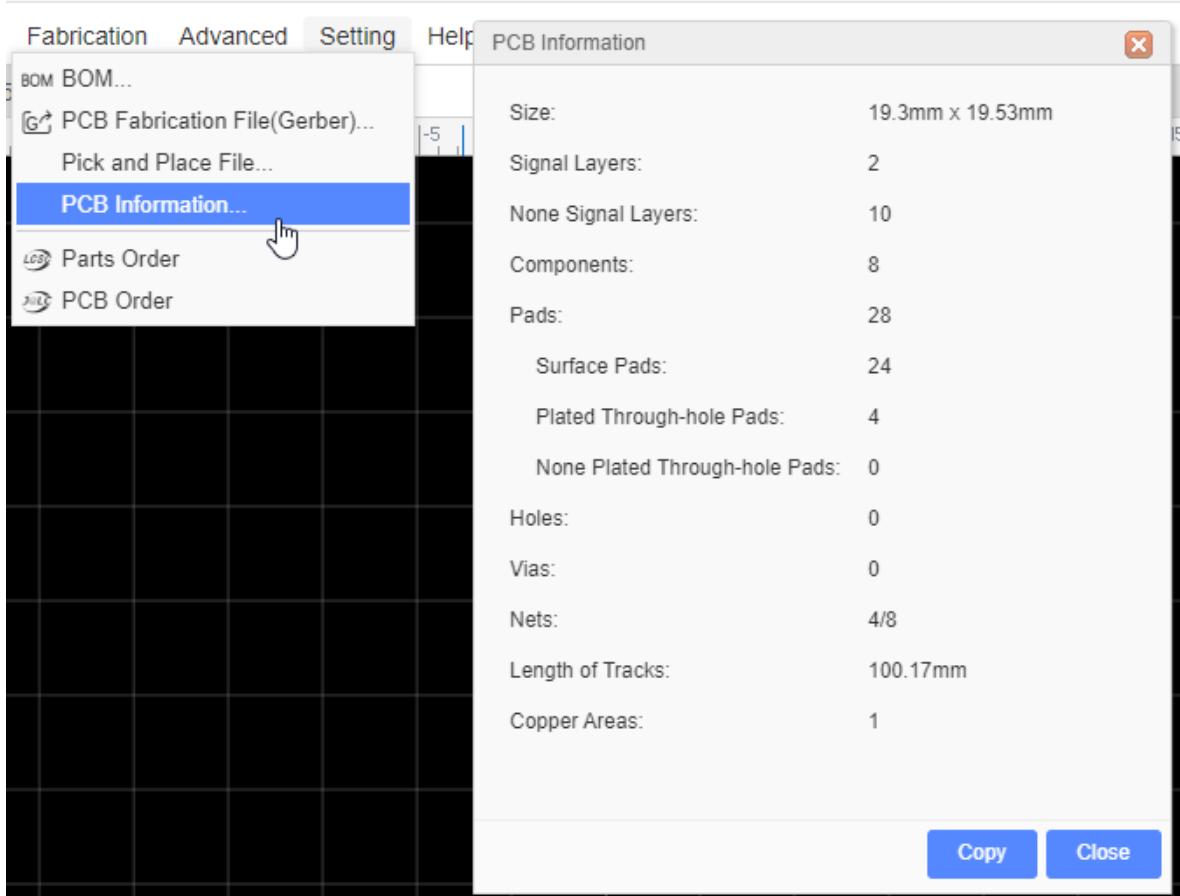
- Change 3D view attributes at the right-hand panel;
- Reset the 3D PCB position at the left-bottom corner icon;
- Keep left-click and drag the canvas can change the view direction;
- Keep right-click and pan can change the 3D PCB position.

3D model view of the component please check "PCB - 3D Model Manager" and "Footprint - Import 3D Model" chapter.

## PCB Information

PCB design information can be easily obtained by checking PCB information.

Entry: Top Menu - Fabrication - PCB Information



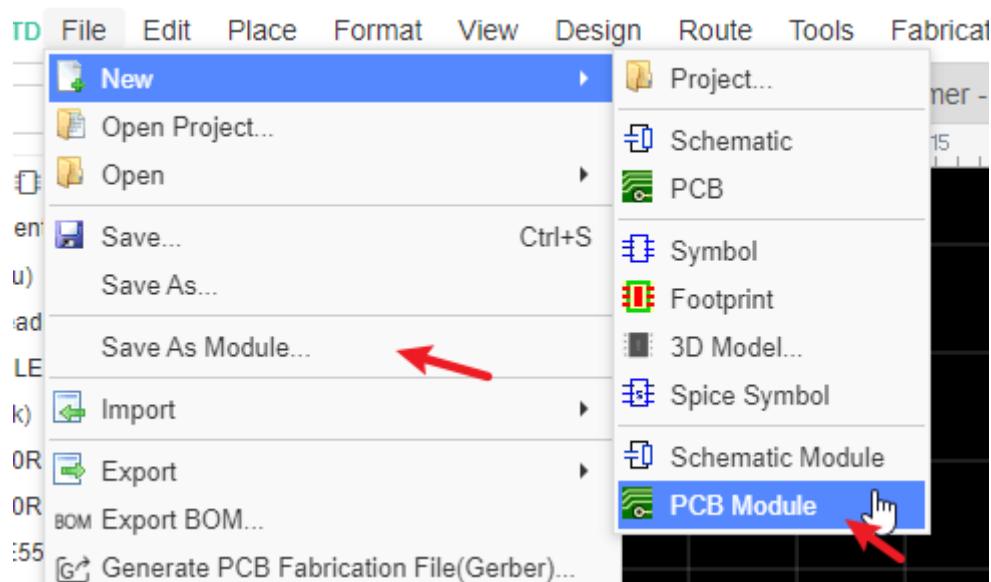
Nets shows: routed nets/total nets.

# PCB Module

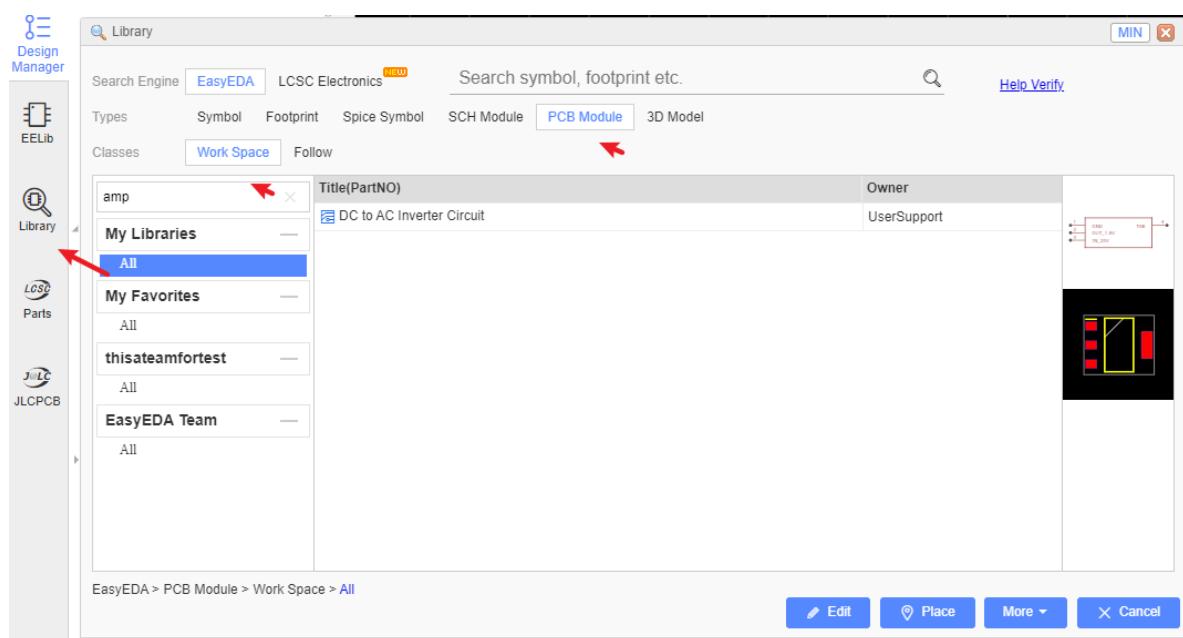
EasyEDA support create the PCB modules, it seems schematic module.

## How to Create

Via: **Save as Module** and **File > New > Schematic/PCB Module**.



PCB module save at **Library > Schematic/PCB module > Work Space > My Libraries**

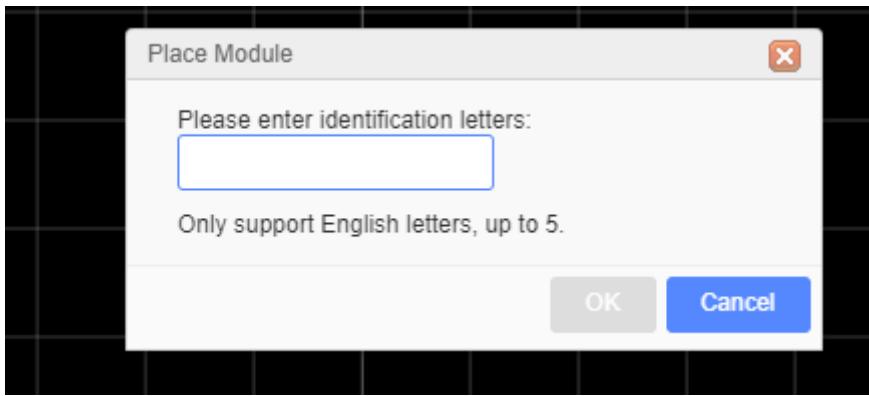


## How to use

Since v6.4.3, after placing schematic modules and PCB modules, after Import Changes, supports to keep the layout location.

How to use:

1. Draw schematic modules and PCB modules, and ensure that their component prefix are one to one, and the footprint is also corresponding. The module's component prefix can not have question marks and duplicate prefix, such as U? or two R1.
2. Open schematic and PCB at a same project.
3. Open "Library", select the module.
4. Click the "Place" button to place the previous saved schematic module and PCB module.
5. It will pop up a window to enter English letter. The letter of schematic module should keep corresponding with PCB modules.



For example: A component at schematic module is U2, enter letter K, press OK to place into canvas, it will be KU2, then PCB module has KU2 too.

Click "OK" and enter the placement mode. After each placement, the pop-up will continue to enter the identification letter. Make sure that the identification letters entered each time are unique.

6. When finish the module place, the PCB component unique ID will same as Schematic component unique ID, then after Import Changes, the component's location will be keep. and you can update the track's net follow the schematic netlabel too.

That implement the multipe chanel placing.

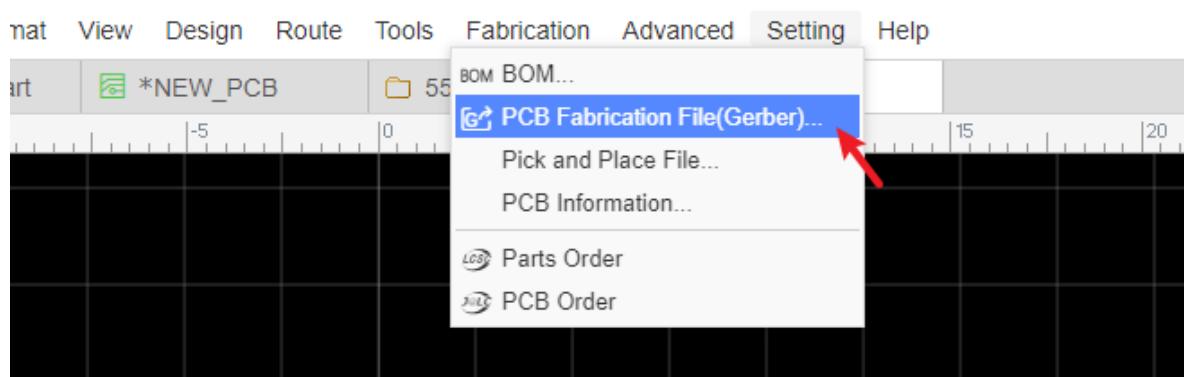
#### Notice:

- Module composes by tracks and components, it doesn't same as symbol binding footprint, the schematic module can not binding PCB module, after placing, the module will be separated by many objects, only the symbol and footprint can be corresponding via component ID, that is why you need to make the identification letter unique for placing each time to make sure schematic module corresponding with PCB module.

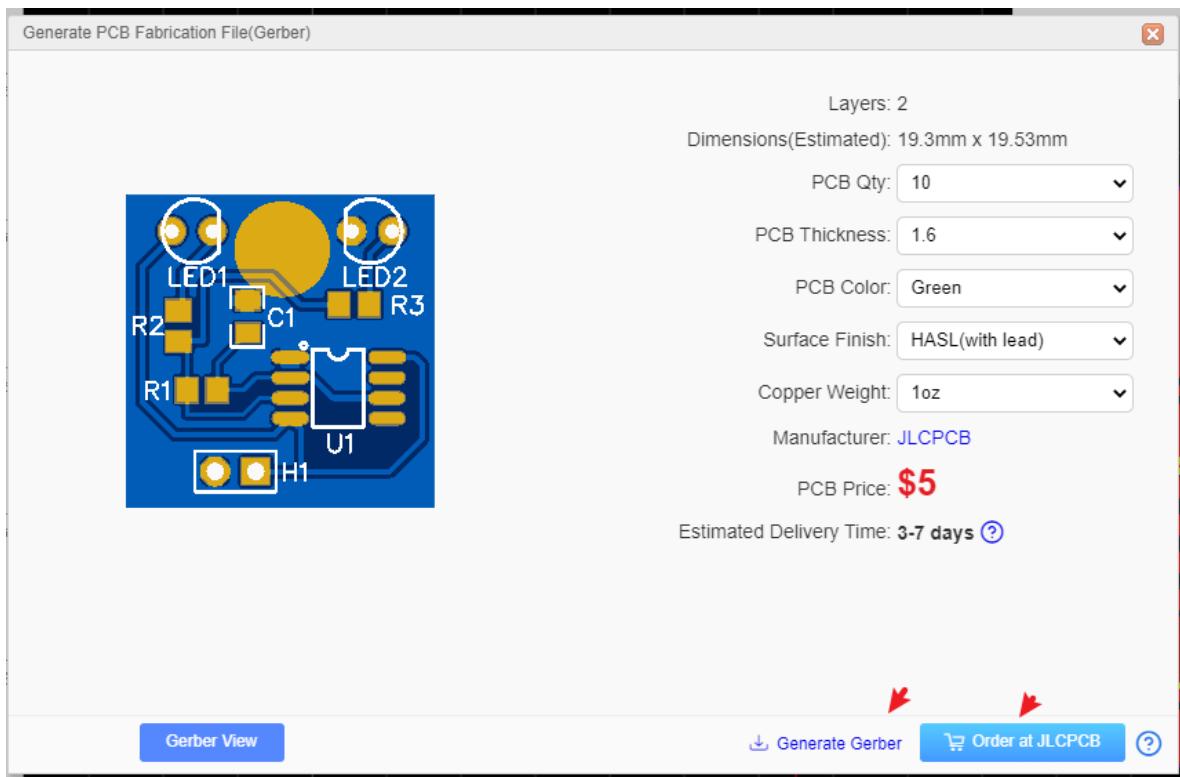
```
1 | # Generate Fabrication File(Gerber)
```

## Generate Fabrication File Gerber

When you finish your PCB, you can output the Fabrication Files(gerber file) via: **File > Generate PCB Fabrication File(Gerber)** , or **Fabrication > PCB Fabrication File(Gerber)**.



After clicking, will open the Gerber generate dialog:



You can calculate the price for the PCB order, click SAVE to CART will go to JLCPCB and add your PCB in the cart.

## Gerber file name

The generated Gerber file is a compressed zip file. After decompression, you can see the following files:

- **Gerber\_BoardOutline.GKO:** PCB Border file. The PCB board factory cuts the shape of the board according to this document. The groove drawn by the EasyEDA, the solid region(Type: NPTH) is reflected in the border file after the Gerber is generated.
- **Gerber\_TopLayer.GTL:** Top side copper layer.
- **Gerber\_BottomLayer.GBL:** Bottom side copper layer.
- **Gerber\_Inner1.G1, Gerber\_Inner2.G1...** : Inner copper layer.
- **Gerber\_TopSilkLayer.GTO:** Top silkscreen.
- **Gerber\_BottomSilkLayer.GBO:** Bottom silkscreen.
- **Gerber\_TopSolderMaskLayer.GTS:** Top solder mask. The default board is covered with green oil, and the elements drawn on this layer correspond to the top layer's area will not be covered with oil.
- **Gerber\_BottomSolderMaskLayer.GBS:** Bottom solder mask. The default board is covered with green oil, and the elements drawn on this layer correspond to the bottom layer's area will not be covered with oil.
- **Gerber\_Drill\_PTH.DRL:** Plated drill through hole layer. This document shows the location of the hole where the inner wall needs to be metallized.
- **Gerber\_Drill\_NPTH.DRL:** Non-Plated drill through hole layer. This document shows the location of the hole where the inner wall don't need to be metallized.
- **Gerber\_TopPasteMaskLayer.GTP:** Top Paste Mask, for the stencil.
- **Gerber\_BottomPasteMaskLayer.GBP:** Bottom Paste Mask, for the stencil.
- **ReadOnly.TopAssembly:** Top Assembly, read only, doesn't affect the PCB manufacture.
- **ReadOnly.BottomAssembly:** Bottom Assembly, read only, doesn't affect the PCB manufacture.
- **ReadOnly.Mechanical:** Record the information on the mechanical layer in the PCB design, and only use it for information recording. By default, the shape of the layer is not

manufactured at the time of production. Some board manufacturers use the mechanical layer to make the frame when using Altium file to production. When using Gerber file, it is only used for text identification in JLCPCB. For example: process parameters; V cut path etc. In EasyEDA, this layer does not affect the shape of the border of the board.

#### Notice:

- Before ordering the PCB, please check the gerber at the Gerber view as below.
- The Gerber files are generated by browser, please use the browser inner downloader to download!

## Gerber View

Before sending Gerber to the factory, please use gerber viewer to check the Gerber carefully.

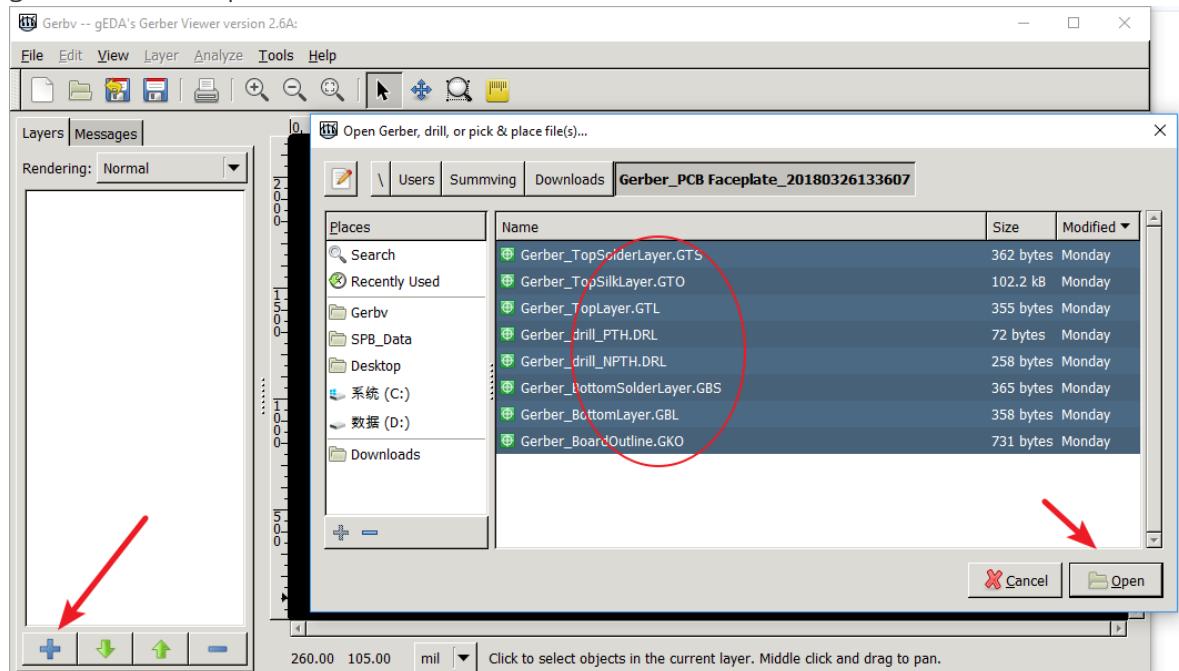
local gerber viewer you can use such as: Gerbv, FlatCAM, CAM350, ViewMate, GerberLogix etc.

Gerber viewer recommend Gerbv:

- Project page:<http://gerbv.geda-project.org/>
- Download: <https://sourceforge.net/projects/gerbv/files/>

How to use Gerbv:

- 1.Download Gerber zip file, and download Gerbv, unzip Gerber file and run the Gerbv;
- 2.Click the + button at the Gerbv dialog bottom-left corner, open the gerber folder, select all the gerber files, and open.



- 3.And then zoom, measure, check every layer, check drill holes and location. etc.

FlatCAM is a nice tool too: <http://flatcam.org/>

FlatCAM lets you take your designs to a CNC router. You can open Gerber, Excellon or G-code, edit it or create from scratch, and output G-Code. Isolation routing is one of many tasks that FlatCAM is perfect for. It's open source, written in Python and runs smoothly on most platforms.

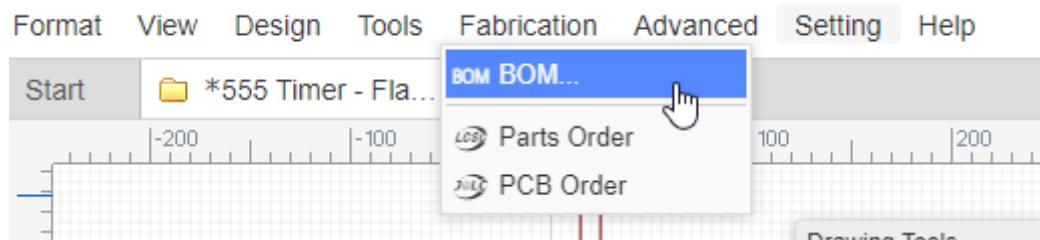
Free Online Gerber Viewer:

Recommend:

[jlpcb.com](http://jlpcb.com)  
[tracespace.io/view](http://tracespace.io/view)  
[gerber.ucamco.com](http://gerber.ucamco.com)

# Export BOM

You can export the Bill of Materials (BOM) for the schematic (Document) and PCB, via: "Top Menu - File - Export BOM", or "Top Menu - Fabrication - BOM".



After clicking the BOM export option, the dialog below will open.

In this dialog, you can click the button to assign LCSC part's order code for your components.

A screenshot of the 'Export BOM' dialog box. It displays a table of components with columns for ID, Name, Designator, Footprint, Quantity, Manufacturer Part, Manufacturer Ref., Supplier, Supplier Part, and Price. Each row has a blue 'Assign LCSC Part#' button in the last column. An arrow points from the text above to this button. At the bottom of the dialog are buttons for 'Export BOM', 'Order Parts/Check Stock', 'Cancel', and a help icon.

After clicking on the assign icon, the components and footprints search dialog will pop up, and you can choose which component you want to assign.

