

# University of Global Village (UGV), Barishal



## Dept. of Electrical and Electronic Engineering (EEE)

### Lab Manual PCB Design

**Noor Md Shahriar**

BSc in EEE, [RUET](#)

[Senior Lecturer](#)

Co-chairman, Dept. of EEE [University of Global Village \(UGV\)](#)

[874/322, C&B Road, Barishal, Bangladesh.](#)  Contact: [+8801743500587](tel:+8801743500587)

 [Facebook](#) |  [LinkedIn](#) |  [Twitter](#)



## Contents

Experiment no: 01 .....	3
Experiment no:02 .....	7
Experiment no:03 .....	13
Experiment no:04 .....	22
Experiment no:05 .....	28
Experiment no:06 .....	34
Experiment no: 07 .....	43
Experiment no: 08 .....	57
Experiment no: 09 .....	63
Experiment no: 10 .....	71
Experiment no: 11 .....	82
Experiment no: 12 .....	88
Experiment no: 13 .....	105
Experiment no: 14 .....	119
Experiment no: 15 .....	129
Experiment no: 16 .....	195
Experiment no: 17 .....	208
Experiment no: 18 .....	221
Experiment no: 19 .....	231

# Experiment no: 01

## Experiment Title: Introduction to EasyEDA

---

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

There is 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, it is highly recommended to use Chrome or Firefox as your browser. You can also download [EasyEDA client](#).

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

### EasyEDA Editor:

<https://easyeda.com/editor>

### Instruction:

- ◆ This tutorial document will be updated as changes are made to the EasyEDA editor.

### Tutorial for PDF

[EasyEDA-Tutorials.pdf](#)

### EasyEDA Provides:

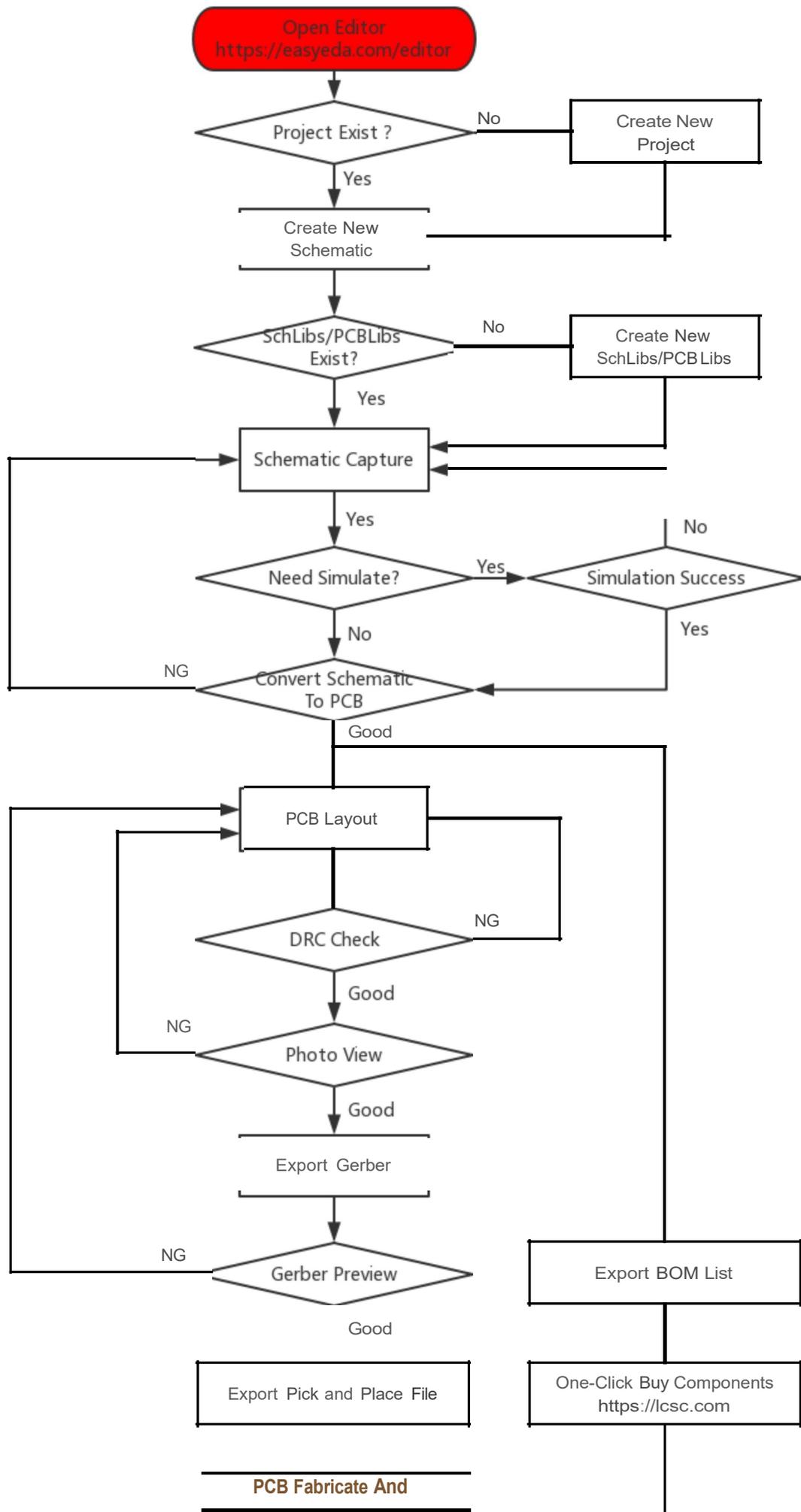
- ◆ Simple, Easy, Friendly, and Powerful drawing capabilities
- ◆ Works Anywhere, Anytime, on Any Device
- ◆ Real-time Team Cooperation
- ◆ Sharing Online
- ◆ Thousands of open-source projects
- ◆ Integrated [PCB fabrication](#) and [Components purchase](#) workflow
- ◆ API provided
- ◆ Script support
- ◆ 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
  - Multi-sheet schematics
  - Schematic module
  - Theme setting
  - Document recovery
- ◆ PCB Layout
  - Design Rules Checking(DRC)
  - Multi-Layer, 6 copper layer supported
  - Document export(PDF, PNG, SVG)
  - EasyEDA source file export(json)
  -