EasyEDA > Symbol > LCSC > NTC Thermistors > NTCG164BH103JT1

\$0.0769

LCSC Part# C524451 Stock: 3195 Minimum: 5 Distributor: LCSC

**Assign** **Cancel**

When you click the "Order Parts/Check Stock" button, we will help you to list all the components of your BOM at LCSC.com (If you haven't login LCSC, you have to login first). If you want to buy the components from LCSC, and you just need to put them to the cart and check out.

| ID | Name        | Designator | Footprint      | Qu... | Manufacturer Part  | Manufact... | Supplier | Supplier Part | Price                               |
|----|-------------|------------|----------------|-------|--------------------|-------------|----------|---------------|-------------------------------------|
| 1  | 47k         | R1         | 0805-RESISTOR  | 1     | ?                  |             |          |               | <b>Assign LCSC Part#</b>            |
| 2  | 470R        | R2         | 0805-RESISTOR  | 1     | ?                  |             |          |               | <b>Assign LCSC Part#</b>            |
| 3  | 220R        | R31        | 0805-RESISTOR  | 1     | ?                  |             |          |               | <b>Assign LCSC Part#</b>            |
| 4  | 10u         | C1         | 0805           | 1     | ?                  |             |          |               | <b>Assign LCSC Part#</b>            |
| 5  | ddd         | U1         | SOIC-8_150MIL  | 1     | NE555DR            | TI          | LCSC     | C7593         | <b>Assign LCSC Part#</b> \$0.143... |
| 6  | Header-M... | H1         | HDR-2X1.254    | 1     | 826629-2           | TE Conne... | LCSC     | C86471        | <b>Assign LCSC Part#</b> \$0.20275  |
| 7  | LED-3MM     | LED1       | LED-3MM/2.54   | 1     | 204-10SURD/S530-A3 | EVERLIGHT   | LCSC     | C99772        | <b>Assign LCSC Part#</b> \$0.0308   |
| 8  | LED-3MM     | LED2       | LED-3MM/2.5... | 1     | 204-10SURD/S530-A3 | EVERLIGHT   | LCSC     | C99772        | <b>Assign LCSC Part#</b> \$0.0308   |

**Export BOM** **Order Parts/Check Stock** **Cancel** **?**

You can open the BOM in any text editor or spreadsheet.

| A  | B                   | C        | D                | E          | F                        | G            | H        | I      | J      |
|----|---------------------|----------|------------------|------------|--------------------------|--------------|----------|--------|--------|
| id | value               | quantity | package          | components | Manufacturer Part        | Manufacturer | Supplier | LCSC   | price  |
| 2  | 1                   | 150      | 2 AXIAL-0.3      | R1,R4      | 25121WJ020KT4F           | UniOhm       | LCSC     | C45278 | \$0.02 |
| 3  | 2 22k               |          | 2 AXIAL-0.3      | R2,R3      | 25121WF300LT4F           | UniOhm       | LCSC     | C16074 | \$0.03 |
| 4  | 3 22u               |          | 2 CAP-D3.0XF1.5  | C1,C2      | 1812B225K500NT           | FH           | LCSC     | C28503 | \$0.28 |
| 5  | 4 204-10UYC/S530-A3 |          | 2 LED-3MM/2.54   | LED1,LED2  | 67-21S/KK3C-H2727QAR3LED | EVERLIGHT    | LCSC     | C73540 | \$0.04 |
| 6  | 5 2N3904            |          | 2 TO-92(TO-92-3) | Q1,Q2      | MURA220T3G               | ON           | LCSC     | C37995 | \$0.17 |

### Notice:

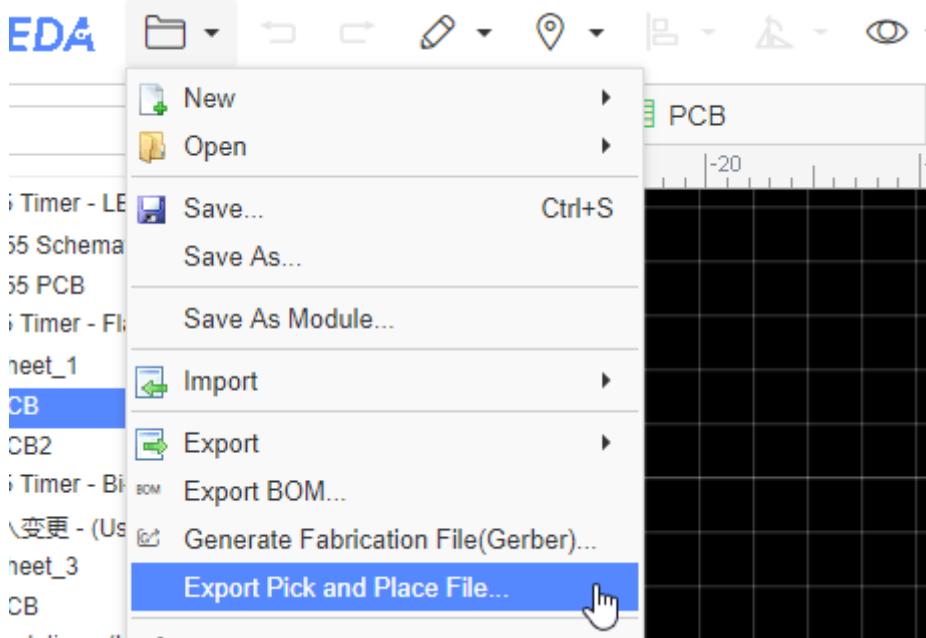
- If your project has schematic and PCB, the BOM data will come from schematic; if the project only has PCB, the BOM data will come from PCB.
- In order to support multiple languages, BOM and coordinate files (CSV file) are UNICODE encoded and tab-based. If the CSV file cannot be read by your components vendor or PCB manufacturer, please convert the encoding and change the delimiter.
- Recommended solution: Save as a new CSV file in Excel or WPS. For example, open a CSV file in Excel, click or select: Save As - Other Formats - CSV (Comma Separated) (\*.csv).

You can also open the CSV file with any text editor (such as Windows Notepad) and save as ANSI or UTF-8 encoding. If necessary, replace all tabs with commas.

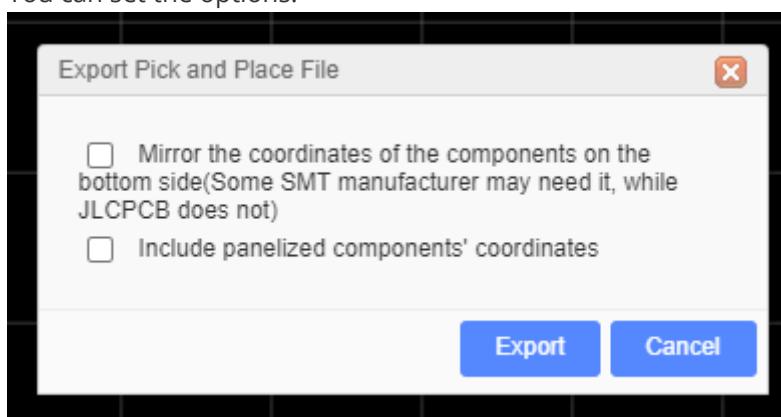
## Export Pick and Place File

In PCB editor, if you want to generate Pick And Place as a CSV file, you can via:

**File > Export Pick and Place File** or **Top Menu - Fabrication - Pick and Place File.**



You can set the options:



If your PCB has been panelize by the editor, you can enable the "Include panelized components coordinate".

When you open the exported CSV file, you can see:

|   | A          | B                   | C       | D       | E       | F       | G       | H       | I     | J        | K                    | L       | M |
|---|------------|---------------------|---------|---------|---------|---------|---------|---------|-------|----------|----------------------|---------|---|
| 1 | Designator | Footprint           | Mid X   | Mid Y   | Ref X   | Ref Y   | Pad X   | Pad Y   | Layer | Rotation | Comment              |         |   |
| 2 | LED2       | LED-3MM/2.15.4mm    | 17.27mm | 16.76mm | 17.27mm | 16.67mm | 17.27mm | T       |       | 270      | LED-3MM              |         |   |
| 3 | C1         | 805                 | 7.62mm  | 11.94mm | 7.62mm  | 10.92mm | 7.62mm  | 10.92mm | T     |          | 90                   | 10u     |   |
| 4 | U1         | SOIC-8_150M         | 13.31mm | 7.49mm  | 10.92mm | 9.4mm   | 10.29mm | 9.4mm   | T     |          | 0                    | NE555DR |   |
| 5 | LED1       | LED-3MM/2.14.16mm   | 17.27mm | 2.79mm  | 17.27mm | 2.89mm  | 17.27mm | T       |       | 90       | LED-3MM              |         |   |
| 6 | H1         | HDR-2X1/2.5 10.16mm | 2.29mm  | 11.43mm | 2.29mm  | 11.43mm | 2.29mm  | T       |       | 270      | Header-Male-2.54_1x2 |         |   |
| 7 | R1         | 0805-RESIST         | 4.76mm  | 7.37mm  | 3.81mm  | 7.37mm  | 3.81mm  | 7.37mm  | T     |          | 0                    | 47k     |   |
| 8 | R2         | 0805-RESIST         | 3.3mm   | 11.36mm | 3.3mm   | 10.41mm | 3.3mm   | 10.41mm | T     |          | 90                   | 470R    |   |
| 9 | R3         | 0805-RESIST         | 14.29mm | 12.7mm  | 15.24mm | 12.7mm  | 15.24mm | 12.7mm  | T     |          | 180                  | 220R    |   |

This file support two units "mm" and "mil", it is following the PCB unit setting.

There is an option "Mirror the coordinates of the components on the bottom side(Some SMT manufacturer may need it, while JLPCB does not)", you can check with your SMT manufacturer, the mostly SMT manufacturer doesn't need it.

#### Notice:

- In order to support multiple languages, BOM and Pick and Place files (CSV file) are UNICODE encoded and tab-based. If the CSV file cannot be read by your components vendor or PCB manufacturer, please convert the encoding and change the delimiter.
- Recommended solution: Save as a new CSV file in Excel or WPS. For example, open a CSV file in Excel, click or select: Save As - Other Formats - CSV (Comma Separated) (\*. csv). You can also open the CSV file with any text editor (such as Windows Notepad) and save as ANSI or UTF-8 encoding. If necessary, replace all tabs with commas.

# How to Order PCB

## Order Parts

1. Finish the schematic and PCB design at EasyEDA.
2. Open schematic, click "- Export BOM" button, the BOM dialog will open, click "Order Parts/Check Stock" button, will open [LCSC.com](#) order page. Check [Export BOM](#)
3. Add the parts to the cart, and then submit the payment.

## Order PCB

1. Open PCB, click "- Generate Fabrication File(Gerber)". Check [Generate Fabrication File\(Gerber\)](#)
2. Before ordering, check the Gerber first: [Gerber Viewer](#)
3. Visit at JLPCB <https://jlpcb.com/quote>, login with EasyEDA account.
4. Order PCB from EasyEDA editor directly(at Generate) or you can add the Gerber file(compressed file, ZIP) on the page and type the order options.
5. If you want to assembly parts, before enable the SMT option, you need to check all your parts are using "LCSC Assembled" class libs, and then upload the BOM file and Pick and Place file.
  - o [LCSC Assembled Libraries](#)
  - o [Export BOM](#)
  - o [Generate Fabrication File\(Gerber\)](#)
6. Save to the Cart, and then submit the payment.

Doesn't support to combine the components order with the PCB order.

More information please refer at:

[How to place an order](#)

[How to order a SMT order](#)

[How to order a stencil](#)

# Create The Footprint

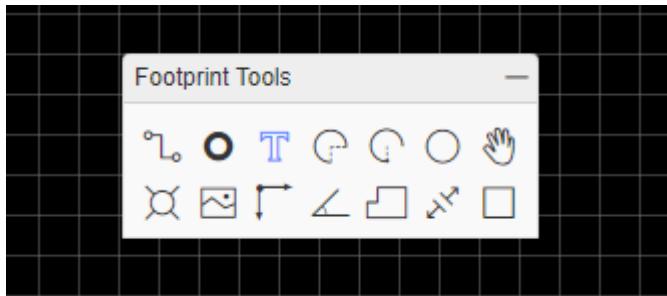
## Create The Footprint

There will be times when you will need a PCB footprint that is not already in the EasyEDA libraries.

### Footprint Tool

The process of creating your own Footprints is very similar to how you make symbols for your own Schematic Libs.

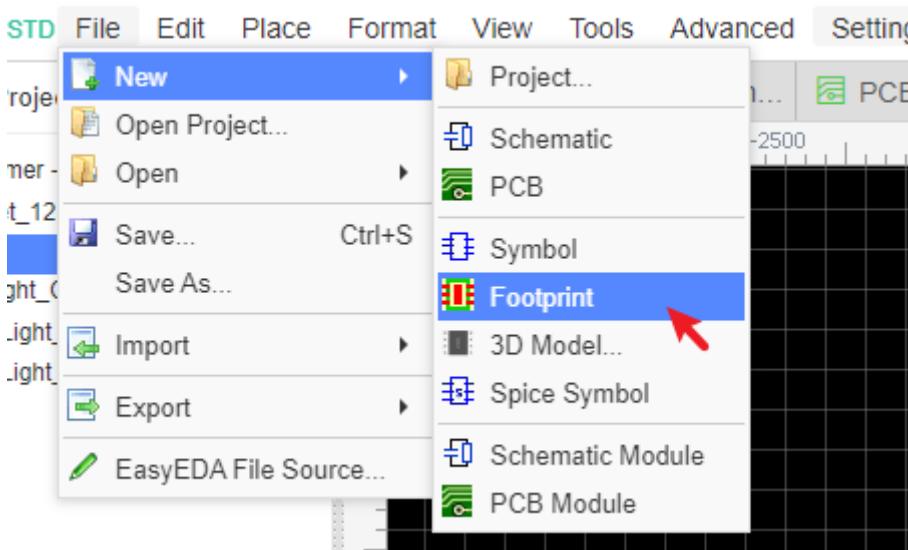
Footprint Tools almost are the same as PCB tools, just lacking some of the functions.



### Create Footprint

Start a new Footprint as shown below or by doing:

**File > New > Footprint**



This opens the New Footprint editor.

### Drawing Steps

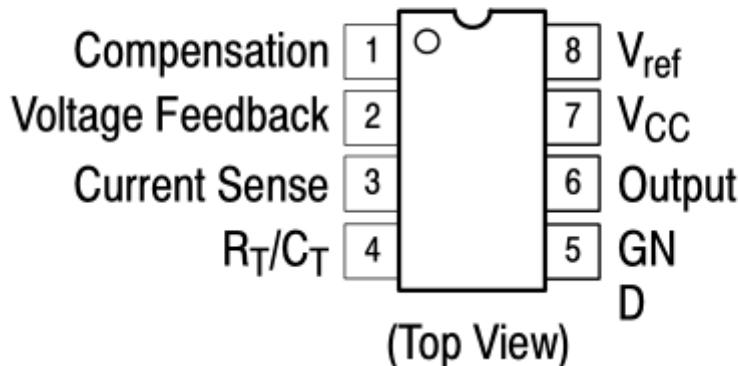
1. Download the datasheet which you need to draw the Footprint, such as SOIC-8.[Such as PDF: UC2844BD1R2G](#)

2. Read the datasheet, notice the 0 degree of the Footprint (The 0 degree is the Footprint's direction when you placed it on the PCB without rotation), the right 0 degree will helpful for PCB SMT.

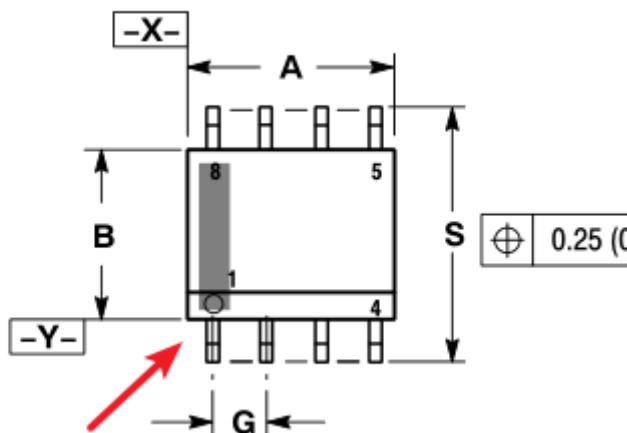
3. Check the footprint size, pad/pin direction and polarity, and then place the Pads on the canvas. You can adjust the pad size base on your real usage situation.

- Component's pin direction, page 1.

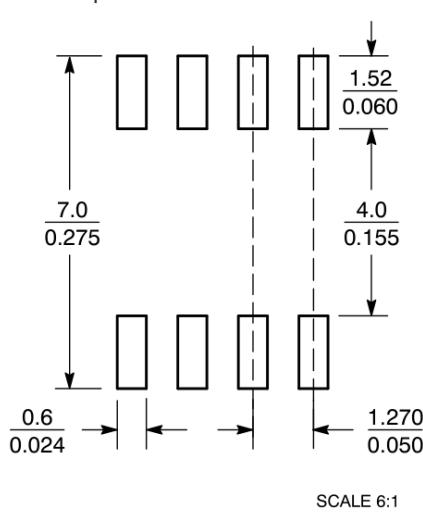
## PIN CONNECTIONS



- footprint polarity, page 1 and 18.

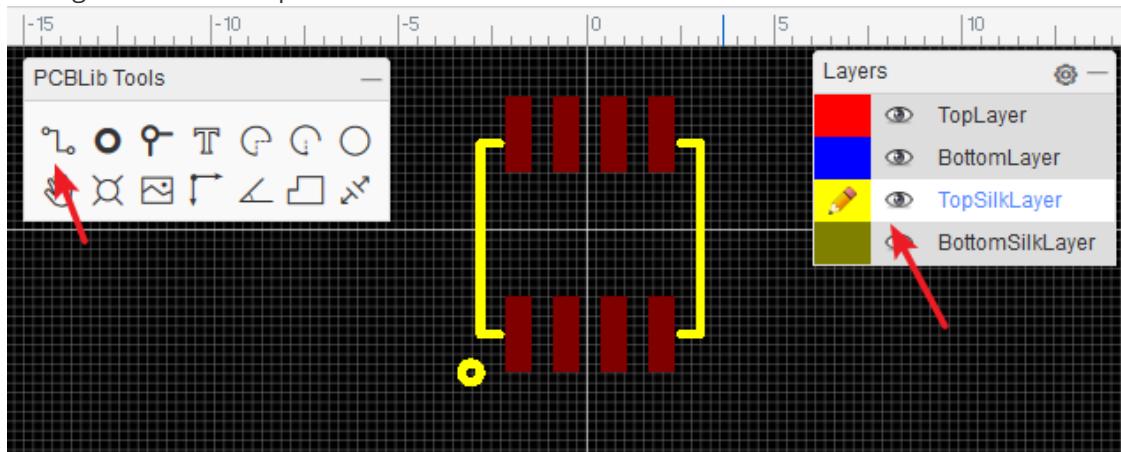


- Depends on page 18, placing one pad on the canvas on the top layer, and then change the pad number, size, shape type etc. And then set the coordinate for it, and place the less pads, you can use the "Top Menu - Align" tools to align the pads to fit the location. If you want to move the pad by mouse or direction key by small steps, you can set a new snap size at the right-hand panel.

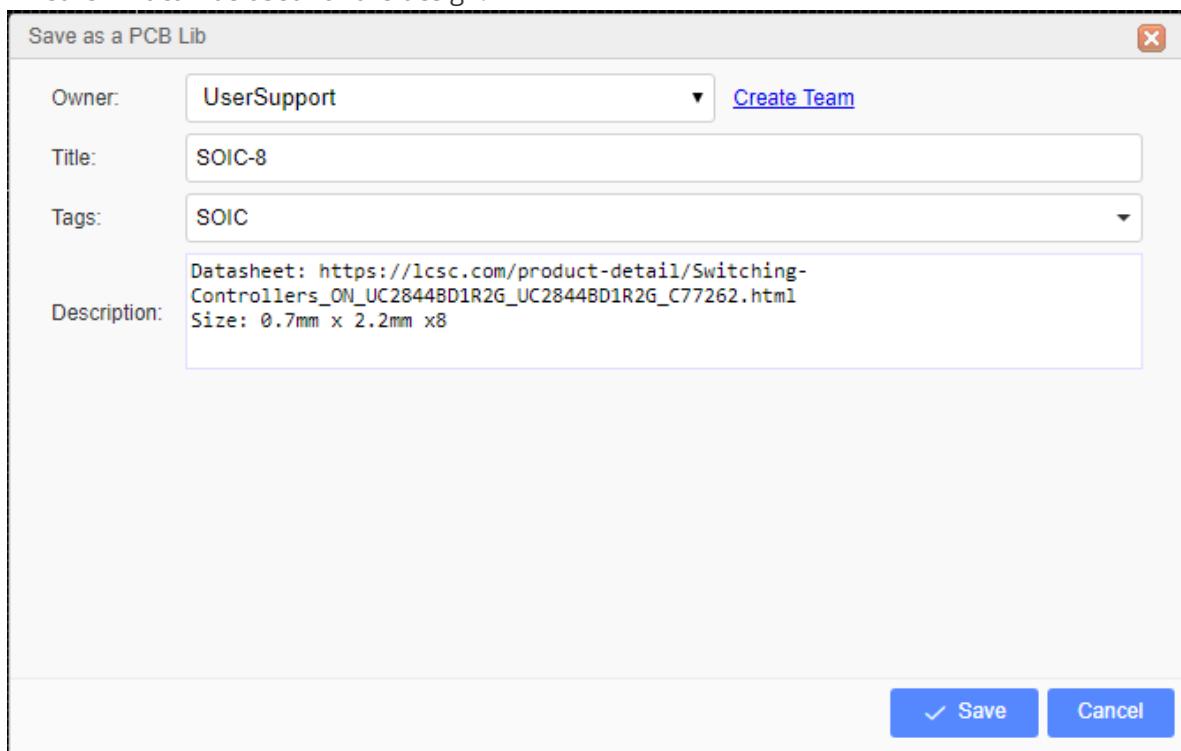


4.Drawing the Footprint silk screen. And sometimes you can add some marking and text on the mechanical or document layer.

- Swicthing layer to TopSlikLayer
- Using the Track and Arc to draw the silk screen. The editor doesn't support draw the retangle silk screen at present.



5.Filling the footprint title and prefix at the right-hand "Custom Attributes", and then Save. When you save it , please fill the tags, description, the description suggesting add the footprint datasheet link and footprint size, that can help you or other people to recognize this Footprint whether if it can be used for the design.



6.Use the dimension tool to check the Footprint size, via: Top Menu - Tools - Check Dimension.

7.Set the origin. You can via: "Top Menu - Place - Set Canvas Origin - By Center of Pads" to set the origin.

8.Save.

Then the PCB footprint creating finish .

#### **Notice:**

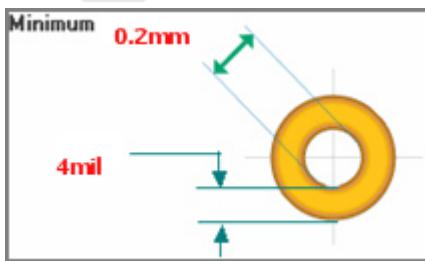
- The Origin Point. To simplify rotating your symbols when they are placed into the canvas, make sure all of your symbols are created as near as possible centered around that point.

Suggesting the footprint center to be the origin point. That will help to rotate when you placing it on the canvas, and help to do the SMT more quickly.

- The pad center suggesting one and more on the grid, avoid when place it on the PCB causing the track hard to connect issue.
- The pad number can be set as number and alphabet, they must match with the SchematicLib's pin number, otherwise the component which was assigned this Footprint will alert the error at the footprint manager, and can't convert the schematic to PCB.
- The pad number will increase by placing with mouse, if you copy and paste it, the number will not increase.

## Others

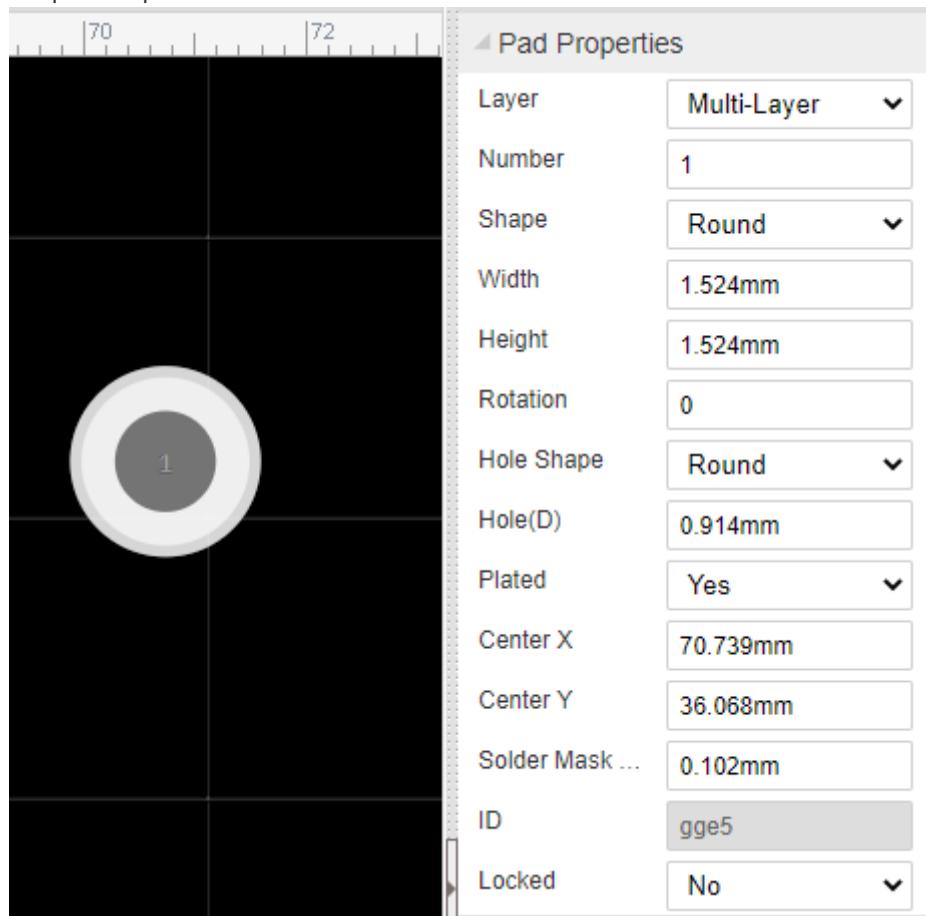
- It is important to set the right Snap and Grid sizes to ensure that the pads on the finished footprint snap exactly to the grid and so connect the nets. For example, if you are creating a DIP footprint, set the Grid size to 100mil.
- Keep all other shapes such as component outlines and any associated pin identification marks or text on the TopSilkLayer. EasyEDA will automatically take care of the actual layer assignment when you place the footprint on the PCB.
- **CTRL+S** to save your footprint designs and you will find them saved into the **Libraries > Classes: Footprint > Personal > Created** section of the left Navigation panel.
- Annular ring of the pad/via is too small, keep the annular ring  $\geq$  4mil. In this case, you can add a **Hole**



## Pad attributes

You can add pads using the Pads button from the Footprint Tools palette or using the **P** hotkey.

After selecting one of the pads, you can view and adjust its attributes in the right hand Properties panel.



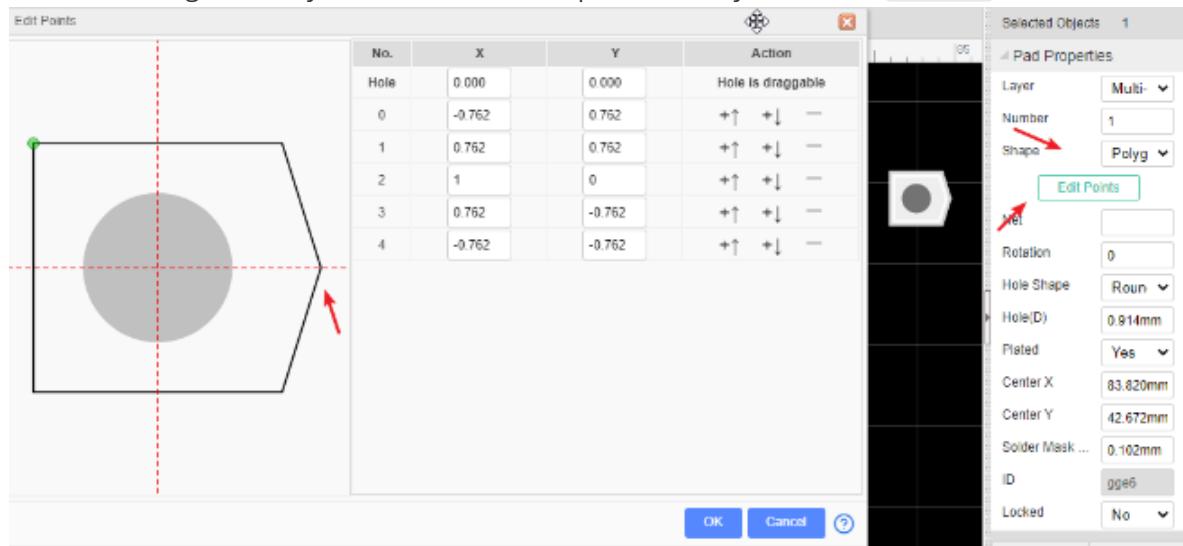
**Number:** Remembering the pin numbers you set in the schematic symbol in your Schematic symbol: to connect those schematic symbol pins to the pads in your PCB footprint, the pad numbers you set here in the Footprint footprint must be the same.

**Shape:** Round , Rectangular , Oval and Polygon.

EasyEDA supports four shapes: Round , Rectangular , OVAL and POLYGON .

- OVAL PAD will give you more space.
- POLYGON PAD will let you to create some strange pad.

Like in the image below, you can edit the PADs points when you select a POLYGON PAD



**Layer:** If the pads are part of a **SMD** footprint, you can set it to **Top layer** or **Bottom layer**. For through hole components you should set it to **Multi-Layer**.

**Net:** You don't need to enter anything here because at present this footprint is not connected to anything in a circuit.

**Width and Height:** When the shape is set to Round, Width will equal Height.

**Rotation:** Here you can set the Pad's rotation as you want.

**Hole(D):** This is the drill hole **diameter** for a through hole pad. For a SMD Pad, set this to **zero**.

**Center-X and Center-Y:** using these two attributes, you can set the pad's position with more precision, compared to using the mouse.

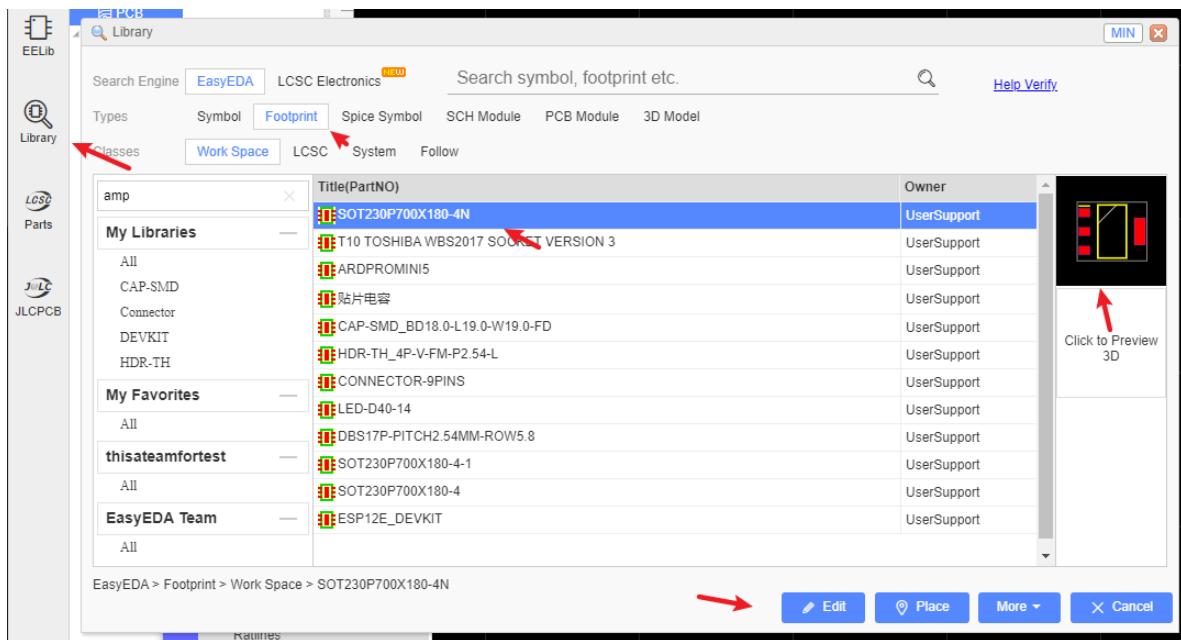
**Plated:** Yes or No. when the pad is multi-layer pad, if it set the plated as no, this pad top side and bottom side will not be connected together.

# Edit Footprints

## Edit Footprint in Library

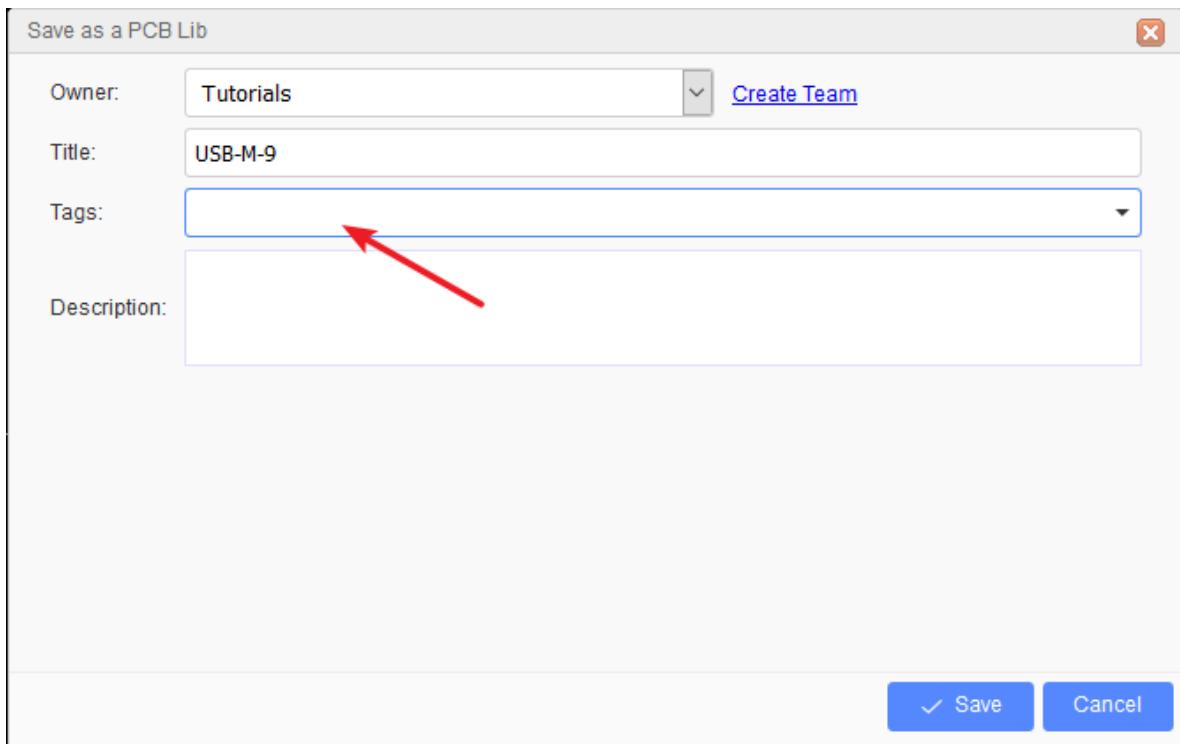
When you found a Footprints(footprint) but it can not be satisfied for your design, you can edit it to be your personal PCB footprint.

Via **Library > Footprint > Search Component/Personal/LCSC/System > Select footprint > Edit**

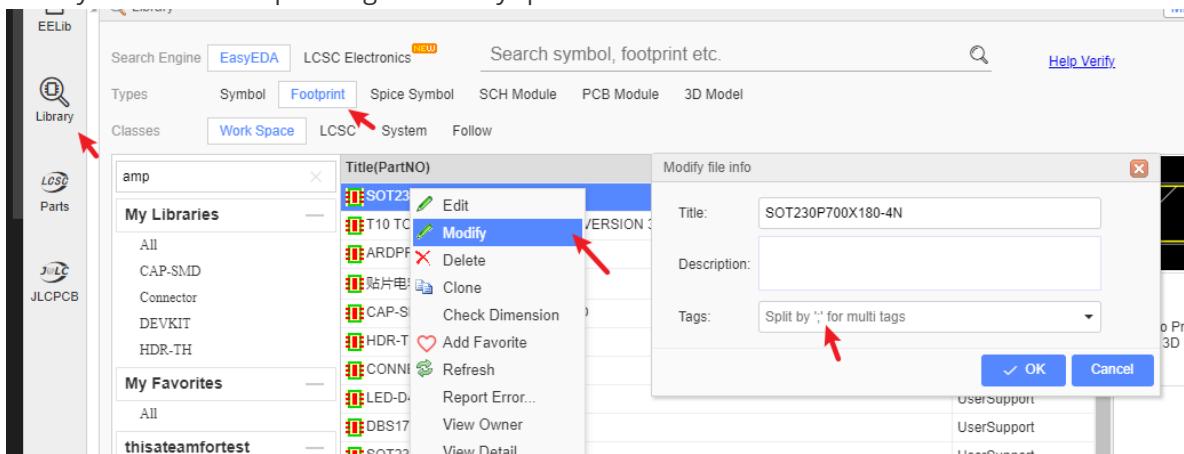


You can edit the pad size, shape outlines, etc. when you finish and save, it will be saved to your personal libraries "Created" and become your personal libraries.

And you can add a tag for your Footprint when you save it:



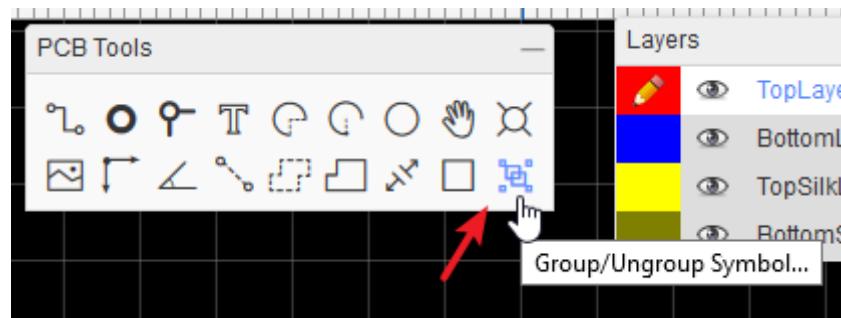
Modify the saved Footprint tag at "Library" part list.



## Edit Footprint in PCB

If you want to edit a package(footprint) in the PCB, you can use the Ungroup/Group function same as the schematic.

On the **PCB Tools** palette there is the **Group/Ungroup Symbol...** button.



This tool is for you to quickly create or edit library symbols.

1. Select a footprint

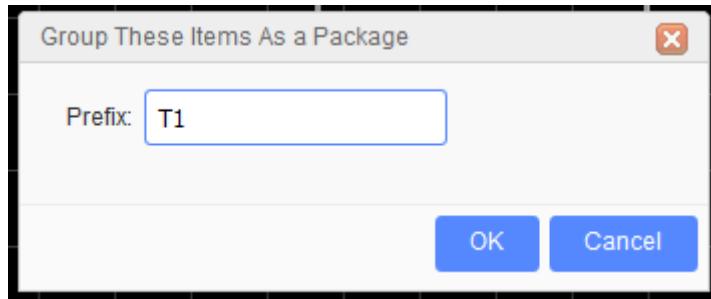
2. Click the **Group/Ungroup Symbol...** button

Up to this point you have a collection of separate pads, a drawn silk layer tracks and some text that are all separate items with no particular association with each other.

3. Edit the shape or pad what you want to change

4. Select all of the items and click the **Group/Ungroup Symbol...** button.

A dialog will be opened:



After you click OK, all those separate elements will be grouped together to form your new symbol directly in the PCB.

Using the group function, you can create/edit any symbol in the Schematic/PCB, easily and quickly.

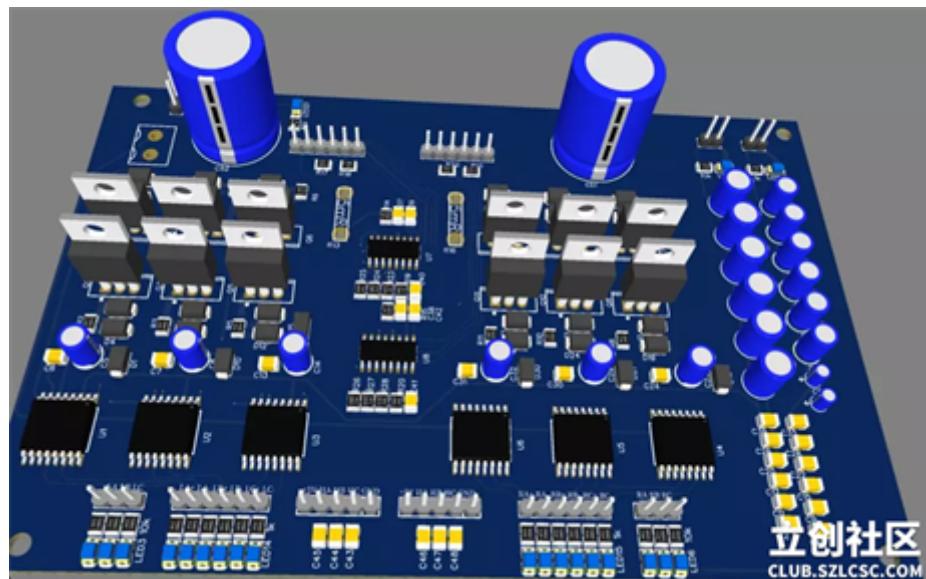
Notice:

- Before ungroup the footprint, please change it's layer to top layer first, because of the footprint after grouping will at top layer.

## Import PCB 3DLib

### Import 3D File

EasyEDA supports for importing 3D models, PCB can view cool 3D models when doing 3D preview. Exporting PCB 3D model files is not supported yet.



### 1. Draw or download 3D model

Note: currently only 3D models in "WRL(VRML)" and "obj" are supported. WRL can be imported directly without the need for compression; Obj must be compressed into a zip file with the MLT file and then imported, and the MLT file is usually taken with you when you download the obj file. Other formats of 3D files will be supported in the future.

Note that file suffixes cannot be capitalized.

Download address:

<https://library.io/explore/3dmodels> (MLT files are automatically downloaded when obj files are downloaded.)

<https://github.com/KiCad/kicad-packages3D>

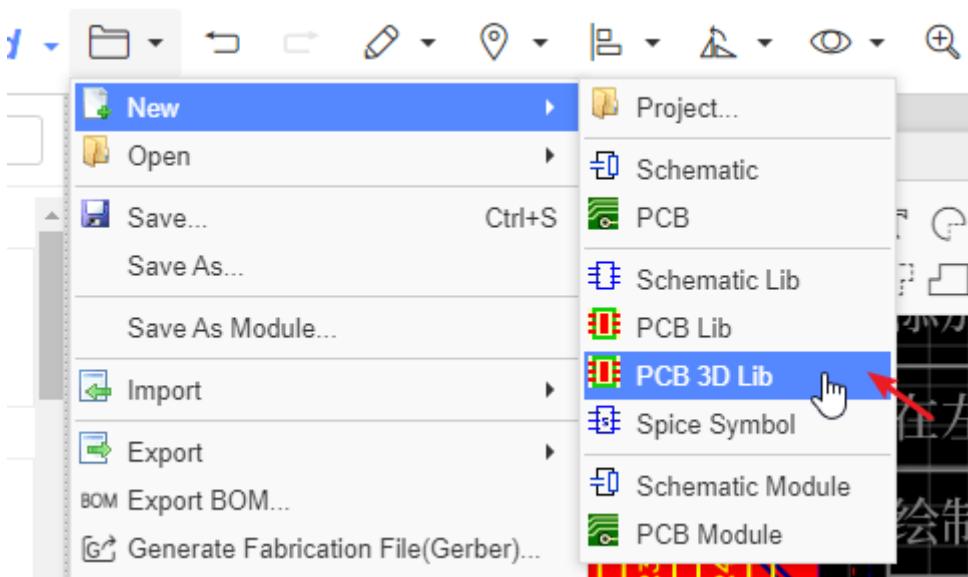
<https://www.traceparts.com/zh>

<https://www.3dcontentcentral.com/>

<https://grabcad.com/>

## 2. Create a new 3D library

in "Top Meun - New - PCB 3D Lib".

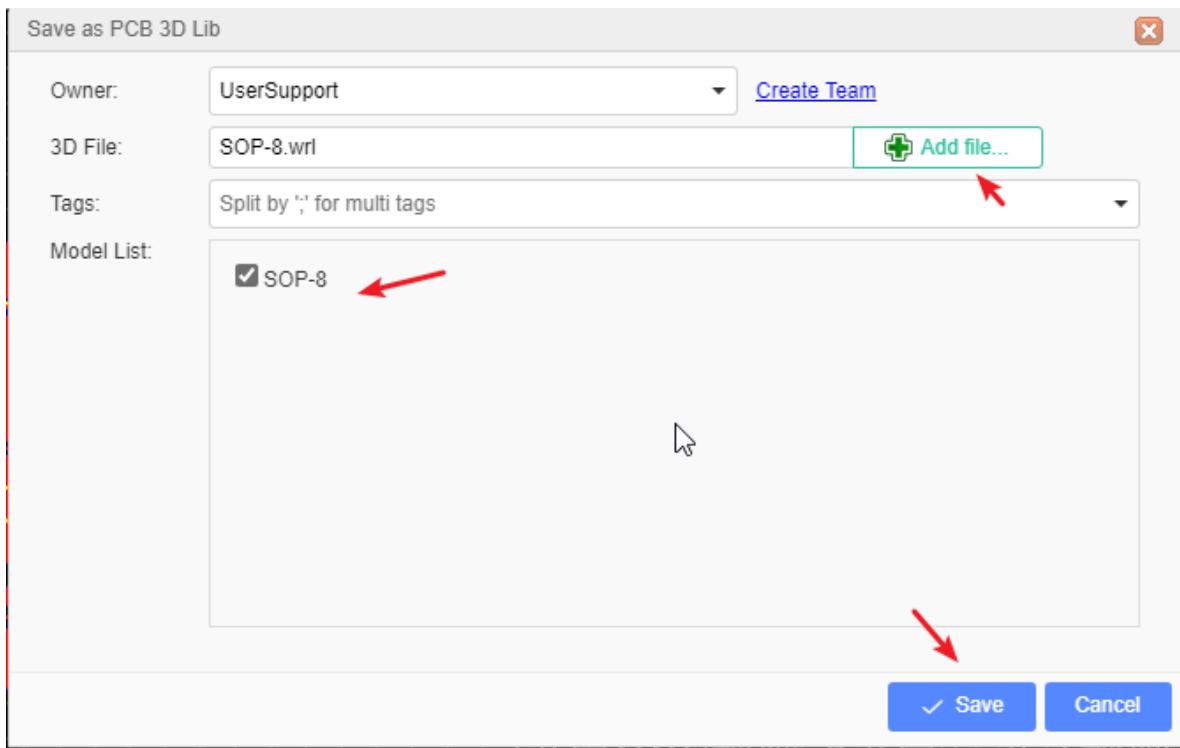


If you have many 3D libraries, you can zip them to a zip file to import, no more than 10 WRL files for one zip file, otherwise it will fail to import.

OBJ format contains many 3D models in one file, you don't need to zip them.

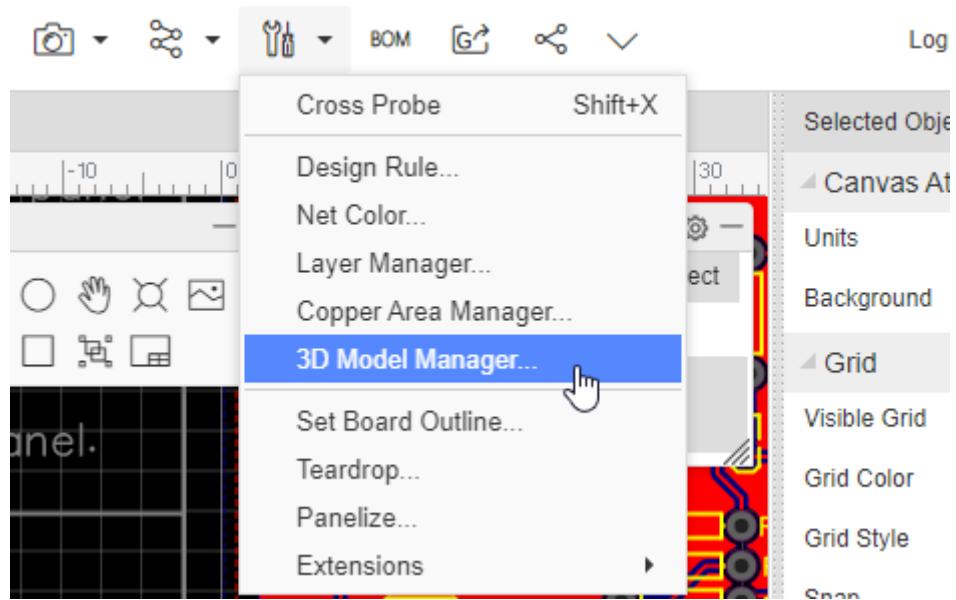
## 3. Import 3D model.

You can check which 3D model you want to import.



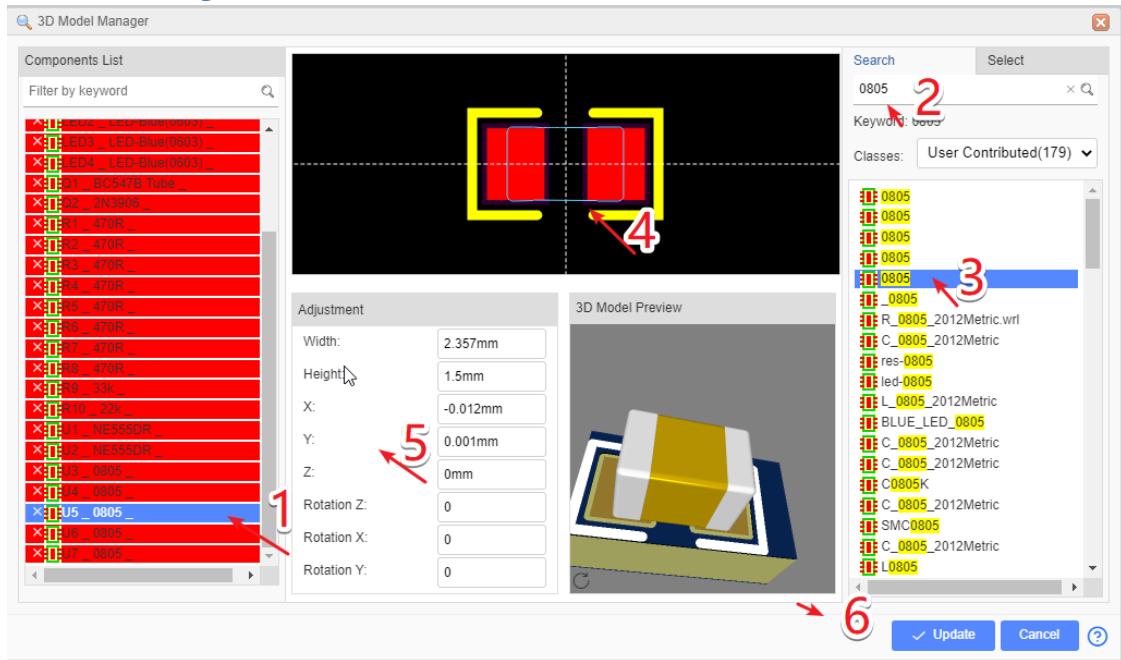
## 4. Specify the 3D model

- Open the PCB or Footprint, and find " - Tools - 3D Model Manager"



- Specify the imported 3D model for the corresponding footprint, which is basically consistent with the footprint manager operation. For the specific use of the tutorial, please see: [PCB -](#)

## 3D Model Manager



- Adjust the position and parameter relationship between the 3D model and PCB packaging, and click update
- After completing all the specified 3D models, you can start the 3D preview of the whole PCB.

## Edit 3D Lib

1. The SHIFT+F shortcut opens the component library dialog box
2. Switch to "PCB 3D library" and "WorkSpace"
3. Right click can edit and delete 3D library

| Title(PartNO) | Owner       |
|---------------|-------------|
| DIP28-600     | 立创EDA团队     |
| DIP40-600     | 立创EDA团队     |
| DIP48-600     | 立创EDA团队     |
| cdrh73        | UserSupport |
| User Library  |             |
| 6120202282    | 立创EDA使用培训   |
| 6130201182    | 立创EDA使用培训   |
| R0402         | 立创EDA团队     |
| R0603         | 立创EDA团队     |
| R0805         | 立创EDA团队     |

FAQ:

Q: Can the official footprint library specify the 3D model first?

A: Yes, later official libraries will specify 3D models. At present, you need to specify to PCB or PCB footprint.

Q: Can EasyEDA export the whole PCB 3D format for structural design? Step, etc.

A: It will be supported in the future, step by step, and will directly support importing the step format in the future. This format is more complicated and needs time to study.

Q: Will EasyEDA support to draw 3D models in the future?

A: Don't. At present, many 3D rendering tools are very mature (Auto CAD, CAXA, SolidWorks, etc.) or open source free (FreeCAD, LibreCAD). Online 3D design tools (onshape) are also available.

# 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 screenshot shows a software interface for managing component footprints. On the left, a sidebar lists categories such as '1.3.2 Axial Through hole Capacitor, cylindrical Th', '1.3.3 Rectangular Through hole Inductor, Axial Ind', '1.3.4 Axial Through hole Fuse, Flat Shape Through', '1.3.5 Axial Diode, Through hole Rectifier bridge', '1.3.6 Through hole Regular, Cylindrical , Long Cy', '2.Regular Package Shape Semiconductor', '2.1 Small Outline Transistor', '2.2 Small Outline Package', '2.3 Dual-In-Line Package', '2.4 Quad Flat Pack', '2.5 Ball Grid Array, Land Grid Array', '2.6 Leadless Chip Carrier', '2.7 Quad Flat No-lead/Dual Flat No-Lead', '3.Other Package Shape Semiconductor', '3.1.Standard Package Semiconductor', '3.2.Non-Standard Package Semiconductor', '3.2.1 Transistor', and '3.2.2 Integrated Circuit'. On the right, there is a detailed technical diagram of a component. The diagram includes a top-down view with dimensions: BL (Body Length), PP (Pin Pitch), LS (Lead Spacing), and BW (Body Width). It also shows a side view and a 3D perspective view. Below the diagram, there are two smaller views: a regular shape component with pin arrangement and a non-regular shape component with pin arrangement. Text below the diagrams provides naming formats and instructions for package types.