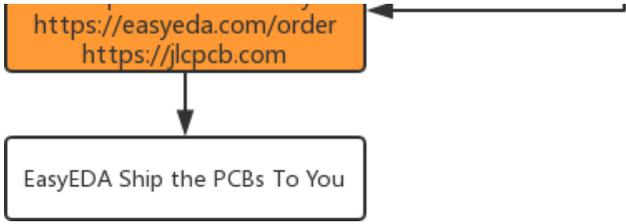
- Altium Designer format export
  - BOM export
  - DXF export
  - Photo view
  - 3D View
  - Generate fabrication file(Gerber)
  - Export Pick and Place file
  - Auto Router
  - PCB module
  - Document recovery
  - ◆ Import
    - Altium/ProtelDXP ASCII Schematic/PCB
    - Eagle Schematic/PCB/Libraries
    - KiCAD Schematic/PCB/Libraries
    - DXF
  - ◆ Libraries
    - More than 1,000,000 public Libraries(Symbol and Footprint)
    - Library 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 design circuits easily using EasyEDA. The design flow is as shown here:





# Experiment no:02

---

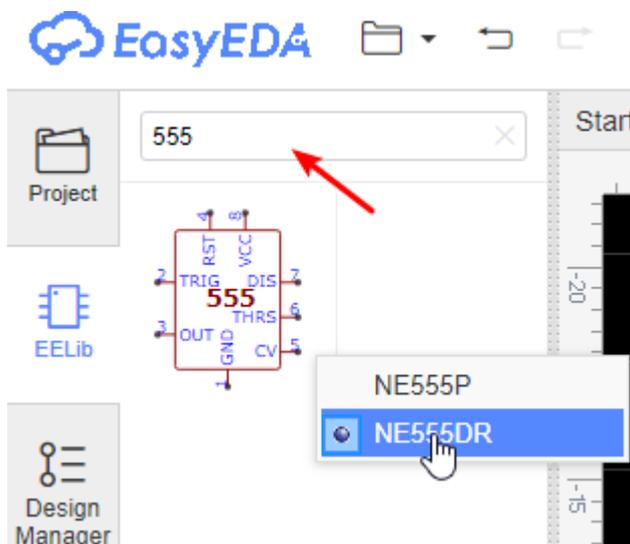
## Experiment Title: UI Introduction

---

EasyEDA Editor has a clear and friendly user interface. It has a short learning curve, and you will be productive in a short time.