Regular shape, regular arrangement of pins naming format:  
[PKT]-[Q]\_L-[BL]-W[BW]-P[PP]-LS[LS]-{TL/TR/BL/BR}-{EP}

Non-Regular shape, regular arrangement of pins naming format:  
[PKT]-[Q1]\_|Q2|P-L-[BL]-W[BW]-P[PP]-LS[LS]-{TL/TR/BL/BR}-{PE[X]}-{EP}\_{(SN/MPN)}

Instructions:

1. Package Type. For example:
  - a. SOP, Small Outline Package
  - b. TSOP, Thin Small Outline Package
  - c. MSOP, Micro Small Outline Package
  - d. HSOP, Heat Sink Small Outline Package
  - e. TSSOP, Thin Shrink Small Outline Package
  - f. HTSSOP, Heat-Sink Thin Shrink Small Outline Package
  - g. SSOP, Shrink Small Outline Package
  - h. VSOOP, Very Small Outline Package
  - i. SOIC, Small Outline Intergrated Circuit
  - j. SOJ, Small Outline IC J-Leaded
  - k. SON, Small Outline No-lead
  - l. SO, Small Outline

The majority of EasyEDA users can also according to this rule:

1. Find the components of the specified package 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 "package 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

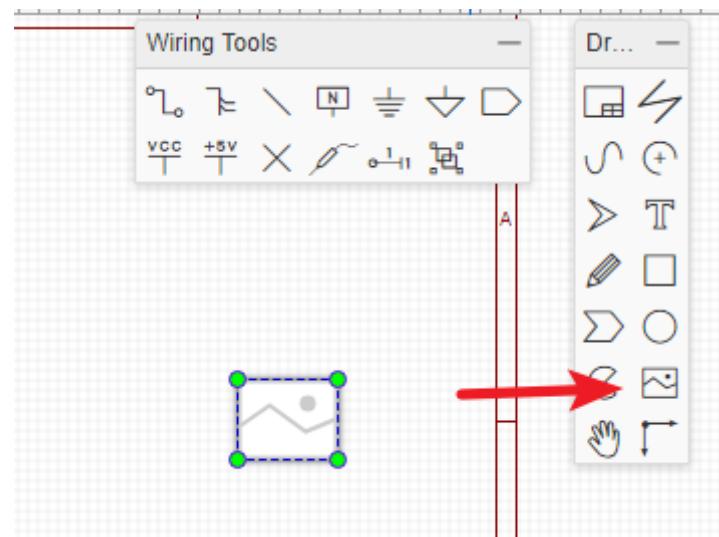
#### **Download:**

Download: [EasyEDA Footprint Naming Rule Reference.pdf](#)

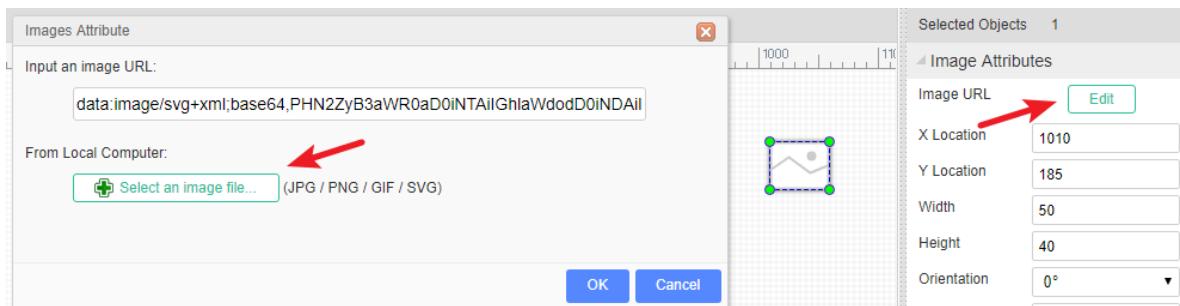
# Import Image

## Import Image to Schematic

When you select Image from the Drawing Tools palette, an image place holder will be inserted into the canvas:



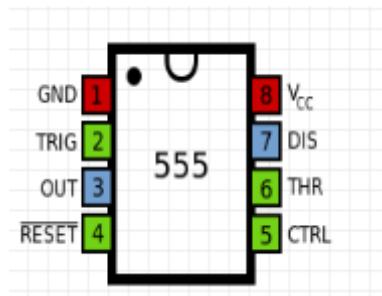
Select the place holder, so you can see the image's attributes in the right hand Properties panel:



Set the URL of your image. For example, setting the URL to:

[http://upload.wikimedia.org/wikipedia/commons/thumb/c/c7/555\\_Pinout.svg/220px-555\\_Pinout.svg.png](http://upload.wikimedia.org/wikipedia/commons/thumb/c/c7/555_Pinout.svg/220px-555_Pinout.svg.png)

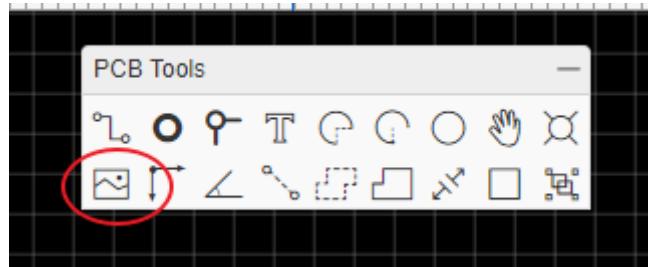
will make your image look like this:



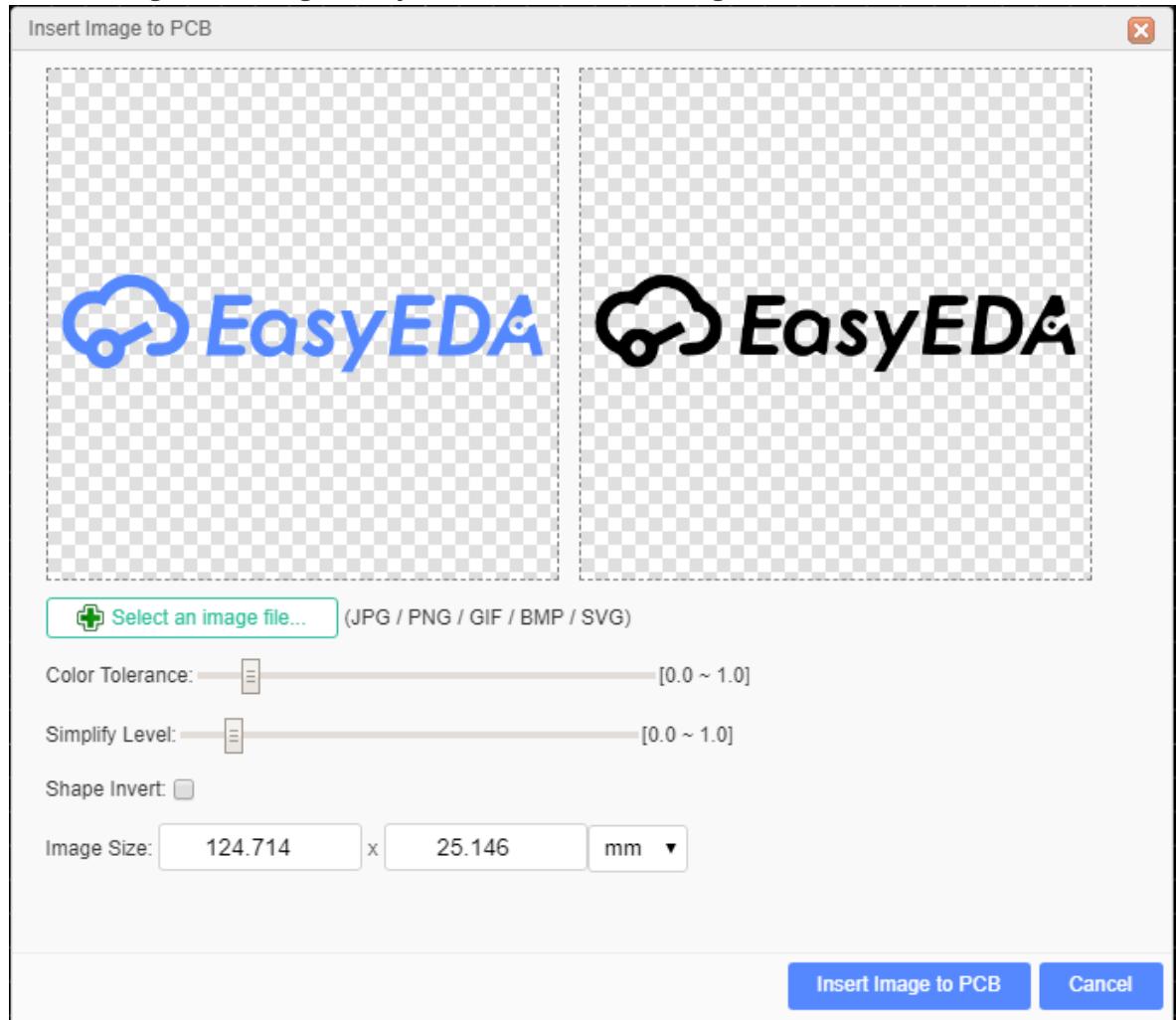
Please note: at present, EasyEDA cannot host images, so you need to upload your images to an image sharing site such as <http://www.imgur.com>.

## Import Image to PCB

On PCB and Footprint editor, there is a nice feature on the PCB Tools bar.



After clicking on the image icon, you will see the Insert Image window as below.

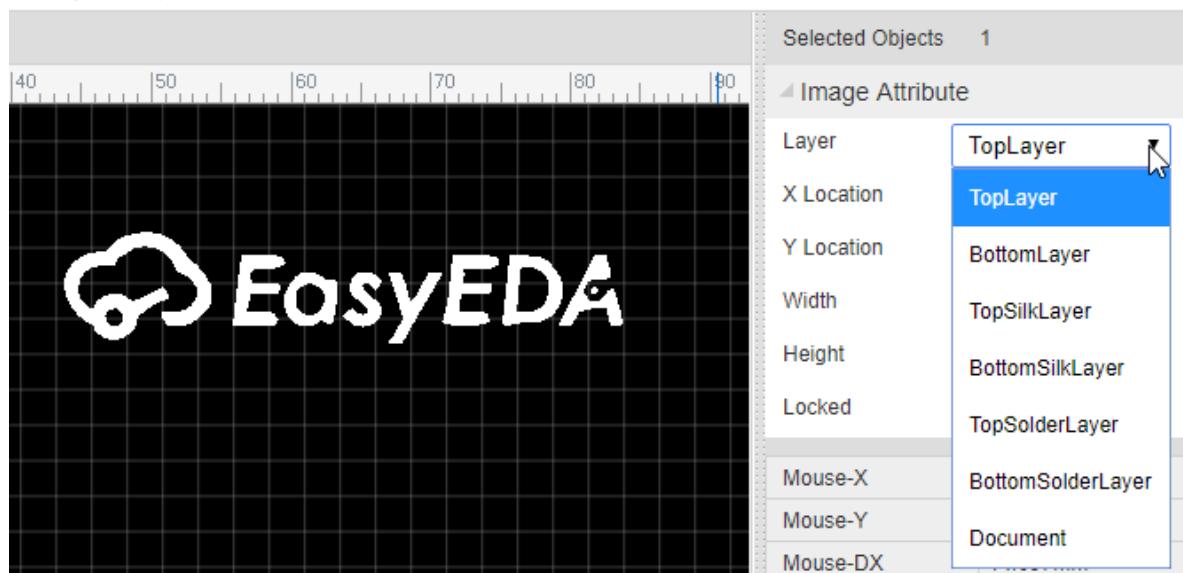


In this dialog, you can choose your favorite image, EasyEDA support [JPG](#), [BMP](#), [PNG](#), [GIF](#), and [SVG](#). Unlike some other EDA tools which only support a Monochrome Bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

You can adjust the color tolerance, simplify level and reset the image size there.

And you can select shape invert.

The image will be inserted to the active layer, if it is not right, you can change the attribute. Such as TopSilkLayer.



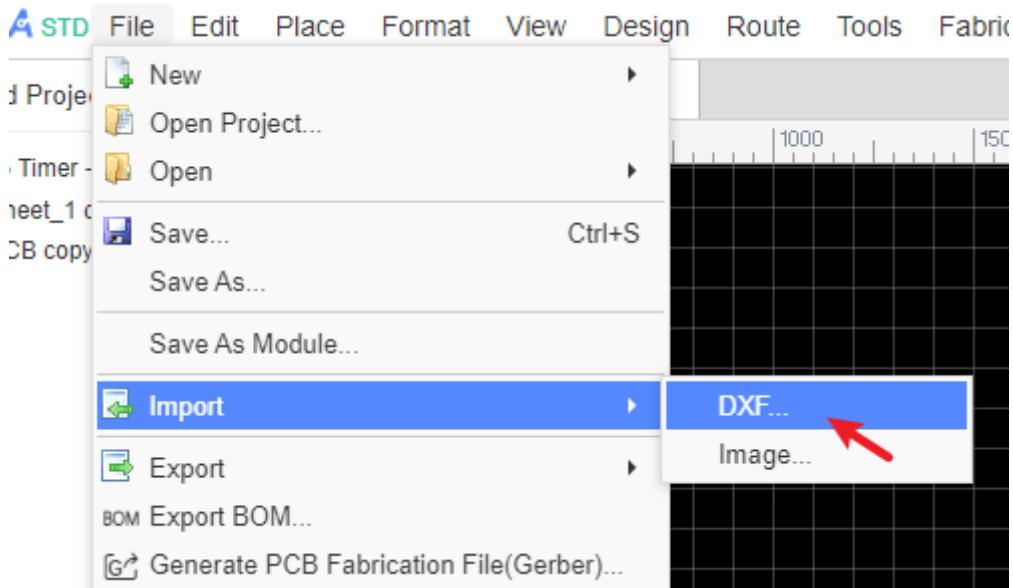
# Import DXF File

How to create irregular board outlines or complex board outline in EasyEDA?

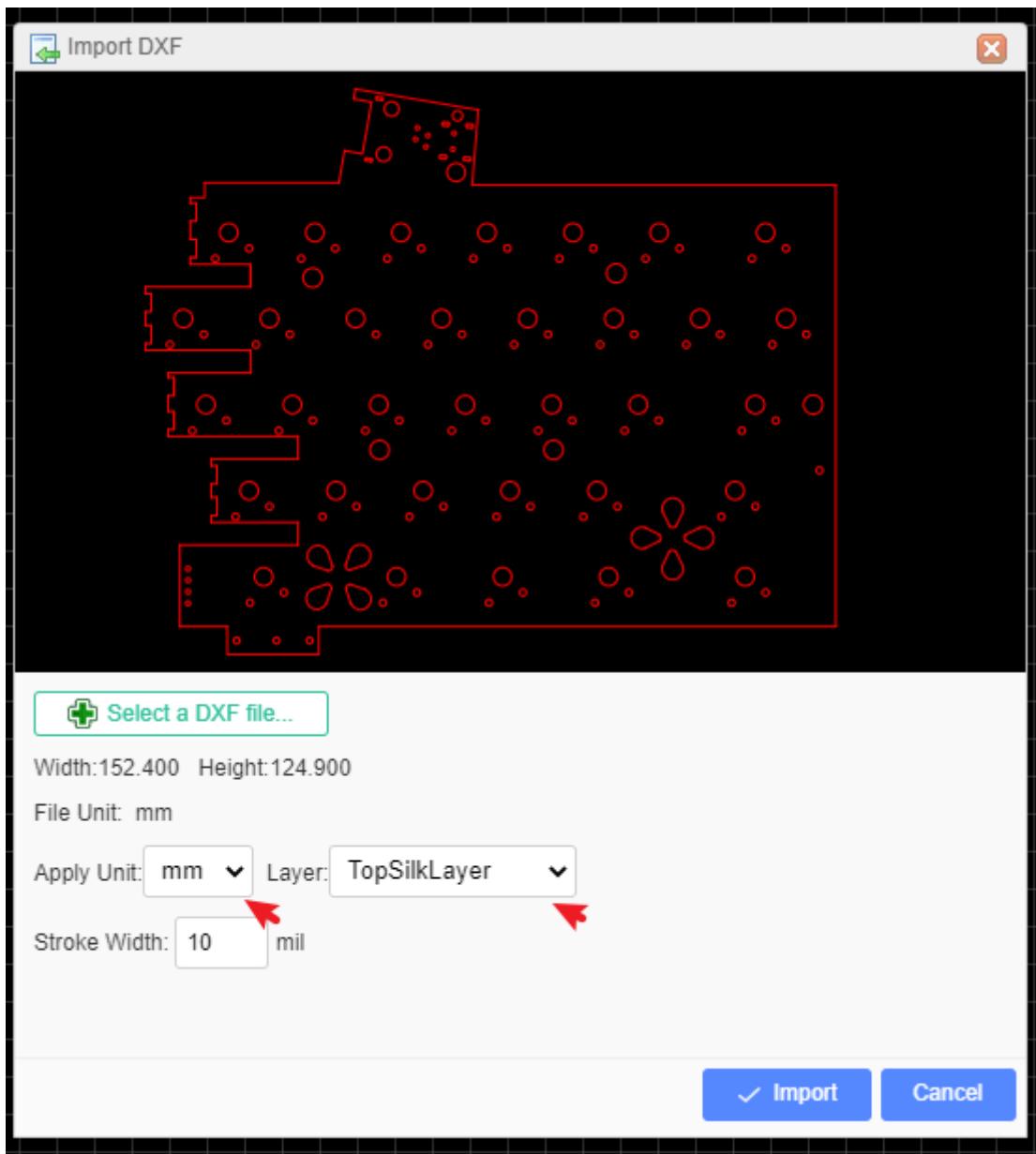
This is sometimes needed when you are designing a PCB for an enclosure that may have a curved profile, or other unavoidable mechanical features for which one must design.

EasyEDA supports that import DXF into PCB.

Find the import DXF menu under the file menu. Via: File - Import - DXF



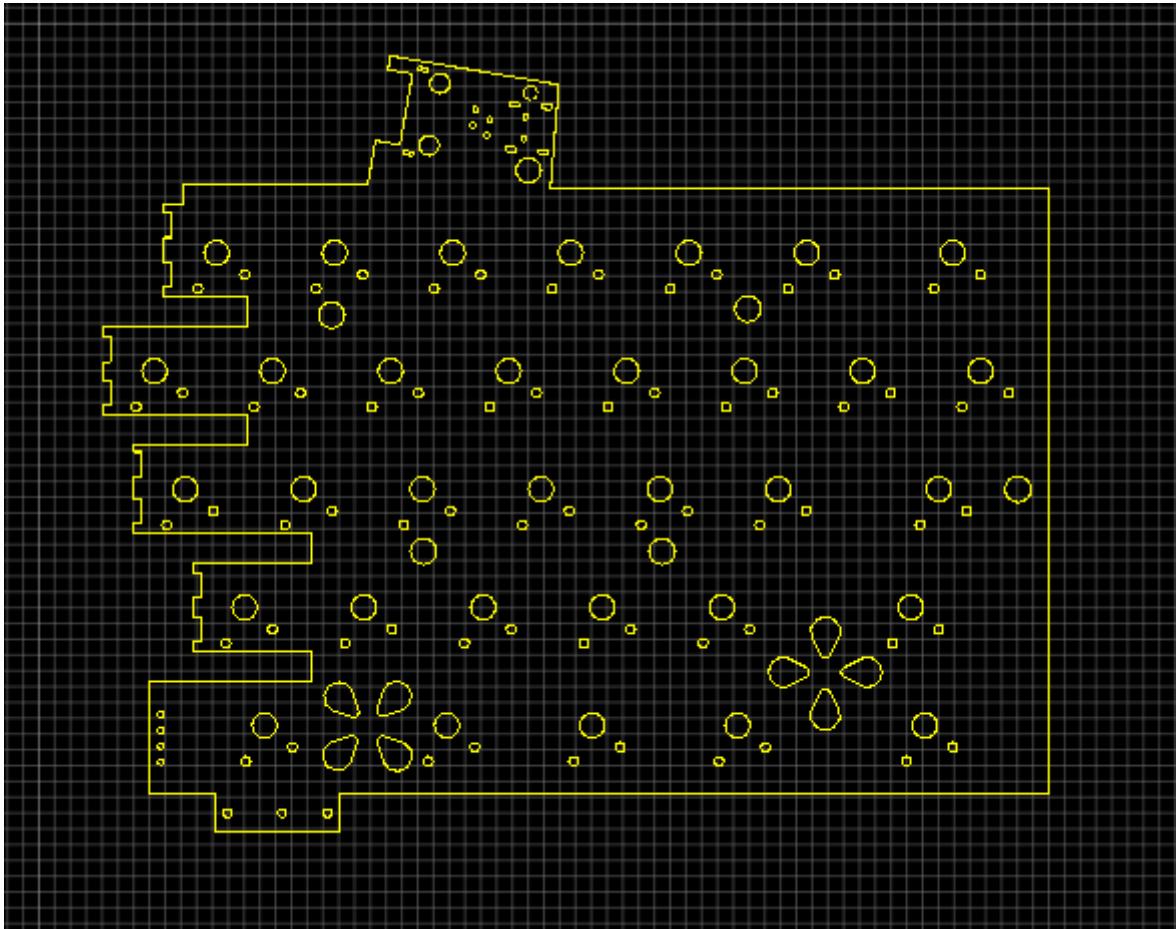
After selecting the \*.DXF file, you will find a dialog like in the image below



EasyEDA provides some options, unit(mm, cm, mil, inch), and select the layer to which the shapes will be applied.

If you import DXF into schematic or symbol, its unit is pixel.

After clicking the import button, you will find them on your PCB canvas.



You can try this to import this example by yourself. [DXF example](#)

Please note:

1. The file must have a \*.dxf filename extension.
2. The circles will be converted to holes if you choose the layer as board outline.
3. There are some items which are not supported. Such as Mirror, spiral line etc.
4. If the DXF objects were grouped, please ungroup them before importing.
5. Do not import the big DXF to copper layer directly, it will make the editor stuck a period of time.

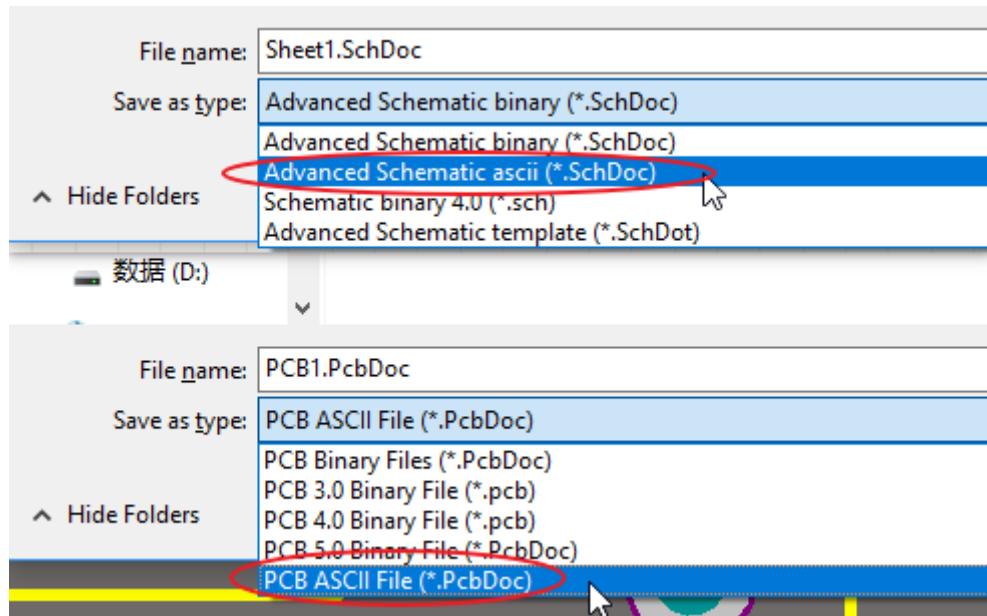
## Import Altium Designer

The import function is beta now, please check carefully after imported.

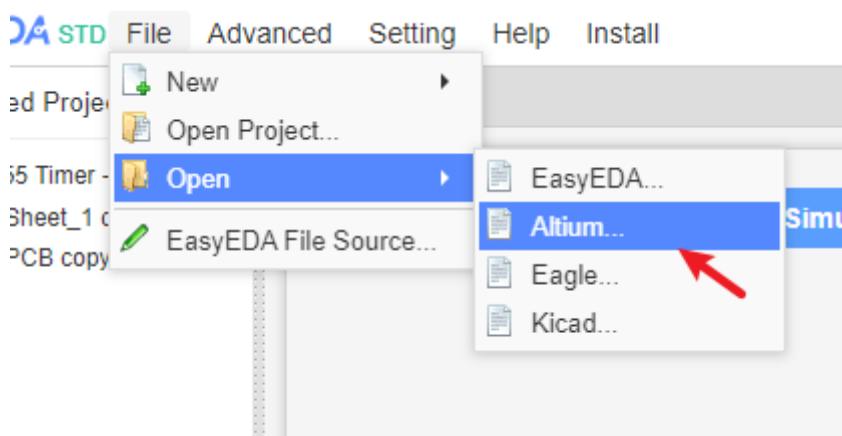
Some design rules EasyEDA doesn't support yet.