### Filter

To use the filter first 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. For example, if you want to find all files containing "NE555" in the title, just type "555" . The filter is non-case sensitive.



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

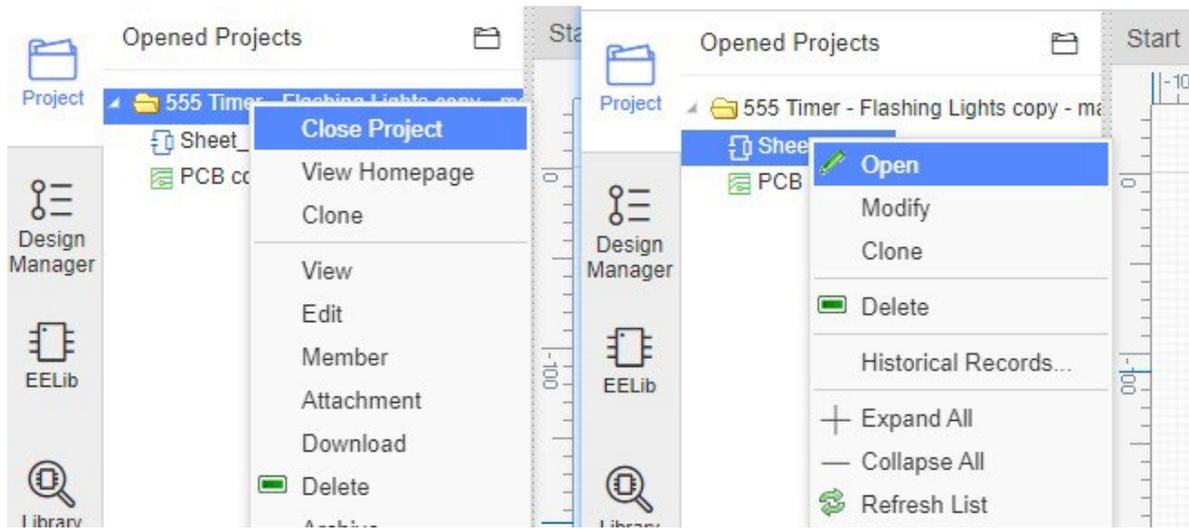
Click the X to clear the filter.

### Navigation Panel

The Navigation panel is especially important for EasyEDA: This is where you can find all of your projects, files, parts and footprints.

### Project

Here you can find all your projects, both private or shared with the public, or fork them from someone else's. These options have a context menu found by right clicking and selecting Projects. You will get a menu tree like:

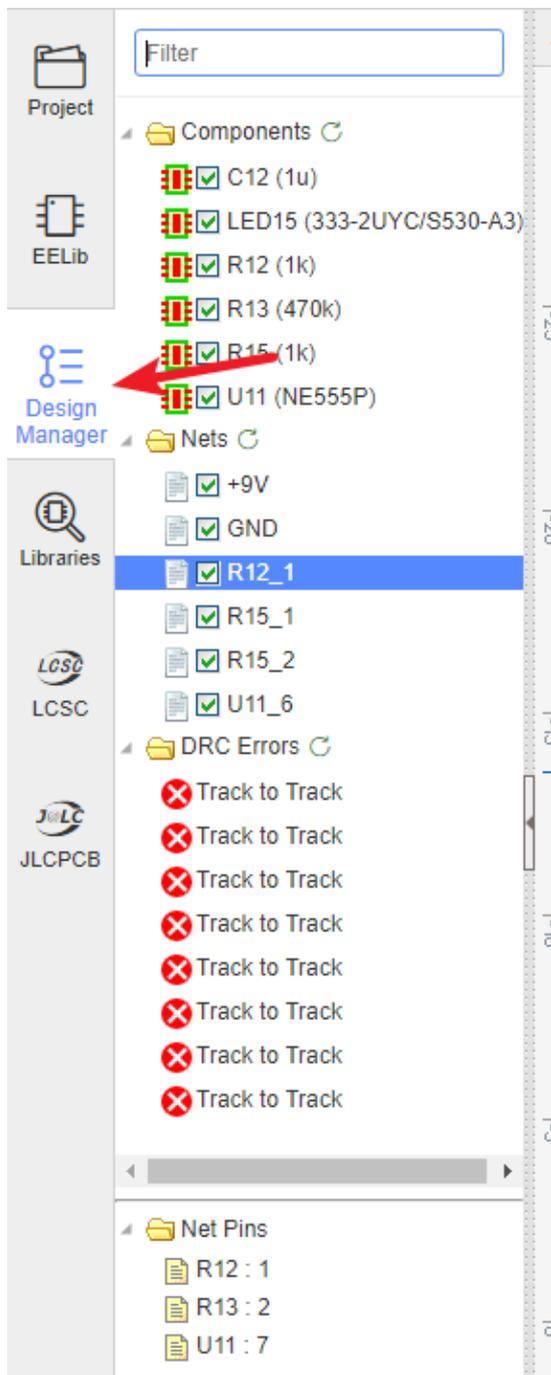


## EELib

EELib means EasyEDA Libraries, it provides lots of components complete with simulation models, many of which have been developed for EasyEDA to make your simulation experience easier.

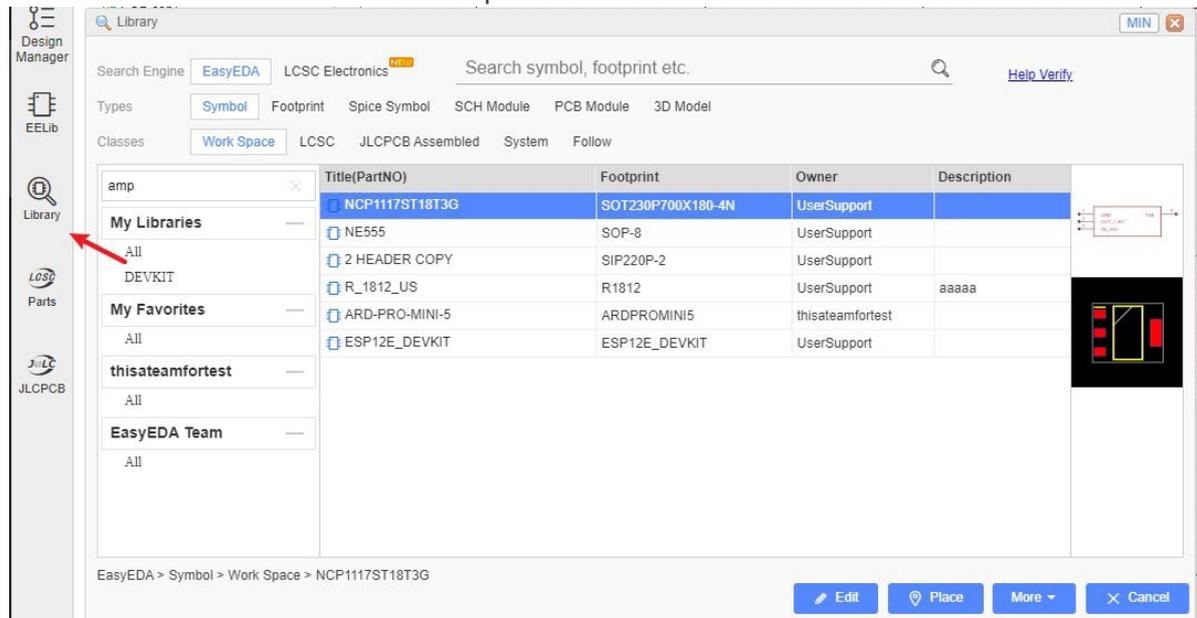
## Design Manager

Design Manager, you can check each component and net wiring, and it provides DRC(Design rule check) to help improve your design.



**Library**

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



#### ◆ LCSC

If you want to buy components to finish your PCB, 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 number of customers and product quantity shipped. <https://lsc.com>.

LCSC means **L**ove **C**omponents? **S**ave **C**ost! We suggest to our users to use LCSC parts to design. Why?

- Low minimum required quantities & Global Shipping.
- More Than 25,000 Kinds of Components.
- All components are genuine high quality.
- Ordering components is easy.
- Savings can exceed 40%.
- You can use LCSC component symbols and footprints directly in EasyEDA editor.

#### ◆ JLCPCB

JLCPCB.com, LCSC.com and EasyEDA are all part of the same company group.