## Import Schematic and PCB

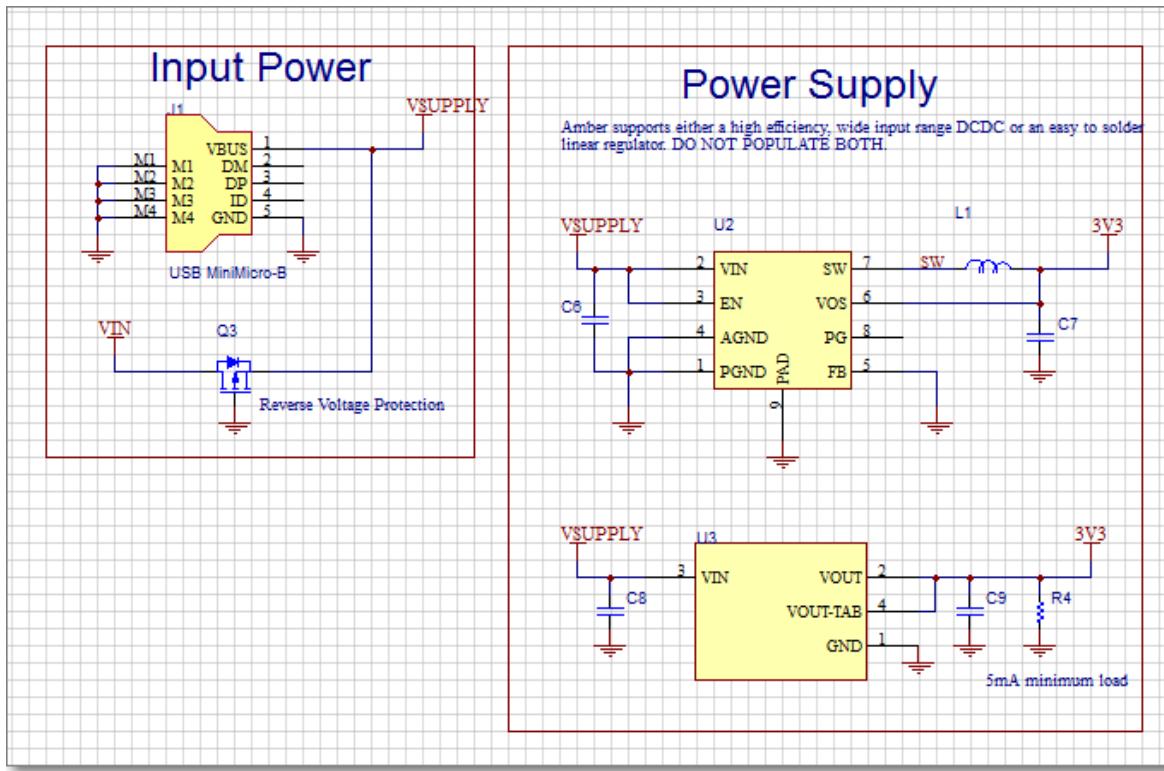
1. You can import Altium Designer's Schematic and PCB files into EasyEDA but only from **ASCII** files, so you need to save the designs as Ascii files like this.



Then import it via: File - Open - Altium...



EasyEDA offers an excellent experience in importing Altium Designer's Schematic and PCB as you can see from the image below of a schematic imported from Altium Designer:

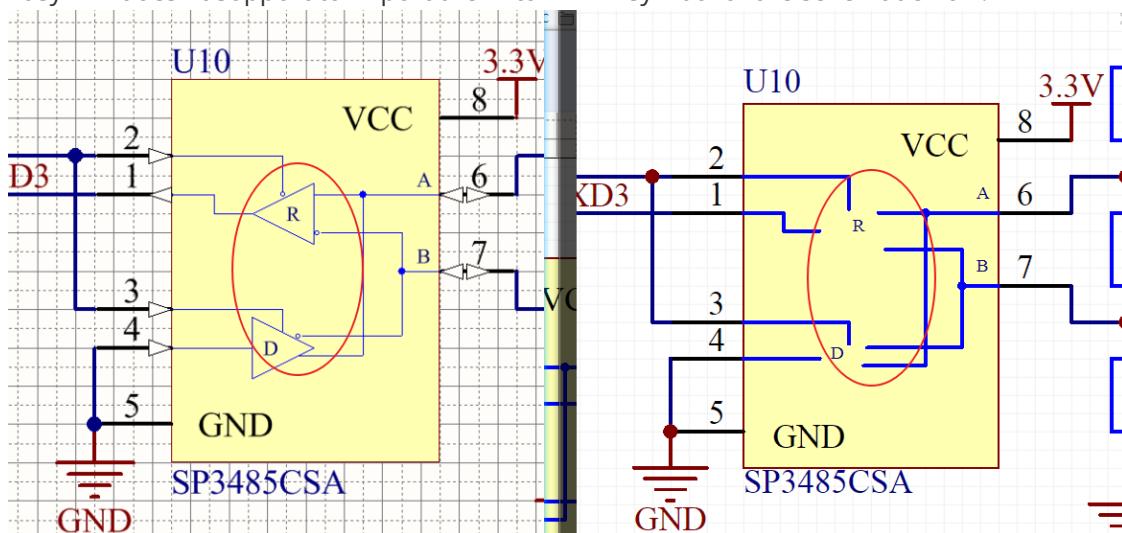


If your schematic and PCB are Protel 99se format files, please open at Altium Designer and save as ASCII format, and then import them. EasyEDA don't support Protel 99se file format directly.

2. If you import Altium schematic found some text became unreachable code, please encode your ASCII file with UTF-8.

#### Notice:

- EasyEDA doesn't support to import the Altium PCB rules now.
- EasyEDA doesn't support to import the Altium PCB inner plane layer, please modify manually after imported.
- EasyEDA doesn't support to import the Altium IEEE symbol of the schematic now.



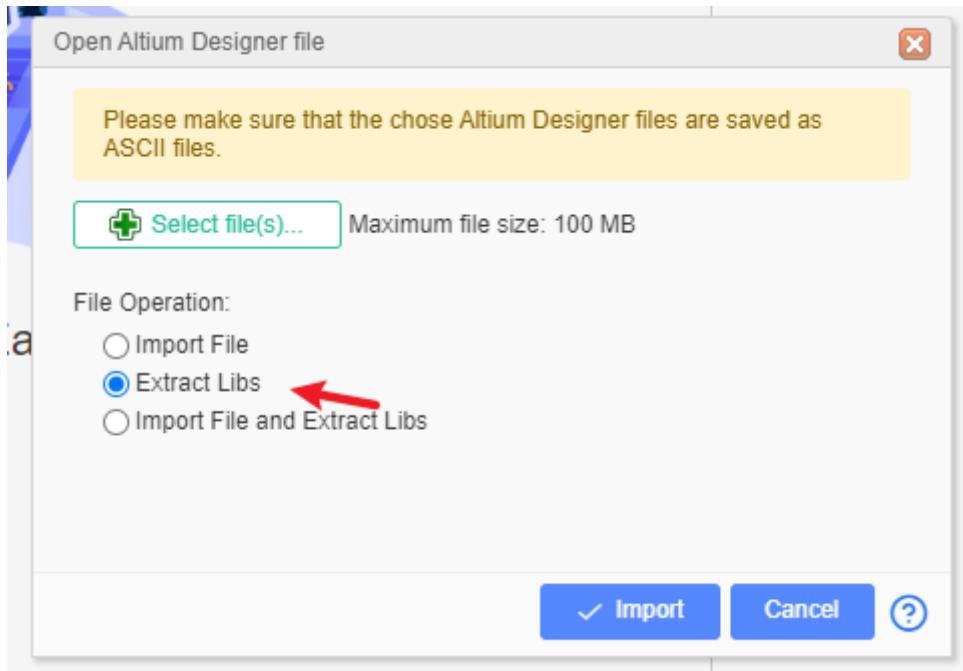
- Please do not export your design to Altium and import it again and again, that will cause some detail missing!!!
- If the Altium file very large that will use a long time to import, suggested remove the copper first before importing.

## Import Altium libraries

Altium Designer's Schematic and Footprint libraries are not available as **ASCII** files, EasyEDA can't import them directly, so how can you import them?

In the Import file from your computer dialog to the right of File Operation; tick the **Extract Libs** option and EasyEDA will extract all of the libs from the Schematic files or PCB Files.

So, if you want to import Altium Designer's Libs, you can add them to your Altium Designer Schematic or PCB and then extract them again into your EasyEDA library.



## Import Eagle

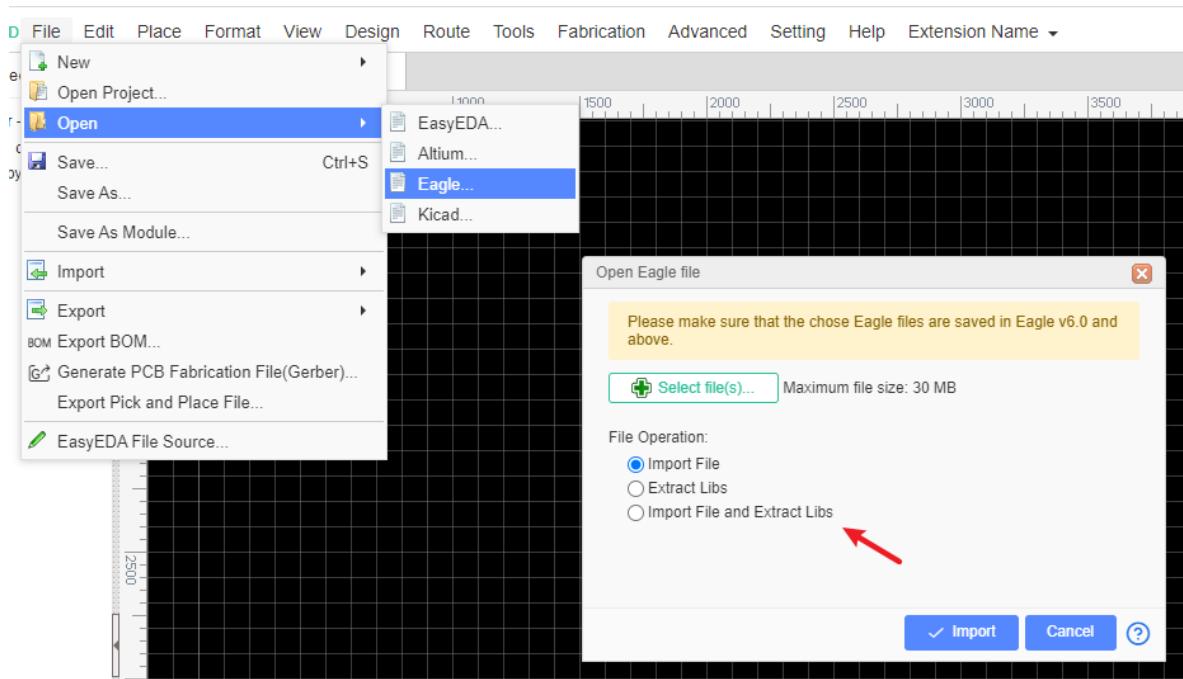
Please refer next section

## Import KiCAD

Please refer next section

## Import Eagle

Eagle Schematic/PCB/libs can be imported, but EasyEDA can only support version 6 and later (6+) because that was when Version 6 Eagle adopted an **ASCII XML** data structure as their native file format.



If your Eagle file can be open in Eagle, but can't be imported in EasyEDA, you can save as a copy with the latest Eagle, and then import it.

If you make sure you have been saved as a copy from v6.0 and greater, but importing still fail, then please edit the Eagle file at Text Editor, find out the Garbled characters, and remove it, and then try again.

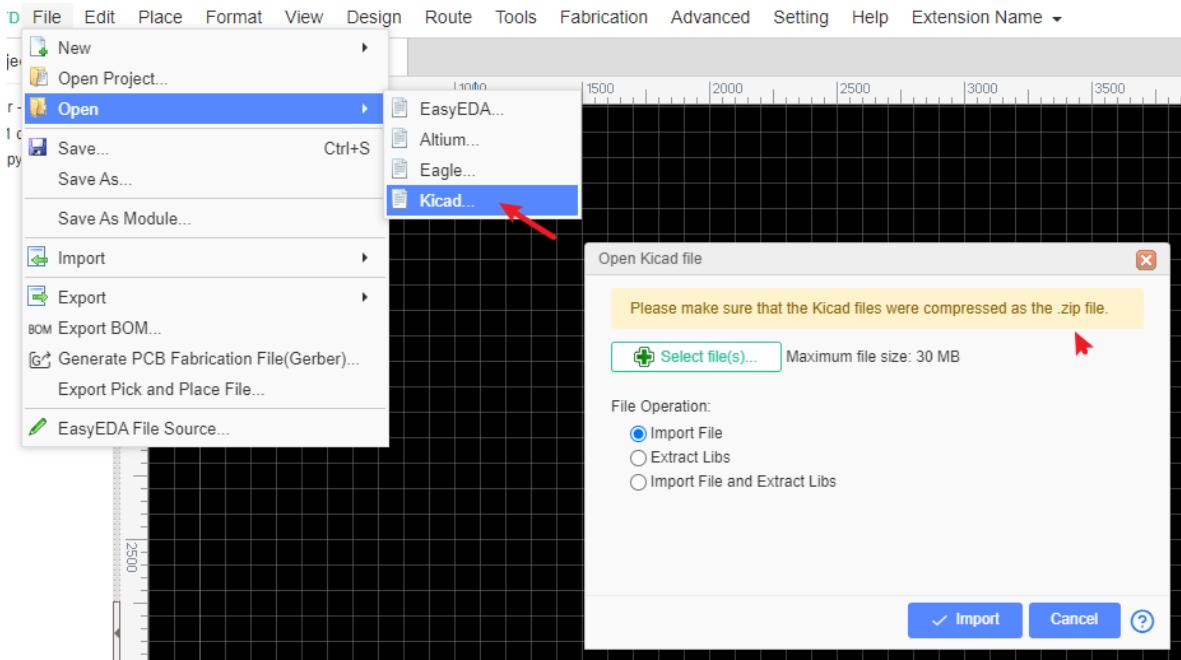
If your schematic needs to update the PCB, please use "Import File and Extract Libs" option, make sure all libraries are imported first.

Some rule or primitive doesn't support, please check carefully after importing.

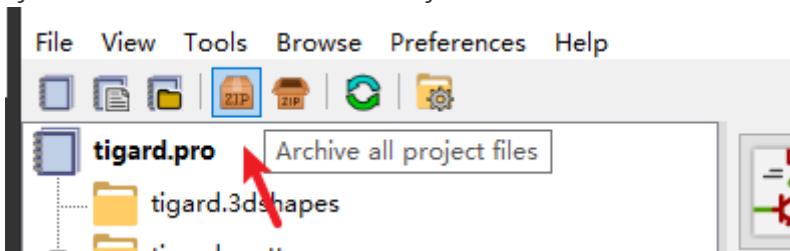
## Import KiCAD

EasyEDA support import KiCAD v4.06 and greater version KiCAD files, if the KiCAD files version less than v4.06, please open them at the latest KiCAD and save as a new one, and then import them.

The KiCAD project files need to be compressed as zip file before importing.



- If you only want to import the PCB, you just need to ZIP the PCB file and then import it.
- If you want to import the schematic, you must ZIP the schematic and symbols together, suggested using KiCAD archive tool when open the project in KiCAD, it will including the symbols in the ZIP file automatically.



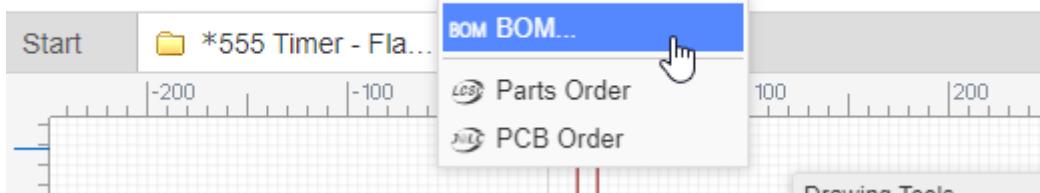
### Notice

- For the KiCAD special symbols such as Power symbol (Power Flag(PWR\_FLAG)), EasyEDA will convert them as the symbol not Netflag, you can delete them if you don't need them.
- The PCB design rule doesn't support yet.
- KiCAD has been update document format since KiCad v5.1.3, if you importing fail, please try the lower version. It is waiting for fixed.

## Export BOM

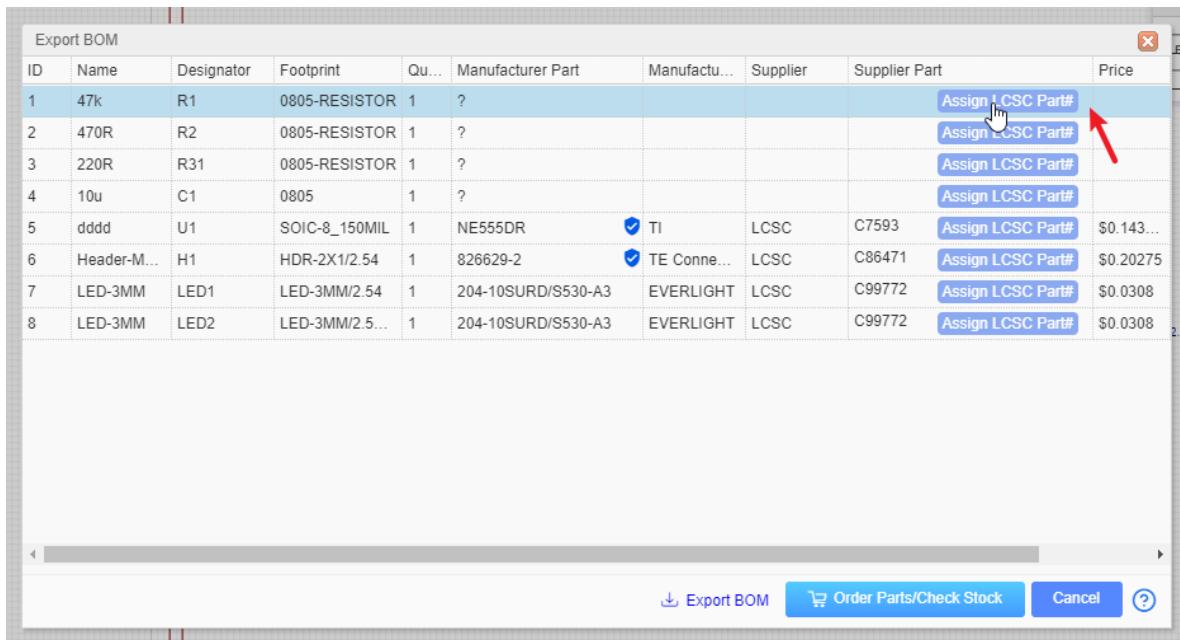
You can export the Bill of Materials (BOM) for the schematic (Document) and PCB, via: "Top Menu - File - Export BOM", or "Top Menu - Fabrication - BOM".

Format View Design Tools Fabrication Advanced Setting Help

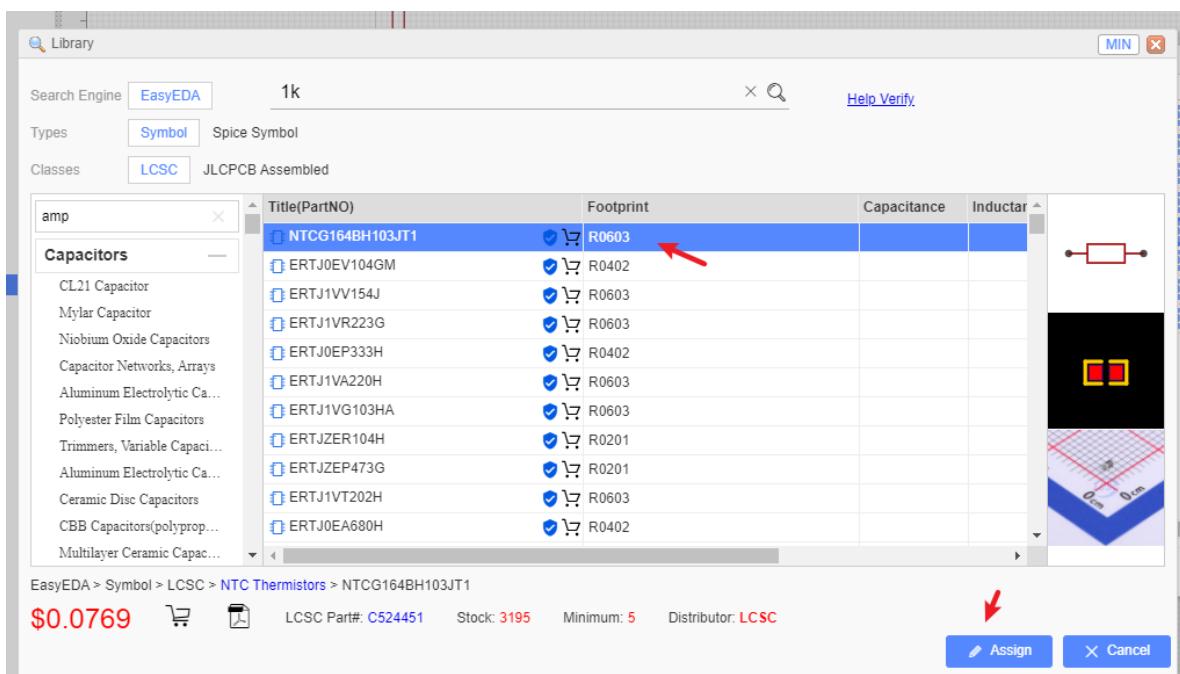


After clicking the BOM export option, the dialog below will open.

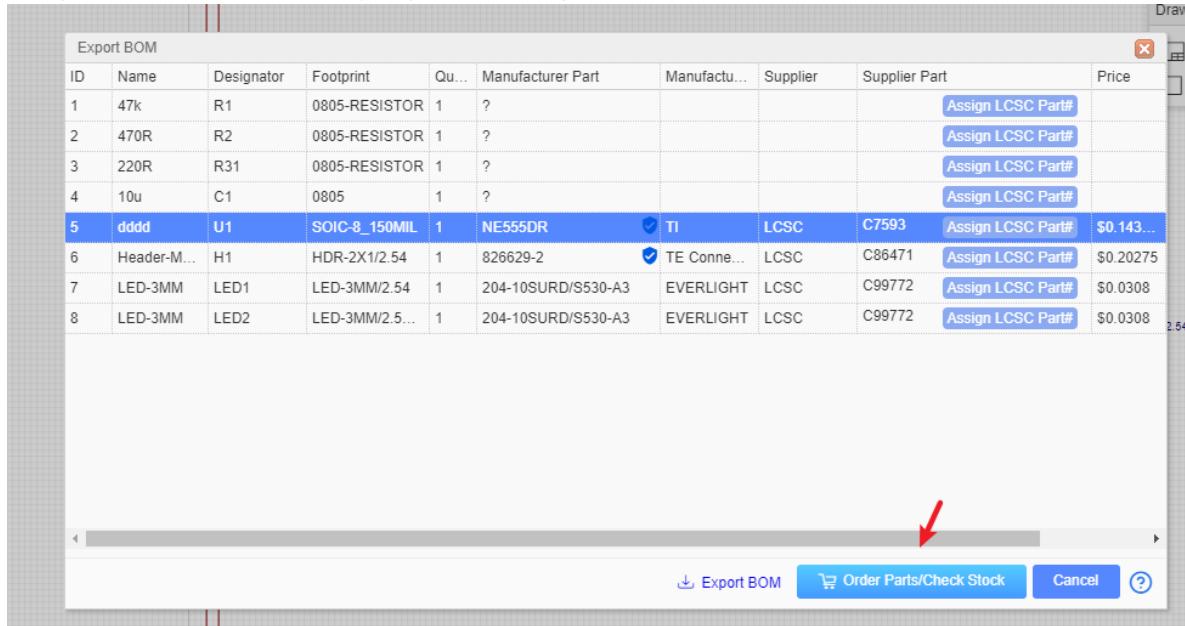
In this dialog, you can click the button to assign LCSC part's order code for your components.



After clicking on the assign icon, the components and footprints search dialog will pop up, and you can choose which component you want to assign.



When you click the "Order Parts/Check Stock" button, we will help you to list all the components of your BOM at LCSC.com (If you haven't login LCSC, you have to login first). If you want to buy the components from LCSC, and you just need to put them to the cart and check out.



You can open the BOM in any text editor or spreadsheet.

|   | A  | B              | C        | D              | E          | F                        | G            | H        | I      | J      |
|---|----|----------------|----------|----------------|------------|--------------------------|--------------|----------|--------|--------|
| 1 | id | value          | quantity | package        | components | Manufacturer Part        | Manufacturer | Supplier | LCSC   | price  |
| 2 | 1  | 150            | 2        | AXIAL-0.3      | R1,R4      | 25121WJ020KT4F           | UniOhm       | LCSC     | C45278 | \$0.02 |
| 3 | 2  | 22k            | 2        | AXIAL-0.3      | R2,R3      | 25121WF300LT4F           | UniOhm       | LCSC     | C16074 | \$0.03 |
| 4 | 3  | 22u            | 2        | CAP-D3.0XF1.5  | C1,C2      | 1812B225K500NT           | FH           | LCSC     | C28503 | \$0.28 |
| 5 | 4  | 204-10UYC/S531 | 2        | LED-3MM/2.54   | LED1,LED2  | 67-21S/KK3C-H2727QAR3LEC | EVERLIGHT    | LCSC     | C73540 | \$0.04 |
| 6 | 5  | 2N3904         | 2        | TO-92(TO-92-3) | Q1,Q2      | MURA220T3G               | ON           | LCSC     | C37995 | \$0.17 |
| 7 |    |                |          |                |            |                          |              |          |        |        |

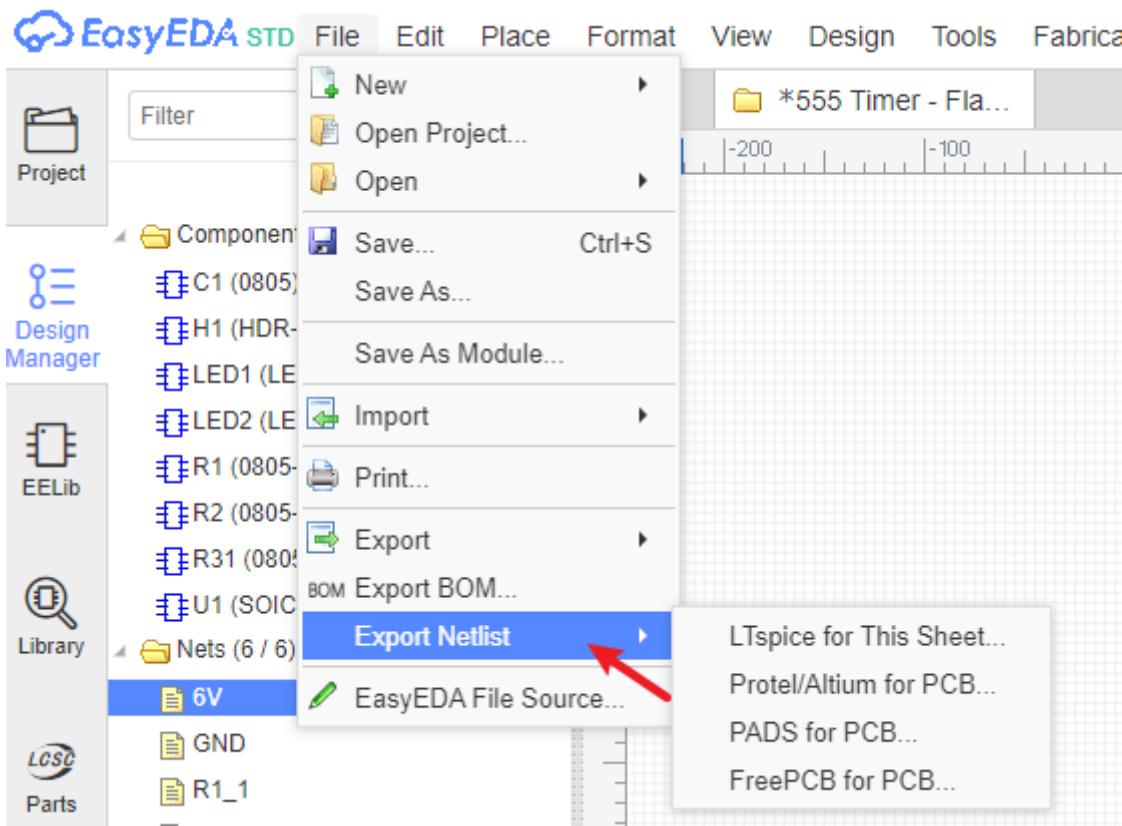
### Notice:

- If your project has schematic and PCB, the BOM data will come from schematic; if the project only has PCB, the BOM data will come from PCB.
- In order to support multiple languages, BOM and coordinate files (CSV file) are UNICODE encoded and tab-based. If the CSV file cannot be read by your components vendor or PCB manufacturer, please convert the encoding and change the delimiter.
- Recommended solution: Save as a new CSV file in Excel or WPS. For example, open a CSV file in Excel, click or select: Save As - Other Formats - CSV (Comma Separated) (\*.csv). You can also open the CSV file with any text editor (such as Windows Notepad) and save as ANSI or UTF-8 encoding. If necessary, replace all tabs with commas.