<https://jlcpcb.com>

More than 200,000 customers worldwide trust JLC, 8000 + online orders per day, JLCPCB (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, high quality boards are fully manufactured in China. Boards are fully e-tested. Pricing is clear and easy to understand.

## Top Menu

Most EasyEDA features can be found on the top menu:



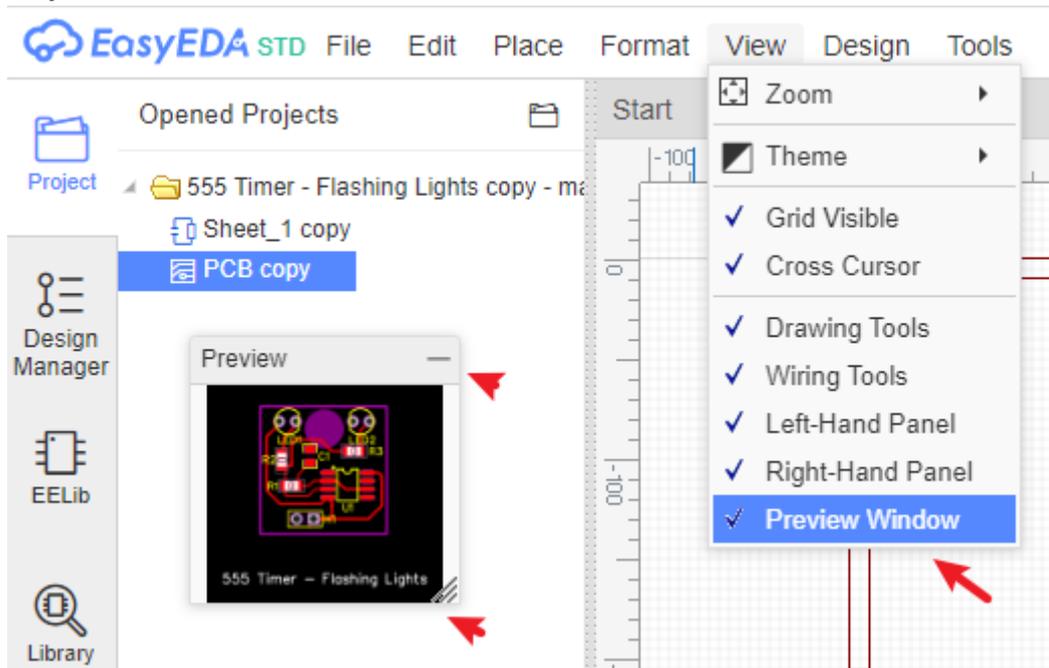
You can find what you need easily and clearly.

## 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 with:

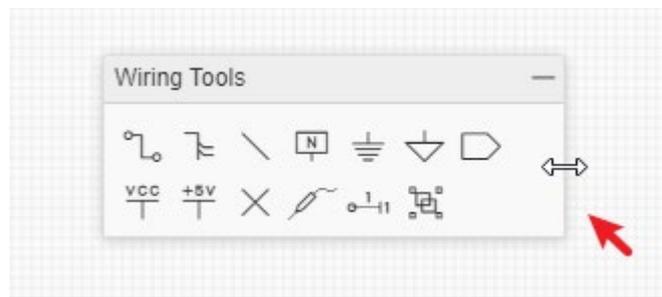
**Top Menu > View > Preview Window.**



- 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 the top banner again toggles it back to the selected size.