## Export NetList

EasyEDA can export the netlist for the whole active project:

**File > Export NetList > Spice...**



EasyEDA can export a netlist in a variety of formats:

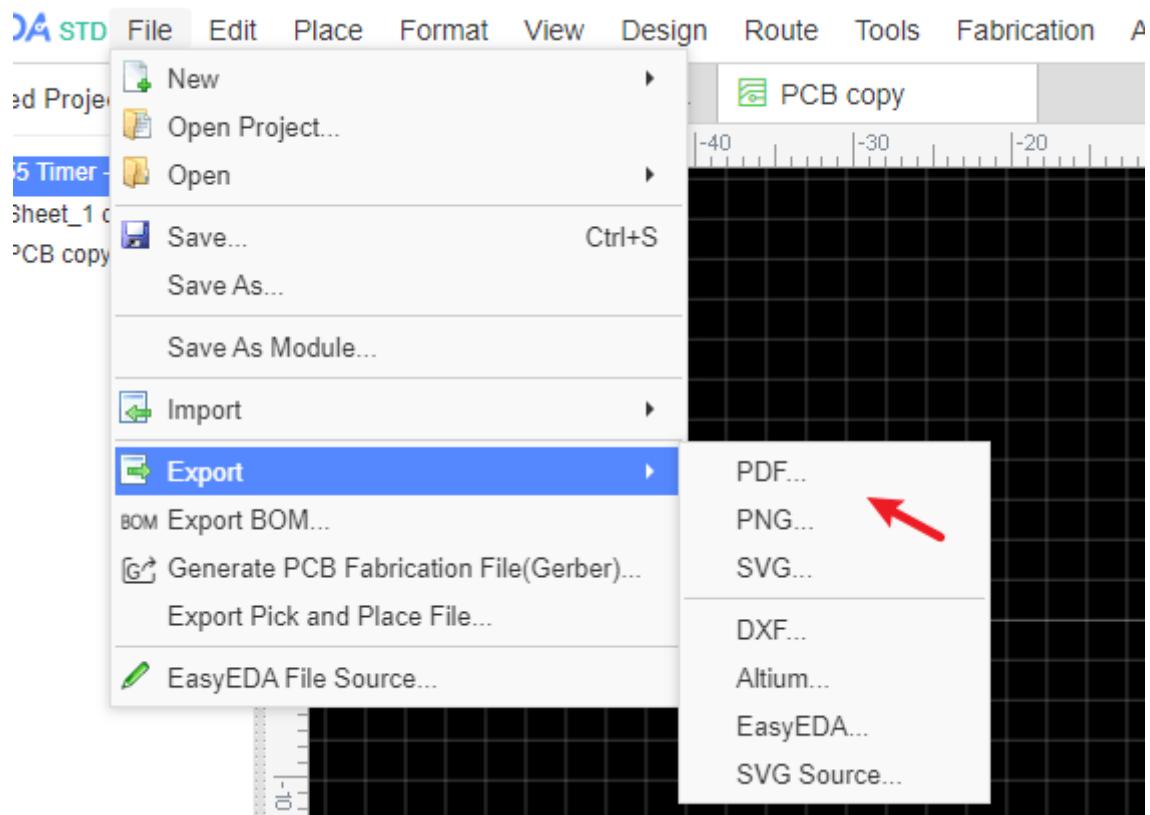
- **LTS spice for this Sheet:** this is a Spice compatible netlist generated by the simulation engine of EasyEDA, It is not normally used as the basis for as a PCB layout.
- **Protel/Altium for PCB:** a PCB netlist in a format that can be imported straight into Altium Designer and it's predecessor, Protel.
- **PADS for PCB:** a PCB netlist in a format that can be imported straight into Pads PCB layout tools.
- **FreePCB for PCB:** a PCB netlist in a format that can be imported straight into FreePCB, a free, open source PCB editor for Windows.

## Export PCB

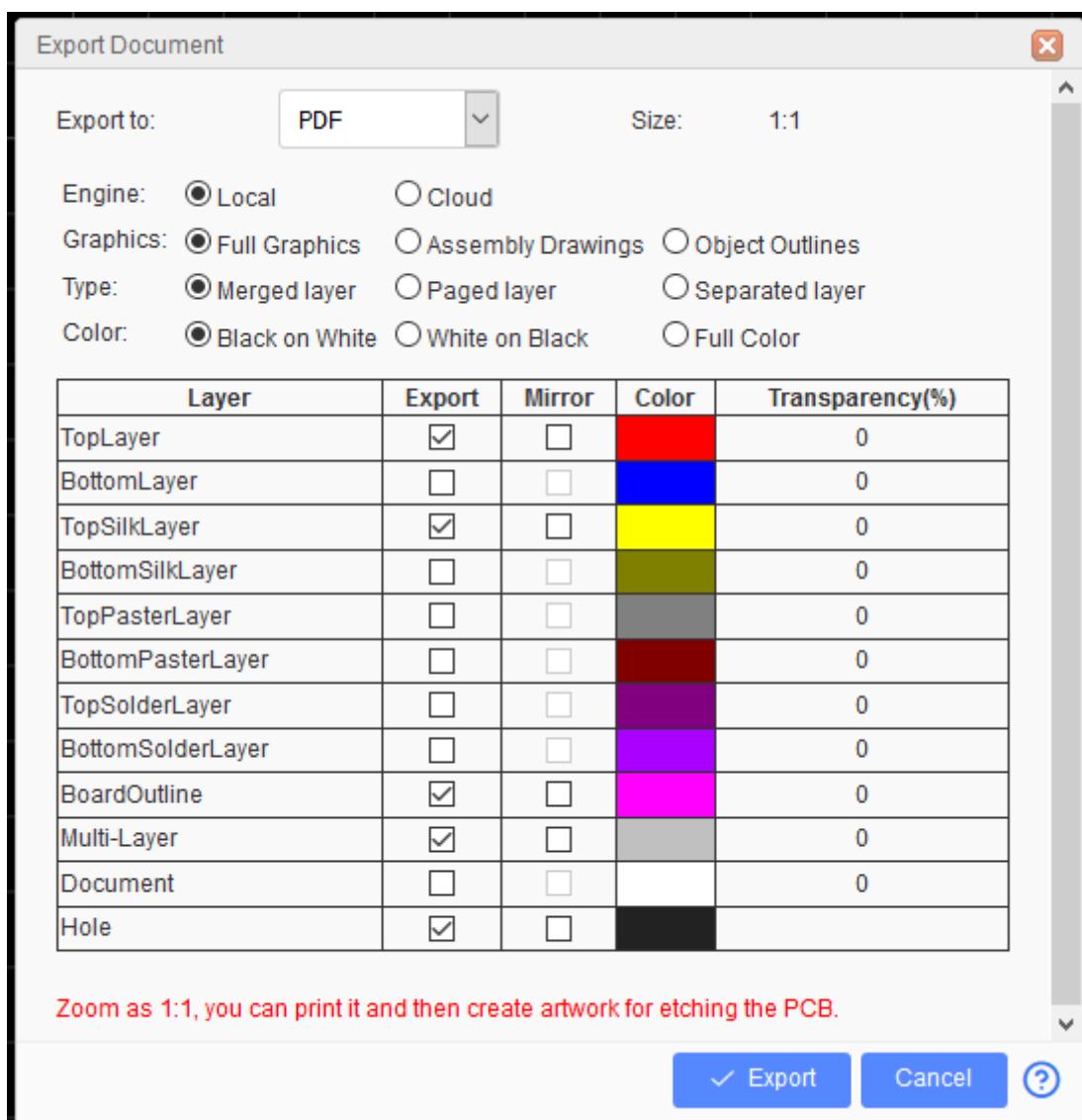
### Export PCB in PDF/PNG/SVG

Exporting a PCB design or footprints from EasyEDA is very similar to exporting a Schematic or a Symbol.

Via: **File > Export > PDF/PNG/SVG...**



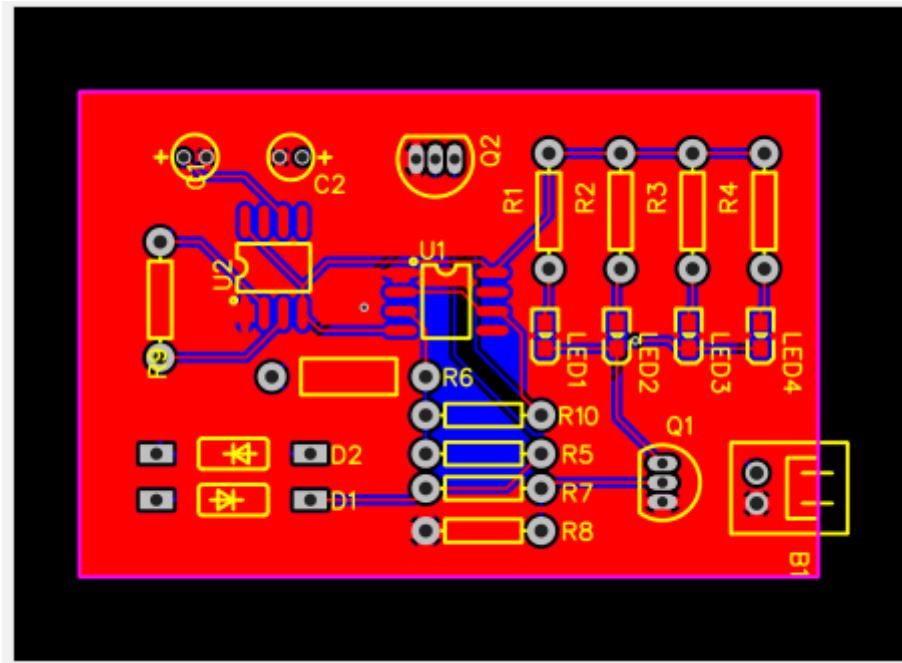
You will open this dialog:



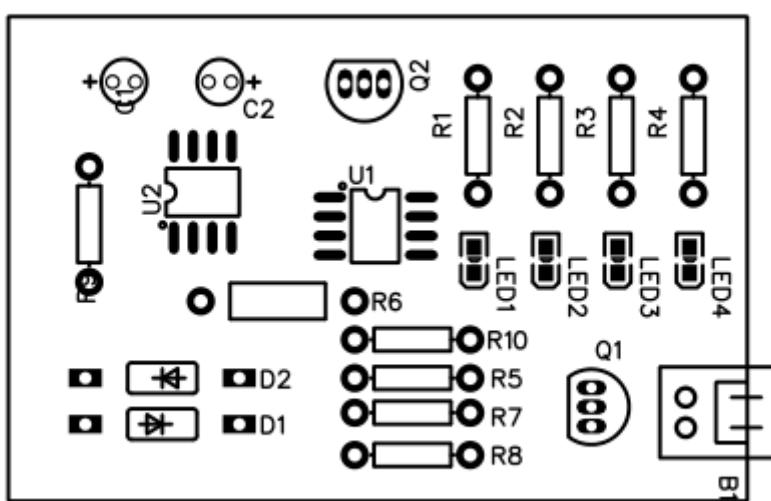
You can select to export in PDF, PNG or SVG format.

**Note:** \*The PDF size is zoom as 1:1 with PCB. \*

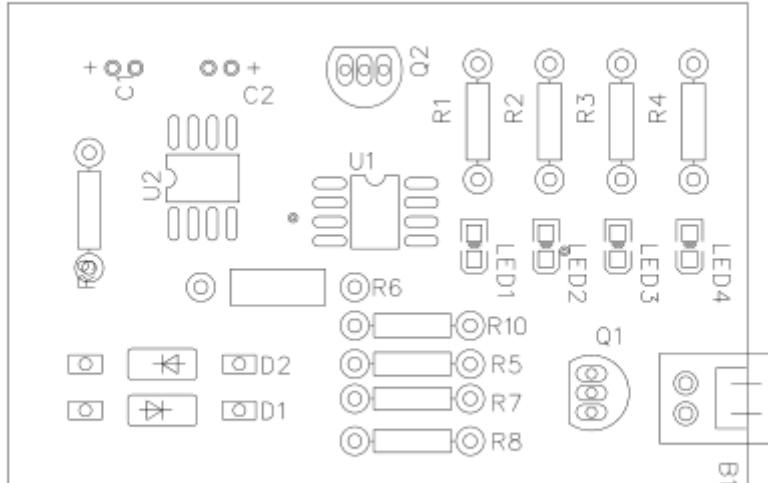
- **Export to:** Support export to PDF, PNG, SVG. If you want to print the PCB 1:1, please choose PDF.
- **Engine:**
  - **Local:** PDF generated by Editor
  - **Cloud:** PDF generated by Cloud Server, in the future, EasyEDA will remove this option.
- **Graphics:**
  - **Full Graphics:** All graphics, objects will be exported.



- **Assembly Drawings:** Only exporting components's prefix and location, hole etc. This is for part assembly.



- **Object Outline:** Only exporting objects' outline, such as Pad and silkscreen outline.



- **Type:**

- **Merged layer:** All selected layers you want to export will be merged in one page.
- **Paged layer:** All selected layers you want to export will be paged in one file.
- **Separated layer:** All selected layers you want to export will be separated in multiple files. Export as a ZIP file.
- **Color:** You can choose "Black on White", "White on Black", "Full Color".
- **Layer:** You can select to print individual layers or selected layers merged into a single file.
- **Mirror:** It is also possible to mirror selected layers for example to show bottom layers in easily readable orientation. Recommend when all your selected layers are bottom type you can enable this option.

If EasyEDA PDF can not satisfy your requirement, please let us know.

[support@easyeda.com](mailto:support@easyeda.com)

And if you generated the Gerber file, you can use the Gerbv to export the PDF, it is very easy.

Via [Gerbv](#)

## Export PCB in Altium Designer Format

The more information please refer at [Export Altium](#)

## Download PCB

Please refer at [Export EasyEDA Source](#)

## Print PCB and Etching

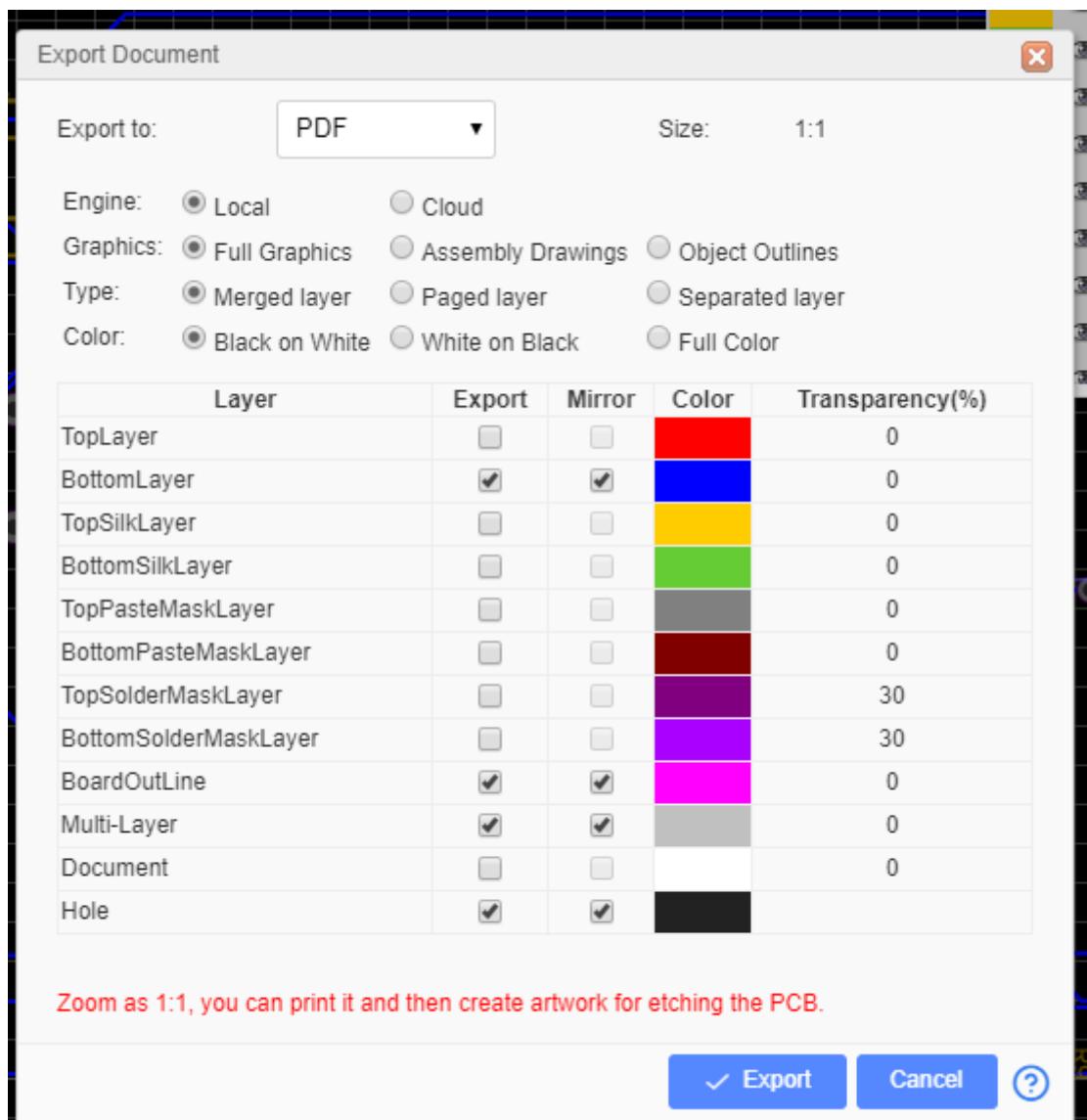
EasyEDA doesn't support to print PCB directly, please export PDF and print.

If you don't want to order your PCBs from EasyEDA then maybe - for single and double sided PCB designs - you might like to try like using some home made PCB tech:

<http://hackaday.com/2012/12/10/10-ways-to-etch-pcb-at-home/>

So here's how you can print your PCB layer by layer and then etch it onto a PCB.

Step 1) Export it to PDF, Using: **File > Export > PDF...**

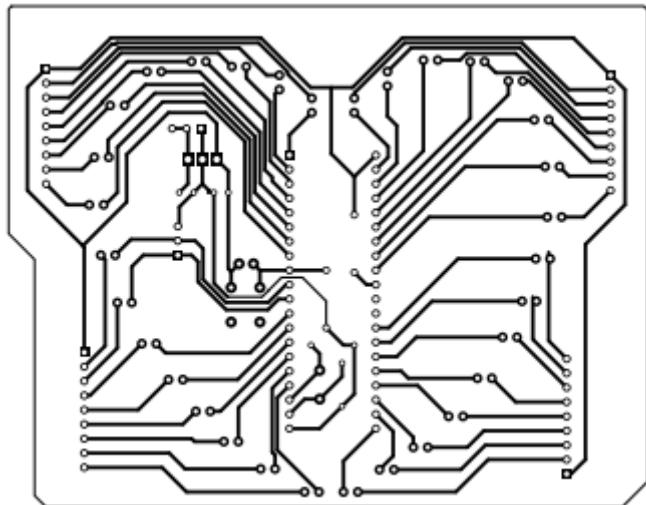


**Note:** Make sure the Colour is Black on White Background.

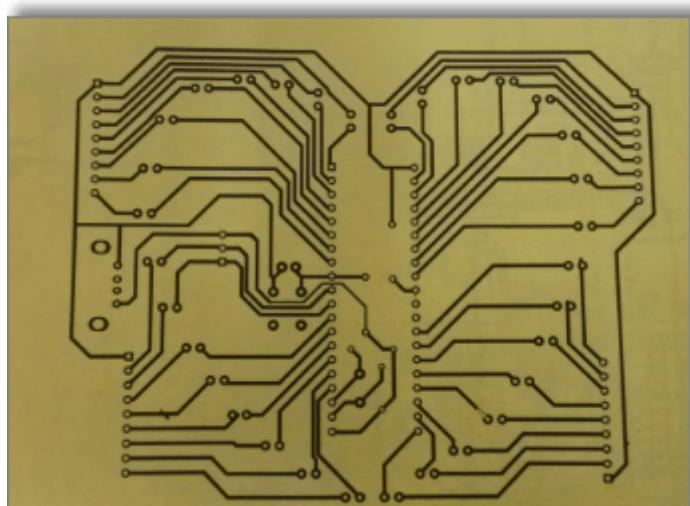
Generally choose the bottom layer. Select if you want to mirror the export as needed.

If you have routed PCB tracks on the top layer, then you need to choose the top layer. Etch PCB by themselves generally need to mirror export printing.

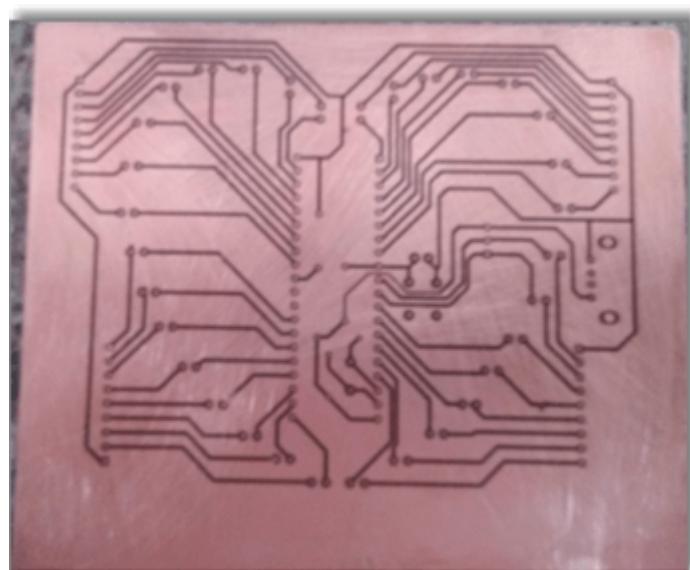
Step 2) Open the pdf file in a viewer



Step 3) Print it to paper

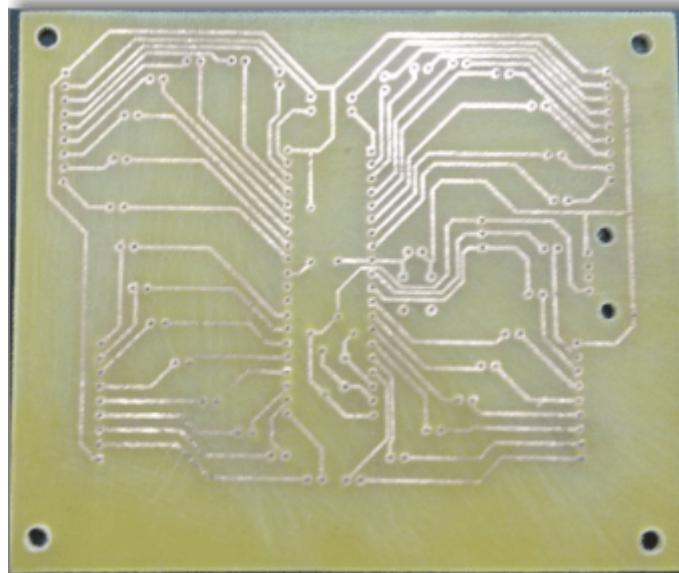


Step 4) Copy it to the copper

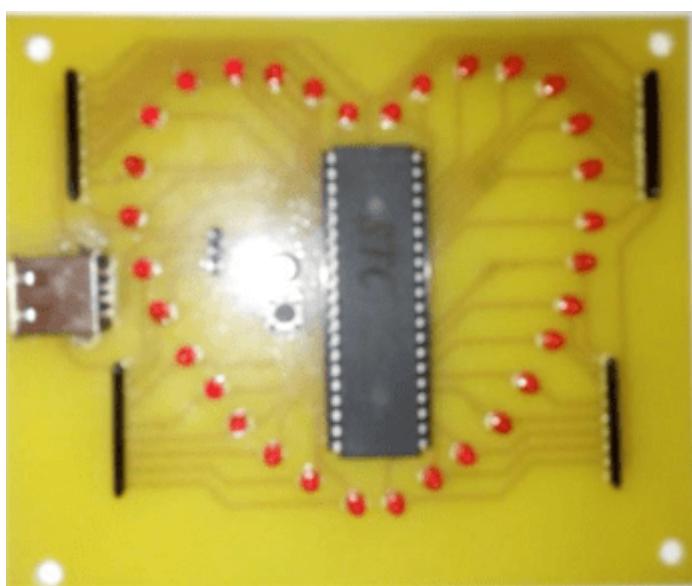


Step 5) Etch it

Step 6) Drill it



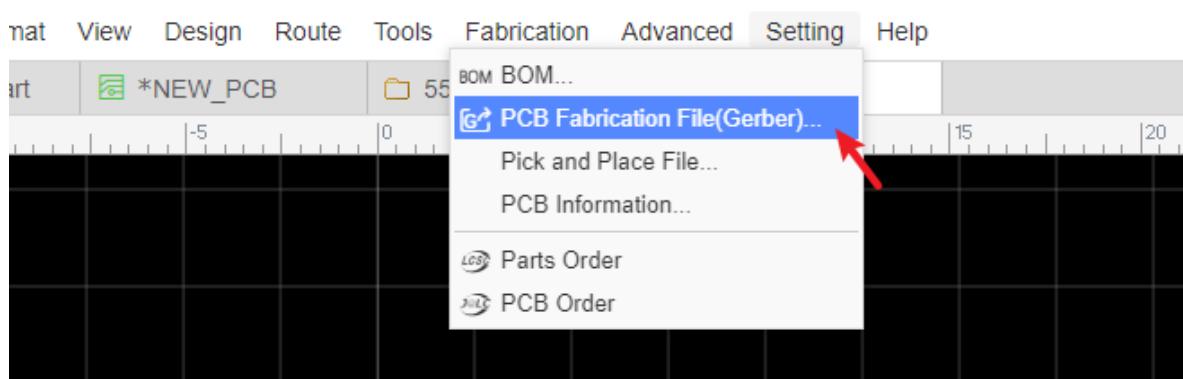
Step 7) Get your soldering iron out!



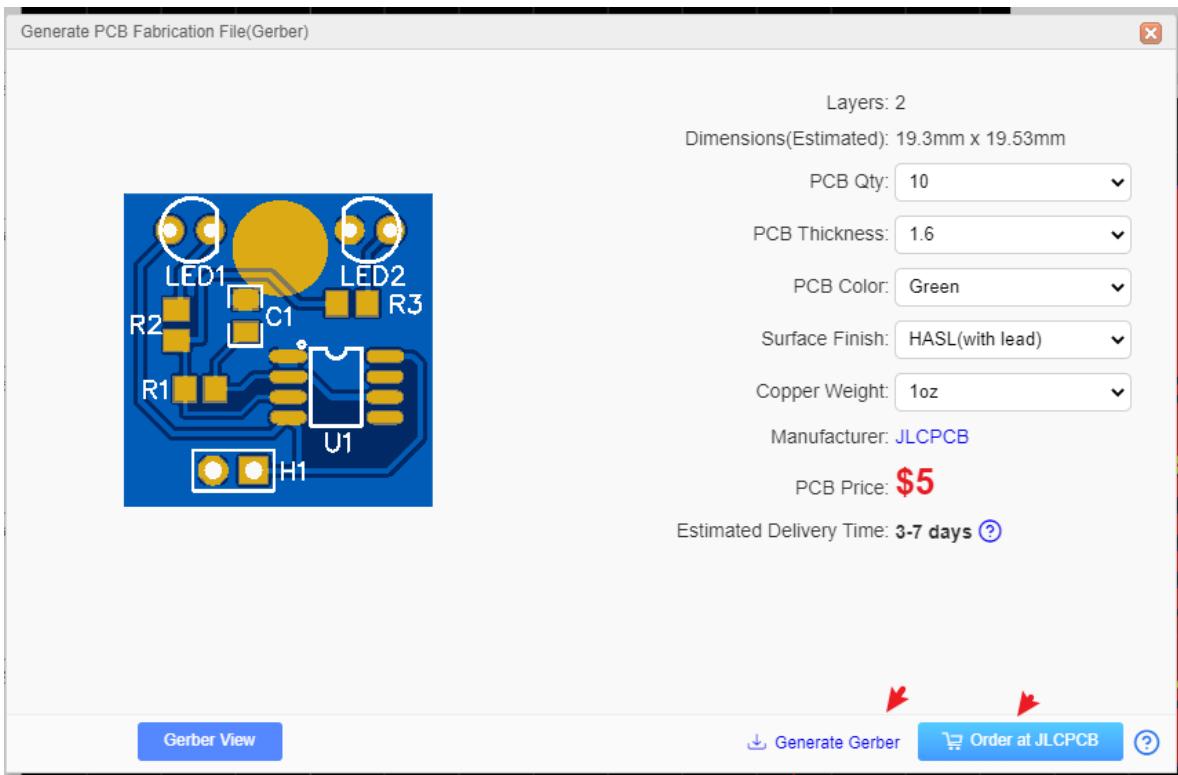
## Generate Fabrication File(Gerber)

### Generate Fabrication File Gerber

When you finish your PCB, you can output the Fabrication Files(gerber file) via: **File > Generate PCB Fabrication File(Gerber)** , or **Fabrication > PCB Fabrication File(Gerber)**.



After clicking, will open the Gerber generate dialog:



You can calculate the price for the PCB order, click SAVE to CART will go to JLCPCB and add your PCB in the cart.

## Gerber file name

The generated Gerber file is a compressed zip file. After decompression, you can see the following files:

- **Gerber\_BoardOutline.GKO:** PCB Border file. The PCB board factory cuts the shape of the board according to this document. The groove drawn by the EasyEDA, the solid region(Type: NPTH) is reflected in the border file after the Gerber is generated.
- **Gerber\_TopLayer.GTL:** Top side copper layer.
- **Gerber\_BottomLayer.GBL:** Bottom side copper layer.
- **Gerber\_Inner1.G1, Gerber\_Inner2.G1...** : Inner copper layer.
- **Gerber\_TopSilkLayer.GTO:** Top silkscreen.
- **Gerber\_BottomSilkLayer.GBO:** Bottom silkscreen.
- **Gerber\_TopSolderMaskLayer.GTS:** Top solder mask. The default board is covered with green oil, and the elements drawn on this layer correspond to the top layer's area will not be covered with oil.
- **Gerber\_BottomSolderMaskLayer.GBS:** Bottom solder mask. The default board is covered with green oil, and the elements drawn on this layer correspond to the bottom layer's area will not be covered with oil.
- **Gerber\_Drill\_PTH.DRL:** Plated drill through hole layer. This document shows the location of the hole where the inner wall needs to be metallized.
- **Gerber\_Drill\_NPTH.DRL:** Non-Plated drill through hole layer. This document shows the location of the hole where the inner wall don't need to be metallized.
- **Gerber\_TopPasteMaskLayer.GTP:** Top Paste Mask, for the stencil.
- **Gerber\_BottomPasteMaskLayer.GBP:** Bottom Paste Mask, for the stencil.
- **ReadOnly.TopAssembly:** Top Assembly, read only, doesn't affect the PCB manufacture.
- **ReadOnly.BottomAssembly:** Bottom Assembly, read only, doesn't affect the PCB manufacture.

- **ReadOnly.Mechanical:** Record the information on the mechanical layer in the PCB design, and only use it for information recording. By default, the shape of the layer is not manufactured at the time of production. Some board manufacturers use the mechanical layer to make the frame when using Altium file to production. When using Gerber file, it is only used for text identification in JLCPCB. For example: process parameters; V cut path etc. In EasyEDA, this layer does not affect the shape of the border of the board.

#### Notice:

- Before ordering the PCB, please check the gerber at the Gerber view as below.
- The Gerber files are generated by browser, please use the browser inner downloader to download!

## Gerber View

Before sending Gerber to the factory, please use gerber viewer to check the Gerber carefully.

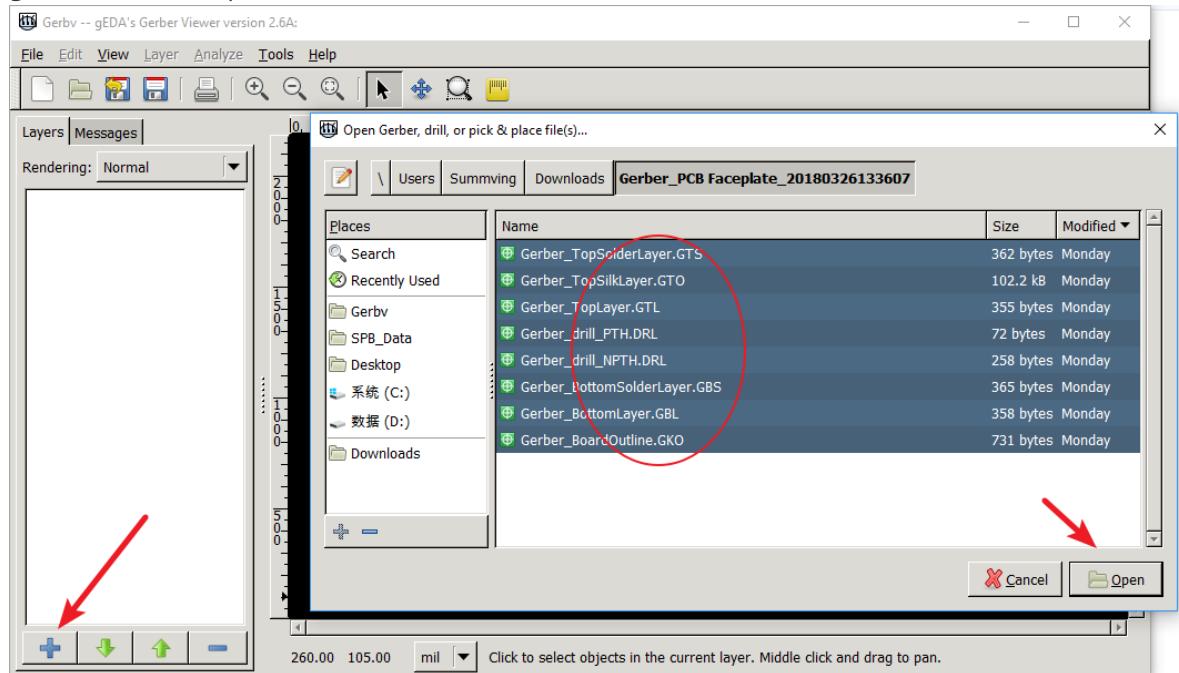
local gerber viewer you can use such as: Gerbv, FlatCAM, CAM350, ViewMate, GerberLogix etc.

Gerber viewer recommend Gerbv:

- Project page:<http://gerbv.geda-project.org/>
- Download: <https://sourceforge.net/projects/gerbv/files/>

How to use Gerbv:

- 1.Download Gerber zip file, and download Gerbv, unzip Gerber file and run the Gerbv;
- 2.Click the + button at the Gerbv dialog bottom-left corner, open the gerber folder, select all the gerber files, and open.



- 3.And then zoom, measure, check every layer, check drill holes and location. etc.

FlatCAM is a nice tool too: <http://flatcam.org/>

FlatCAM lets you take your designs to a CNC router. You can open Gerber, Excellon or G-code, edit it or create from scratch, and output G-Code. Isolation routing is one of many tasks that FlatCAM is perfect for. It's open source, written in Python and runs smoothly on most platforms.

Free Online Gerber Viewer:

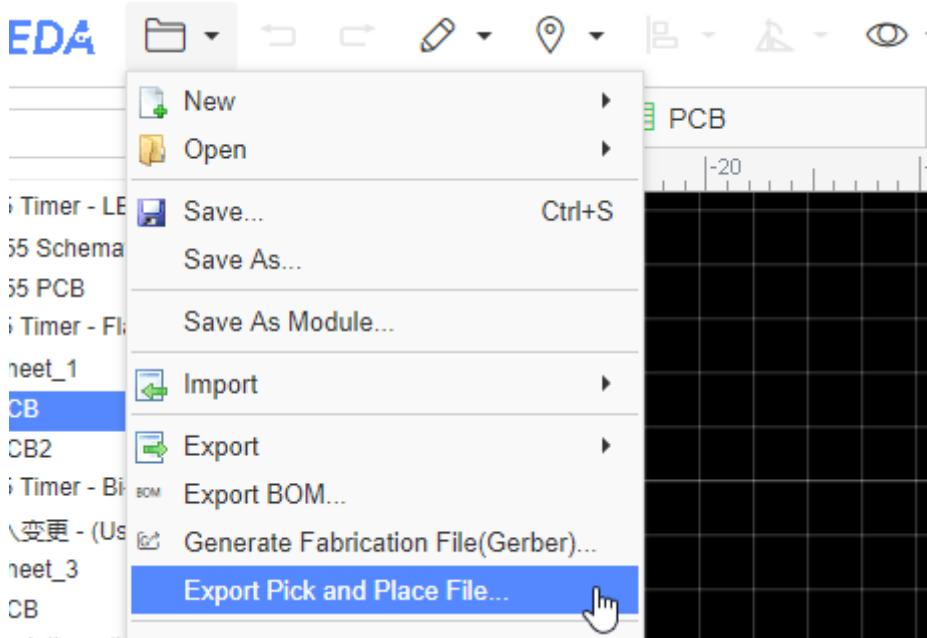
Recommend:

[jlpcb.com](http://jlpcb.com)  
[tracespace.io/view](http://tracespace.io/view)  
[gerber.ucamco.com](http://gerber.ucamco.com)

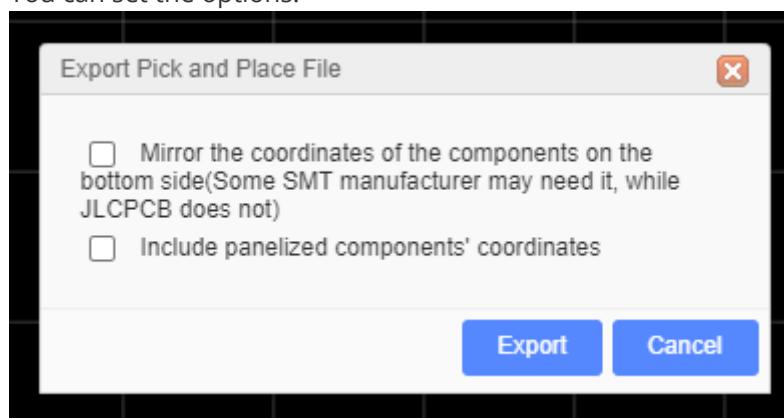
## Export Pick and Place File

In PCB editor, if you want to generate Pick And Place as a CSV file, you can via:

**File > Export Pick and Place File** or Top Menu - Fabrication - Pick and Place File.



You can set the options:



If your PCB has been panelize by the editor, you can enable the "Include panelized components coordinate".

When you open the exported CSV file, you can see:

|   | A          | B                   | C       | D       | E       | F       | G       | H       | I     | J        | K                    | L | M |
|---|------------|---------------------|---------|---------|---------|---------|---------|---------|-------|----------|----------------------|---|---|
| 1 | Designator | Footprint           | Mid X   | Mid Y   | Ref X   | Ref Y   | Pad X   | Pad Y   | Layer | Rotation | Comment              |   |   |
| 2 | LED2       | LED-3MM/2.15.4mm    | 17.27mm | 16.76mm | 17.27mm | 16.67mm | 17.27mm | T       |       | 270      | LED-3MM              |   |   |
| 3 | C1         | 805                 | 7.62mm  | 11.94mm | 7.62mm  | 10.92mm | 7.62mm  | 10.92mm | T     |          | 90 10u               |   |   |
| 4 | U1         | SOIC-8_150\13.31mm  | 7.49mm  | 10.92mm | 9.4mm   | 10.29mm | 9.4mm   | T       |       |          | 0 NES55DR            |   |   |
| 5 | LED1       | LED-3MM/2.14.16mm   | 17.27mm | 2.79mm  | 17.27mm | 2.89mm  | 17.27mm | T       |       |          | 90 LED-3MM           |   |   |
| 6 | H1         | HDR-2X1/2.5 10.16mm | 2.29mm  | 11.43mm | 2.29mm  | 11.43mm | 2.29mm  | T       |       | 270      | Header-Male-2.54_1x2 |   |   |
| 7 | R1         | 0805-RESIST\4.76mm  | 7.37mm  | 3.81mm  | 7.37mm  | 3.81mm  | 7.37mm  | T       |       |          | 0 47k                |   |   |
| 8 | R2         | 0805-RESIST\3.3mm   | 11.36mm | 3.3mm   | 10.41mm | 3.3mm   | 10.41mm | T       |       |          | 90 470R              |   |   |
| 9 | R3         | 0805-RESIST\14.29mm | 12.7mm  | 15.24mm | 12.7mm  | 15.24mm | 12.7mm  | T       |       |          | 180 220R             |   |   |

This file support two units "mm" and "mil", it is following the PCB unit setting.

There is an option "Mirror the coordinates of the components on the bottom side(Some SMT manufacturer may need it, while JLCPCB does not)", you can check with your SMT manufacturer, the mostly SMT manufacturer doesn't need it.

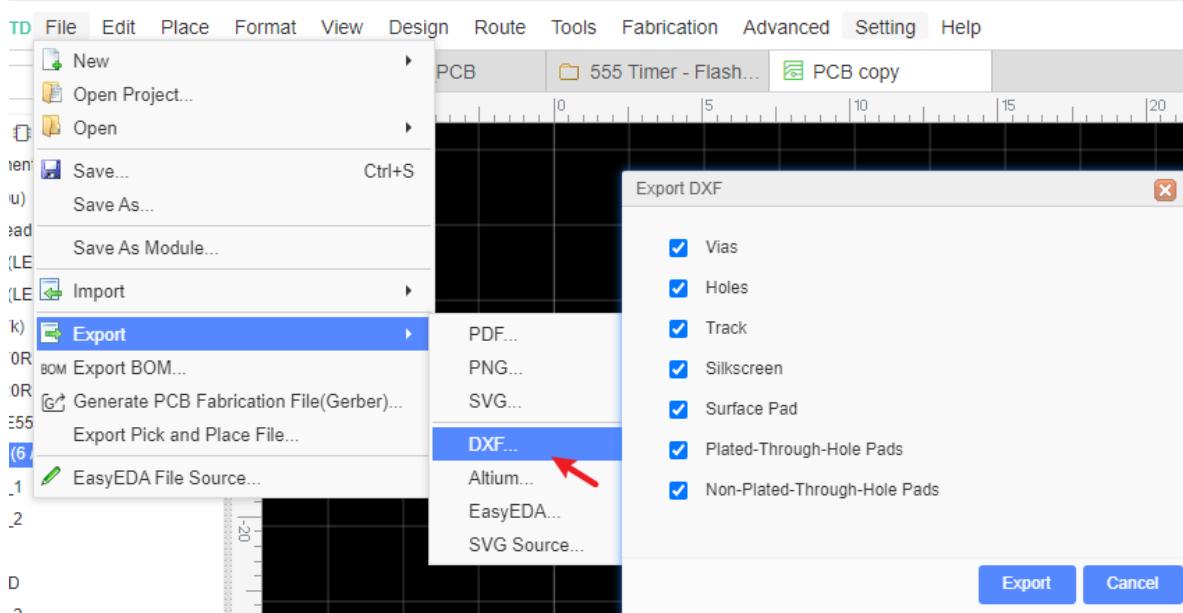
#### Notice:

- In order to support multiple languages, BOM and Pick and Place files (CSV file) are UNICODE encoded and tab-based. If the CSV file cannot be read by your components vendor or PCB manufacturer, please convert the encoding and change the delimiter.
- Recommended solution: Save as a new CSV file in Excel or WPS. For example, open a CSV file in Excel, click or select: Save As - Other Formats - CSV (Comma Separated) (\*. csv). You can also open the CSV file with any text editor (such as Windows Notepad) and save as ANSI or UTF-8 encoding. If necessary, replace all tabs with commas.

## Export DXF

EasyEDA support to export PCB to DXF.

At present only support Board Outline, Hole, NPTH, PTH etc.



EasyEDA does't support to export the DXF which is seperated the layers and objects.

Other side, you can export the PDF, and the using the CAD tool convert PDF to DXF.

# Export Altium Designer Format

EasyEDA support exporting the schematics and PCB in Altium Designer format.

The "export to Altium" function is beta now, Please check carefully after exported the design to Altium, EasyEDA cannot guarantee that is no errors!!! EasyEDA does not bear any loss due to library errors and format conversion!!! If you do not agree, please do not carry out Altium export!!!

If you want to order the PCB please generate the Gerber instead of exporting to Altium! Please do not export your design to Altium and import it again and again, that will cause some details missing!!!

Doesn't support Altium 19 yet, please open exported file at Altium 18 and less, recommend Altium 17

If you find out some incorrect detail, please contact us to fix, including detail and files.

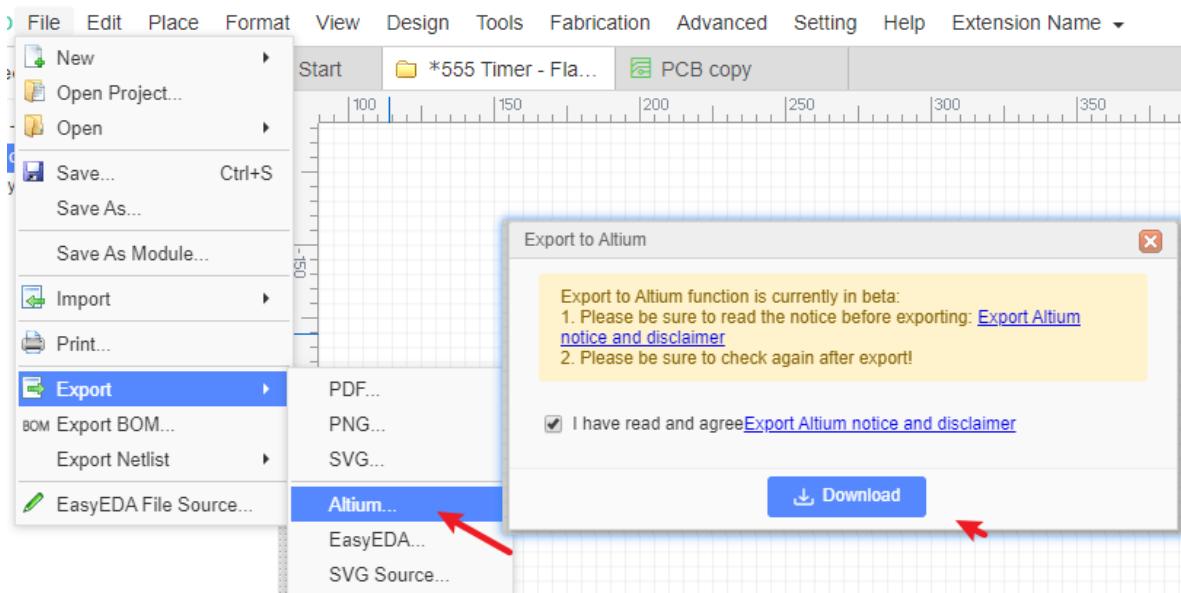
[support@easyeda.com](mailto:support@easyeda.com)

When exporting, you don't need to save document at firstly, but you need to login.

## Exporting Schematics In Altium Designer Format

EasyEDA support exporting the schematics in Altium Designer format.

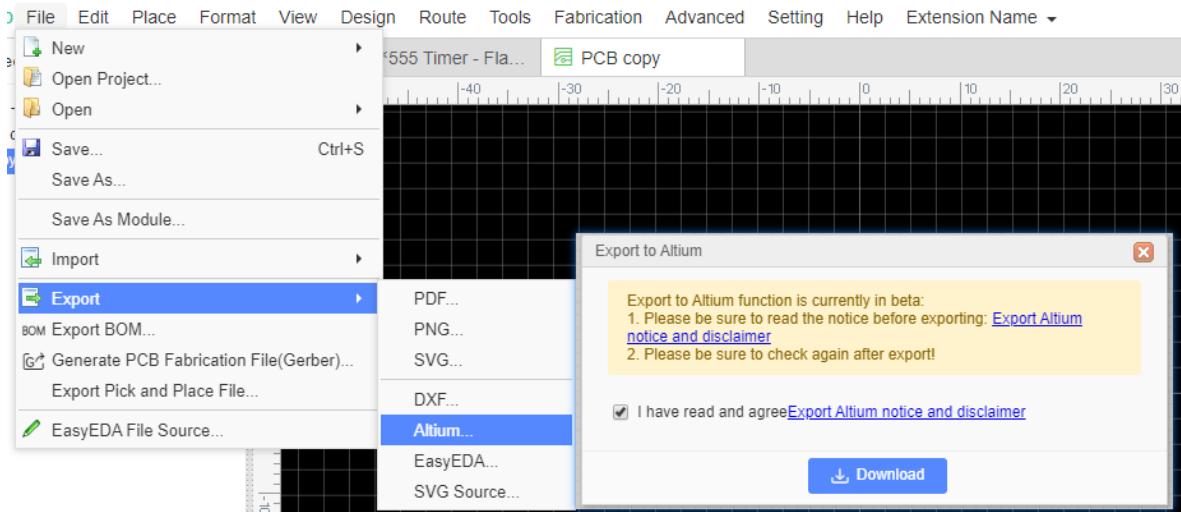
Via "File > Export > Altium...", and click the "Download" you will get a .schdoc file.