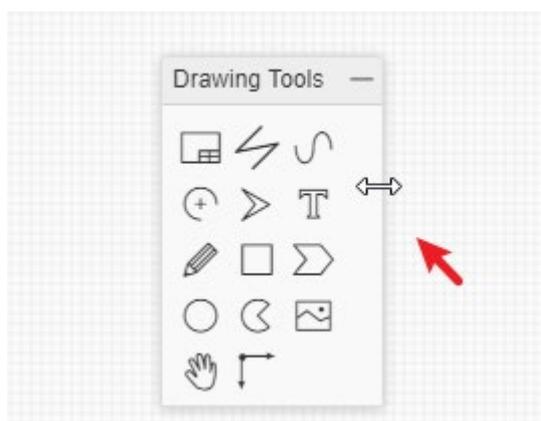
## Wiring Tools

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



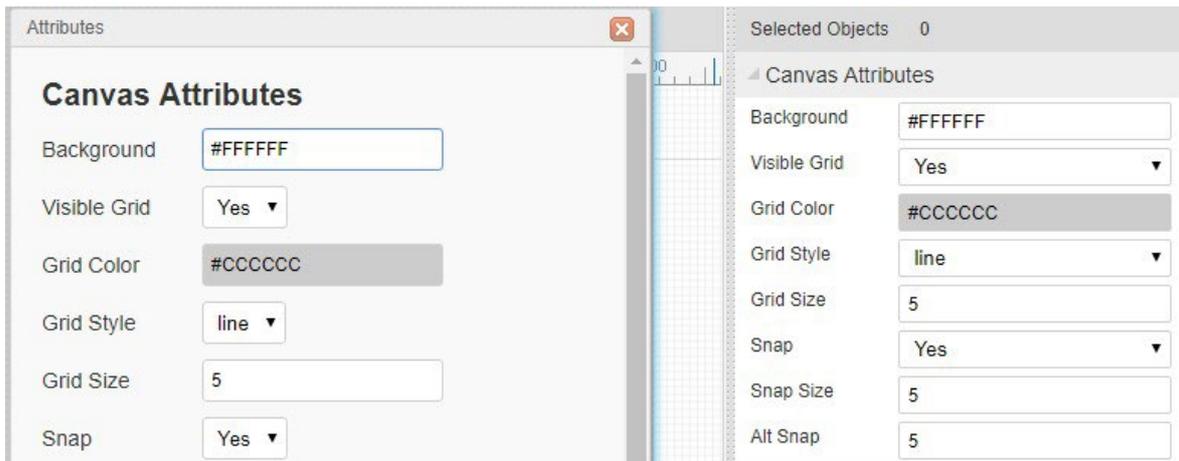
## Drawing Tools

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 or have a smaller monitor, you can roll up or hide them to free up more monitor space and reduce the clutter.



# Canvas Attributes

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

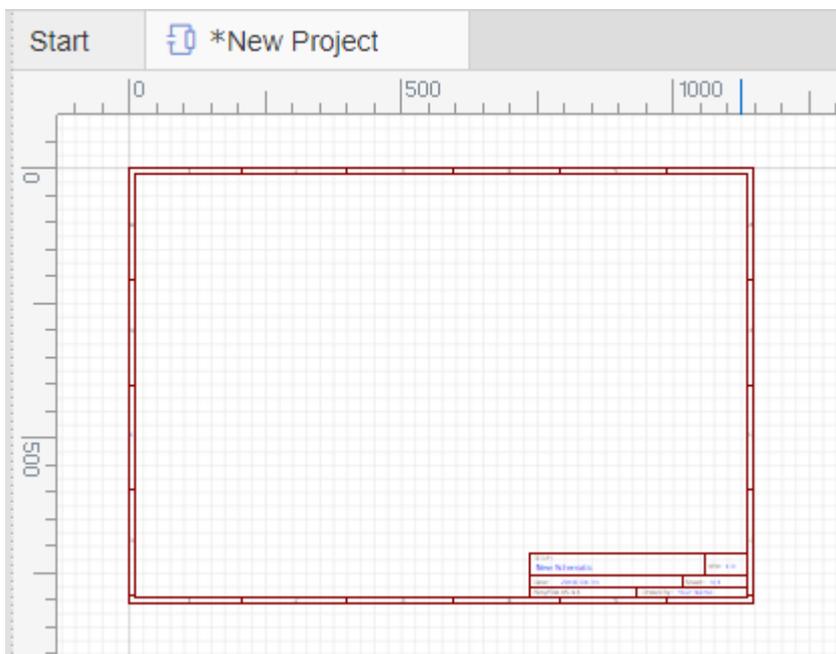


The background and grid colors and the style, size, visibility and snap attributes of the grid can all be configured.

The canvas area can be set directly by the Width and Height or from the 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. You also run simulations and display Waveform traces from here.