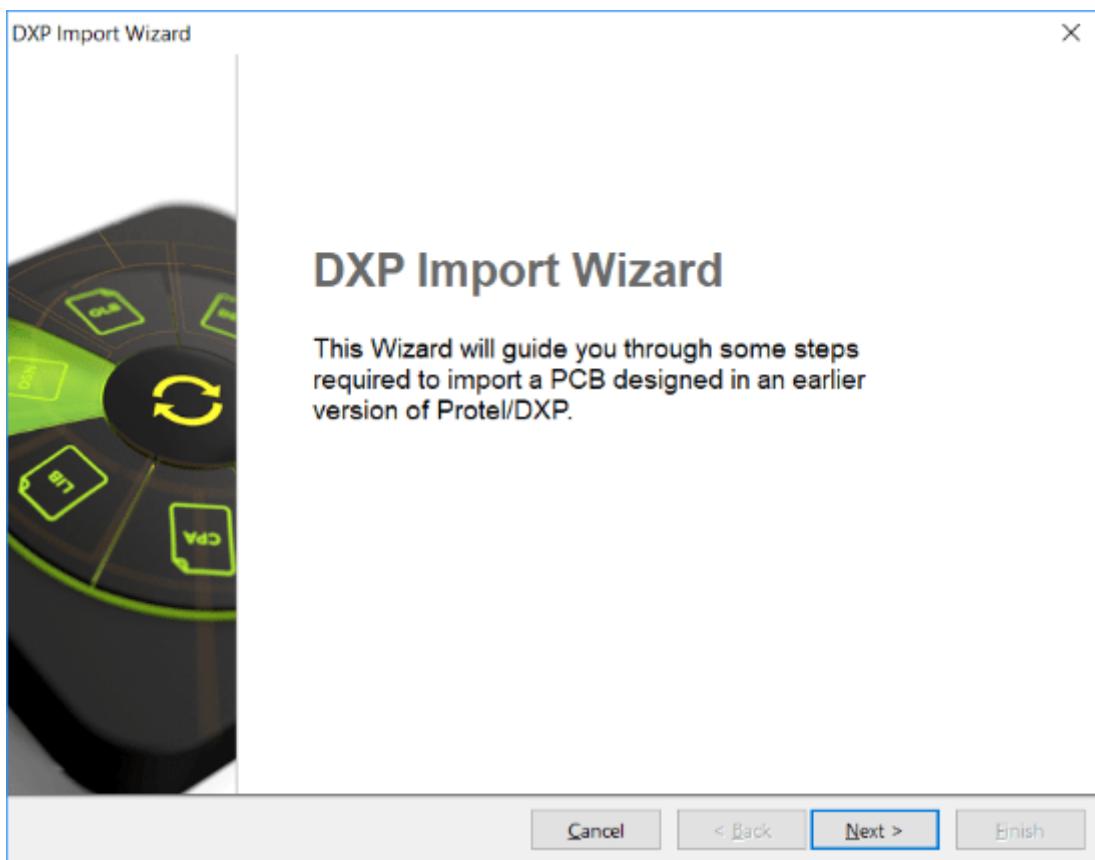
## Exporting PCB in Altium Designer Format

EasyEDA support exporting the PCB in Altium Designer format.

Via "File > Export > Altium..." you will get a .pcbdoc file.



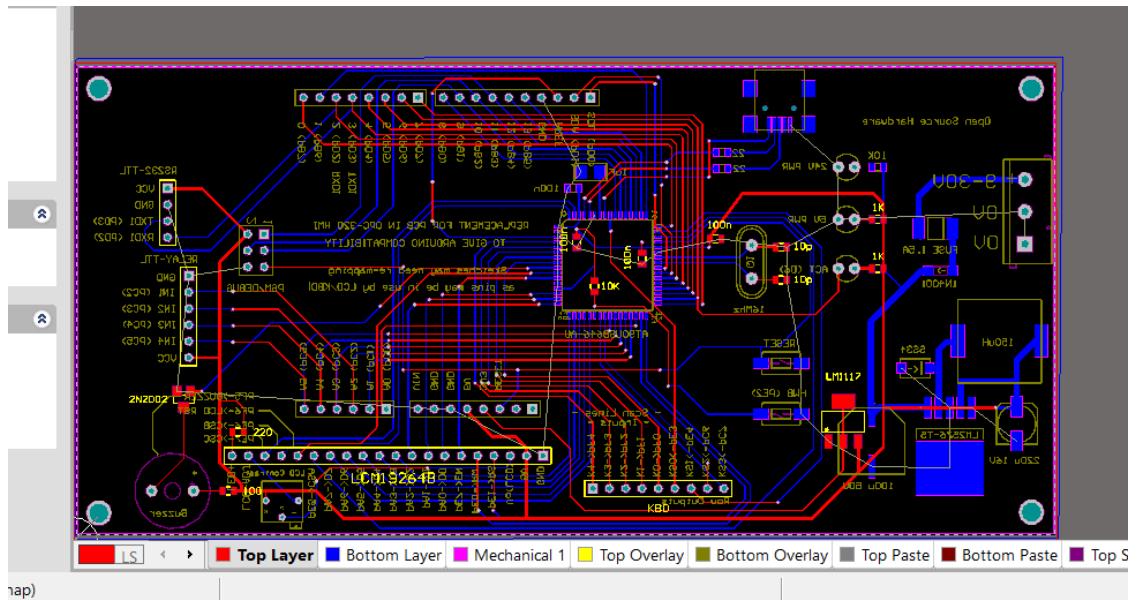
When open the exported PCB file at Altium Designer, there will open a dialog of DXP Import Wizard, don't worry, just cancel it to continue.



## Known Issue:

- **1. No Copper Area fill data.**

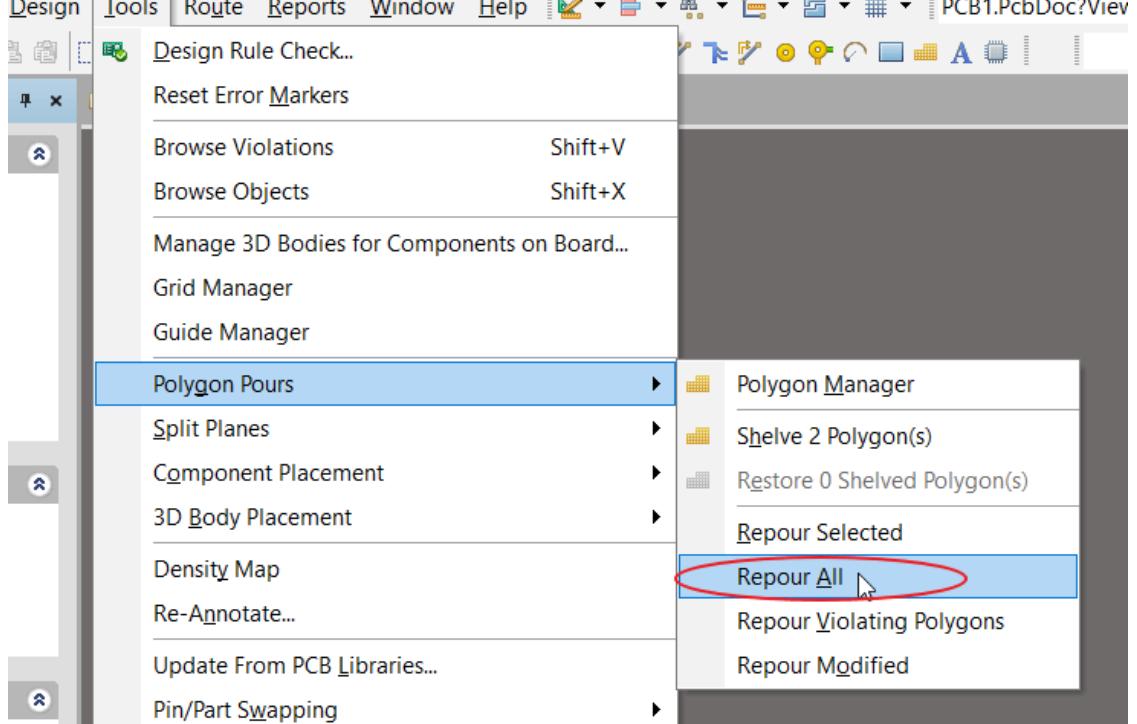
And then, you will see the PCB file, which looks like without copper area as below:



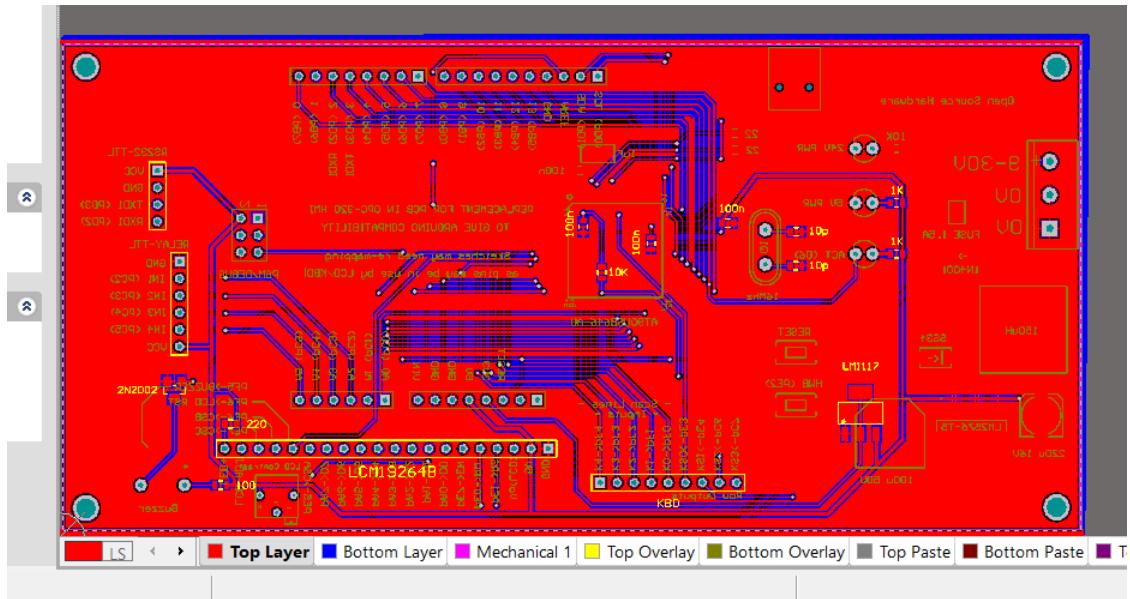
You need to repour all polygons at Altium Designer. Via: **Tools > Polygon Pours > Repour All**:

All:

ee Documents. Not signed in.



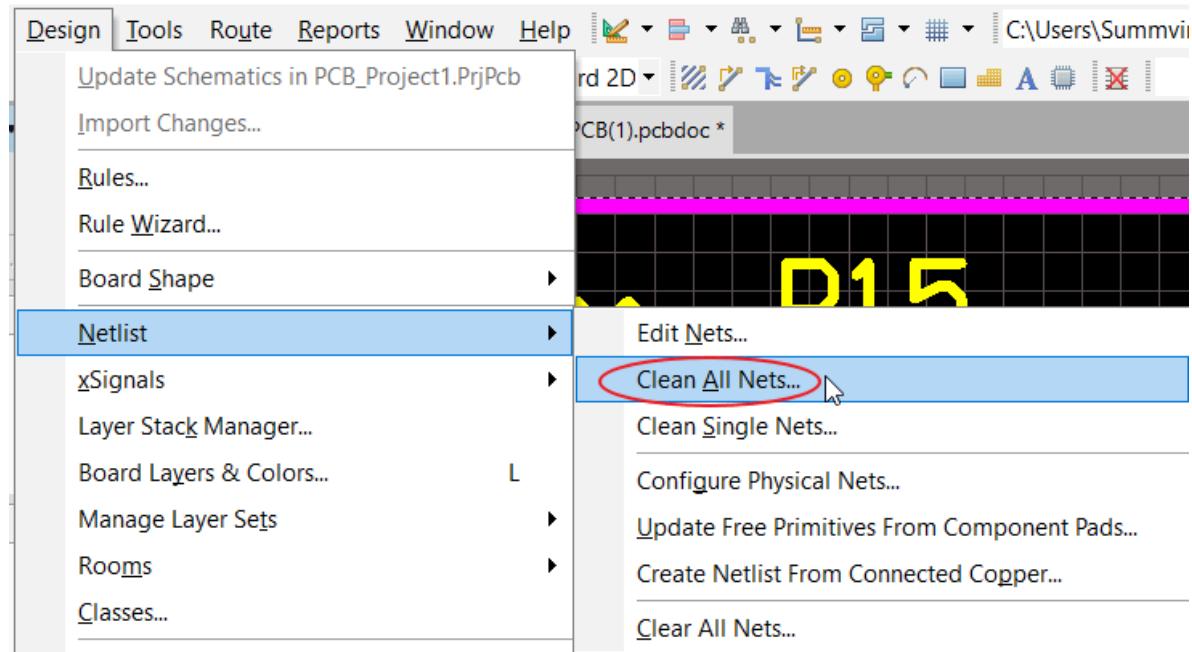
And the last, save it.



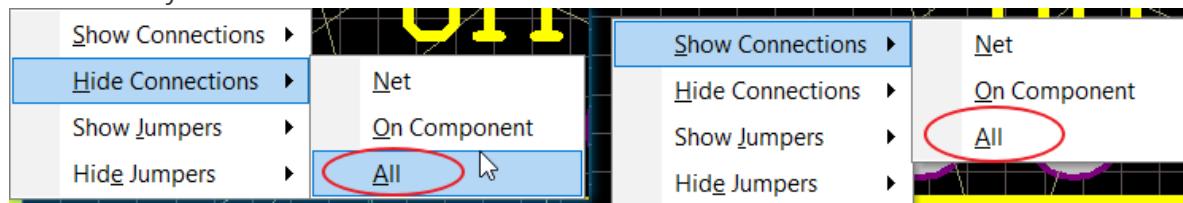
- **2. No Ratlines.**

If you export the PCB without ratlines, you need to show all connections first before routing :

Via: **Design > Netlist > Clean All Nets** (**D > N > A**), and then (**V > C > S**)



Or use hotkey: **N > H > A** and then **N > S > A**:



- **3. Inner layer Plane Zone doesn't export perfectly.**

You need to rebuild the plane zone, and re-assgin plane zone's net.

- **4. Doesn't support DRC rule.**

Please check the DRC manually.

- **5. The text may be changed.**

Because of the font family, some text maybe will change the position. And it maybe will display incorrect, please modify the text manually.

## Exporting Footprint and Symbol in Altium Designer Format

EasyEDA don't support to export the Symbol or Footprint as Altium Designer library format, but you can place the libraries to the schematic or PCB, and export that in Altium Designer format, and then extract the libraries at Altium Designer.

**EasyEDA does not bear any loss due to library errors and format conversion!!! If you do not agree, please do not carry out Altium export!!!**

## Export SVG Source

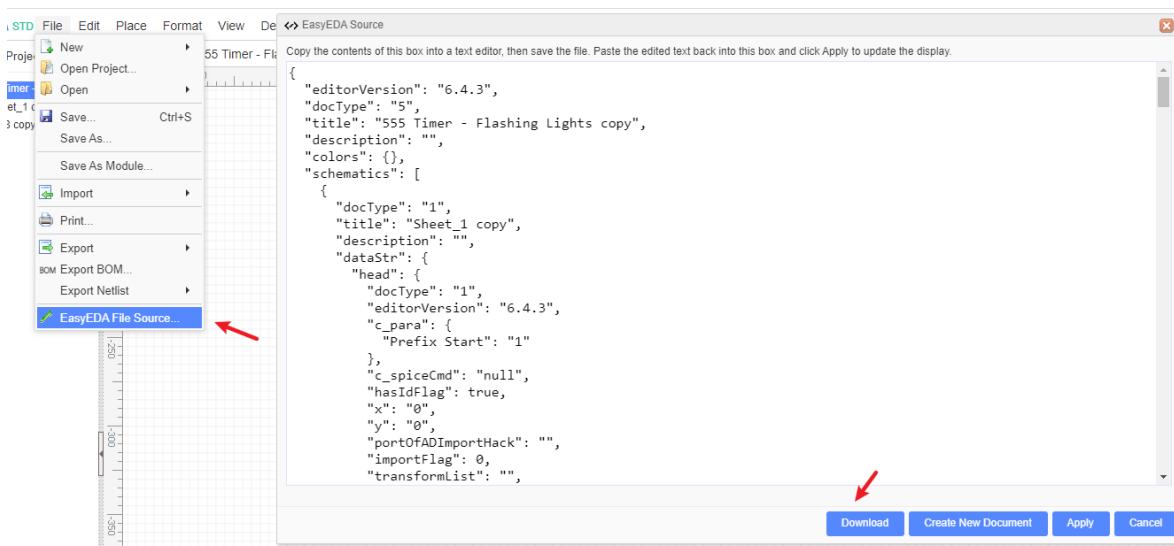
EasyEDA support to export or edit SVG source.

You can create an SVG source file via:

**File > Export > SVG source...**

then copy the contents of this box into a text editor and save the file with a .svg extension. You can edit it in [Inkscape](#) or open it in your browser.

This solution doesn't need an Internet connection you can use it off-line on EasyEDA.



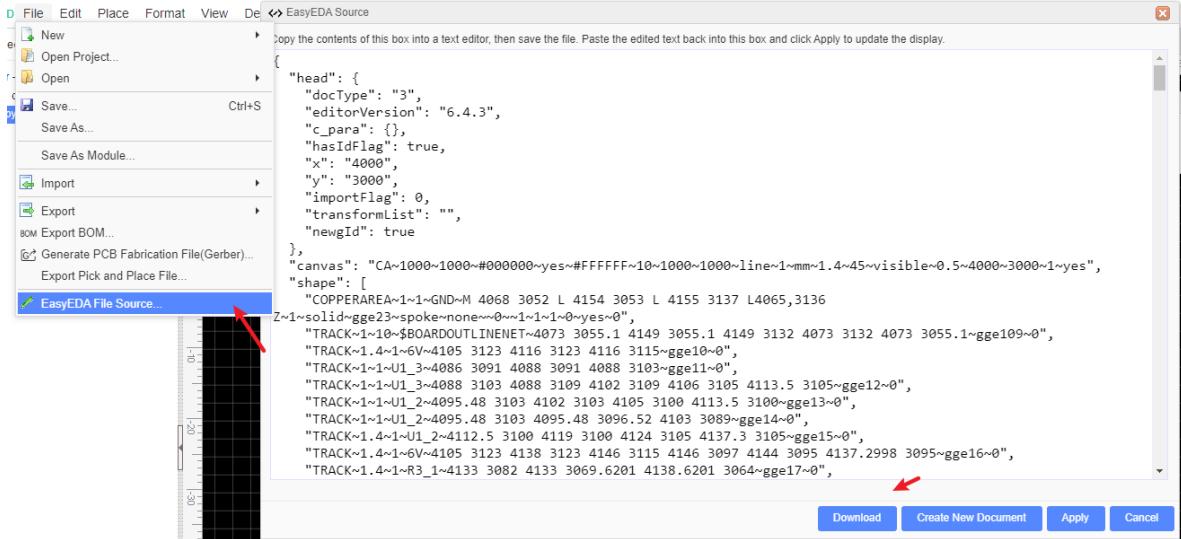
## Export EasyEDA Source

EasyEDA support you save your file to local, you can download your design as EasyEDA source file.

## 1. Export EasyEDA document directly

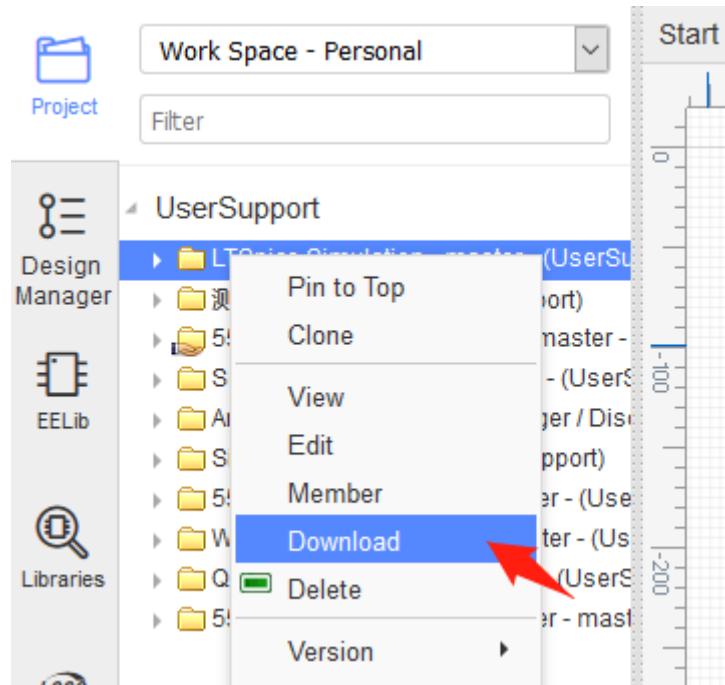
You can create an EasyEDA source file via:

\*\* > File > EasyEDA File Source...\*\*

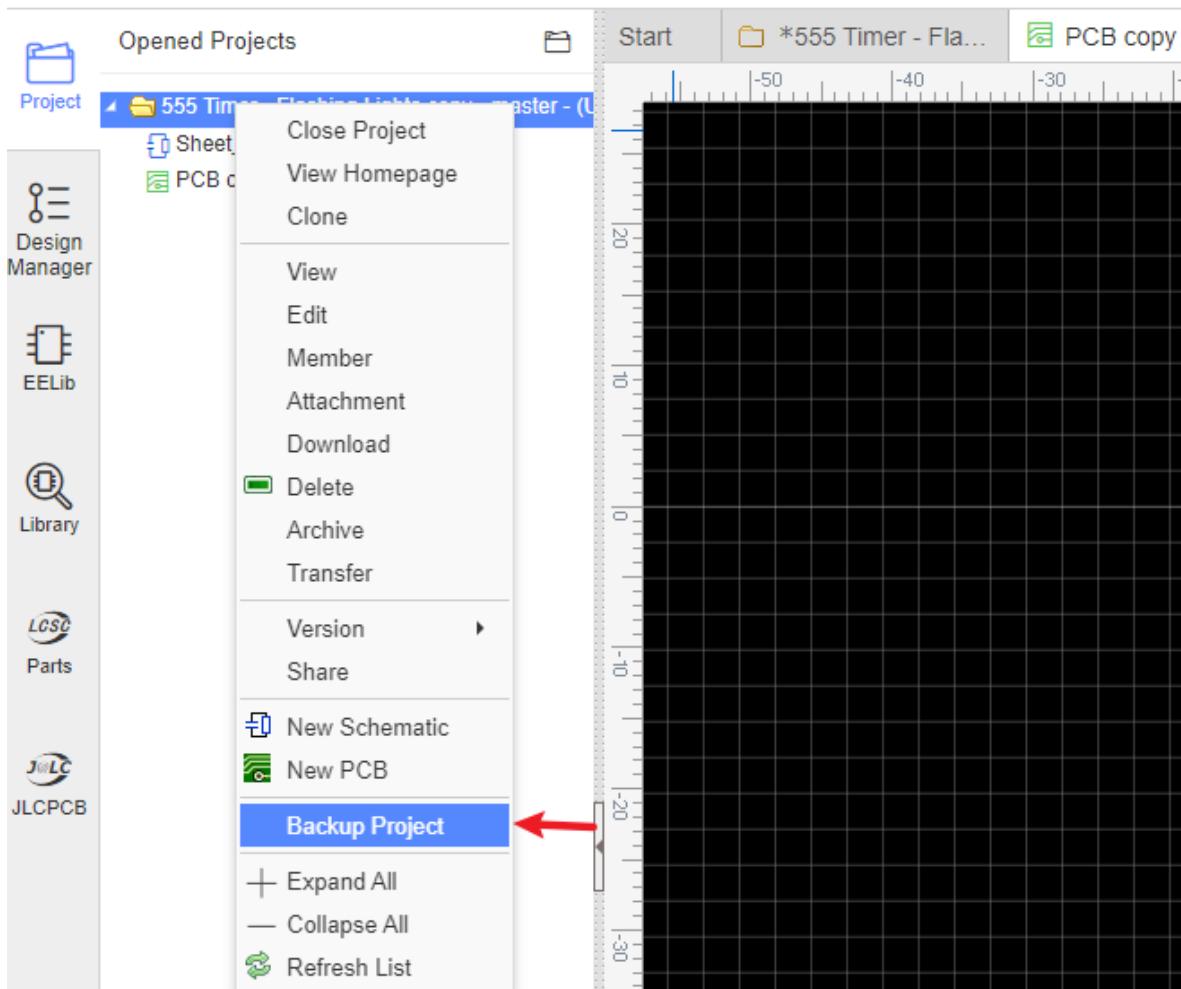


## 2. Download the project

Via: Project folder > Right Click > Download, you will download a zip file with EasyEDA Source files for Schematics and PCBs.



Or you can backup the projects, via: Project folder > Right Click > Backup Project



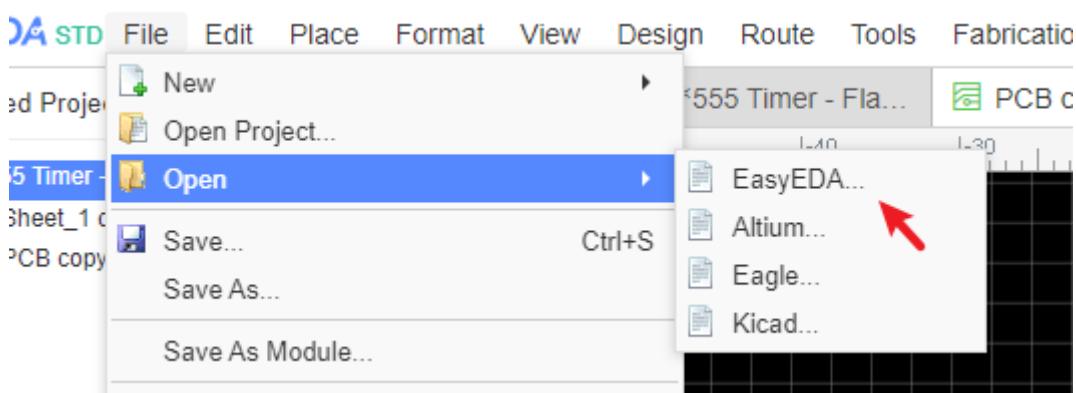
it will open a dialog, you can select the projects what you want to backup. There is only backup projects once per day.

EasyEDA Source File is a **JSON** file which can be read by many other programs. JSON format please see:

<http://en.wikipedia.org/wiki/JSON>

### 3. Open EasyEDA File

If you want to open the EasyEDA file you exported, you can try: " - File - Open - EasyEDA...".



Then you can edit and save the document.