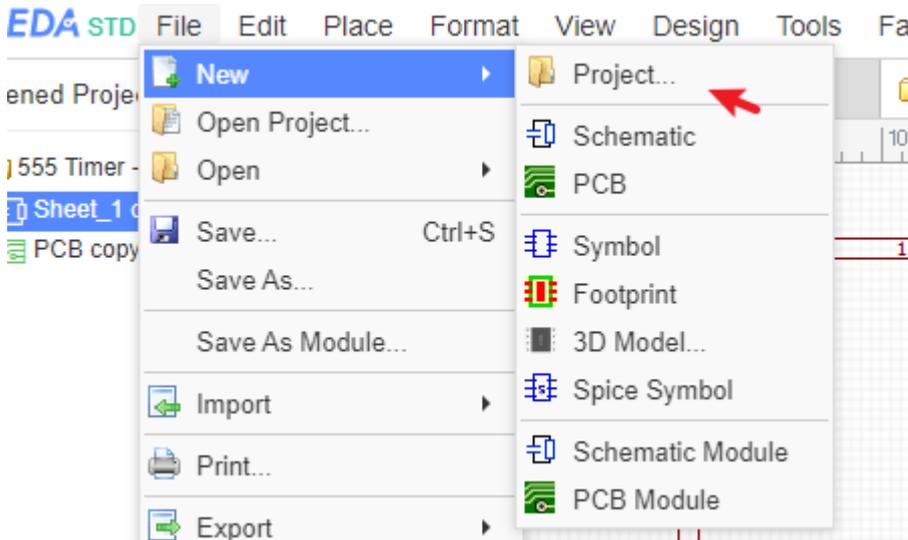
# Experiment no:03

## Experiment Title: Creating New Project or File

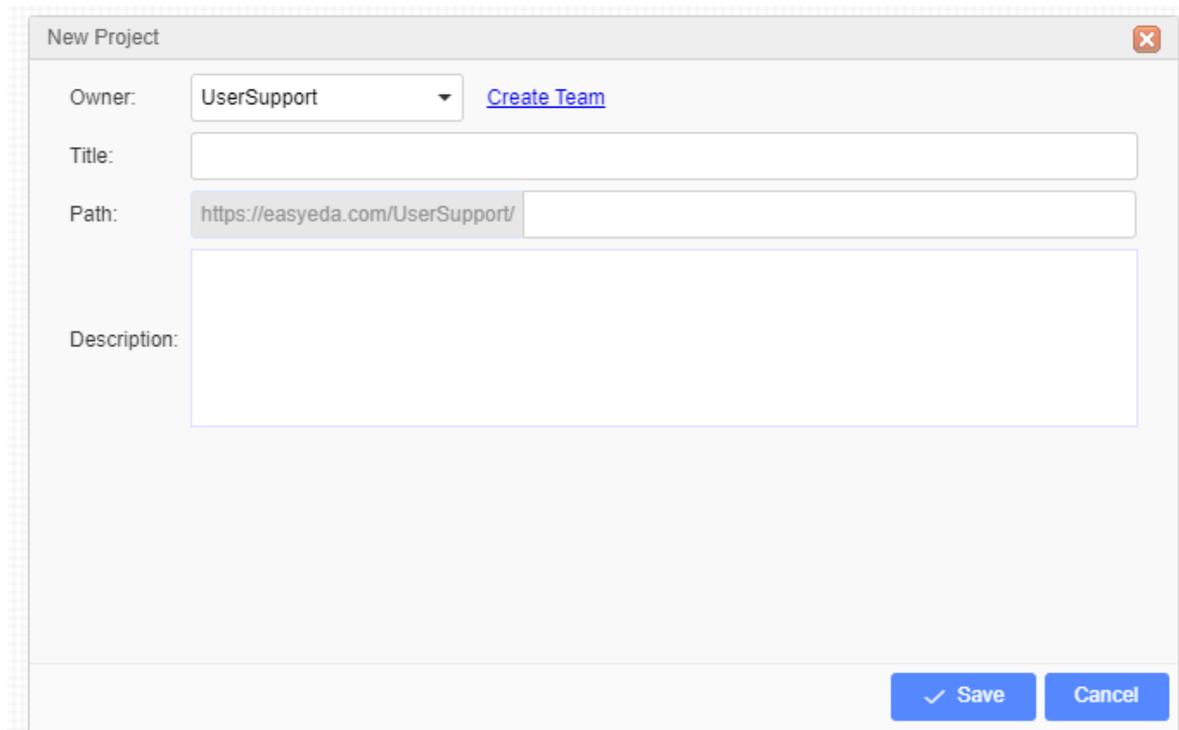
### Create Project

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 organizing your designs.

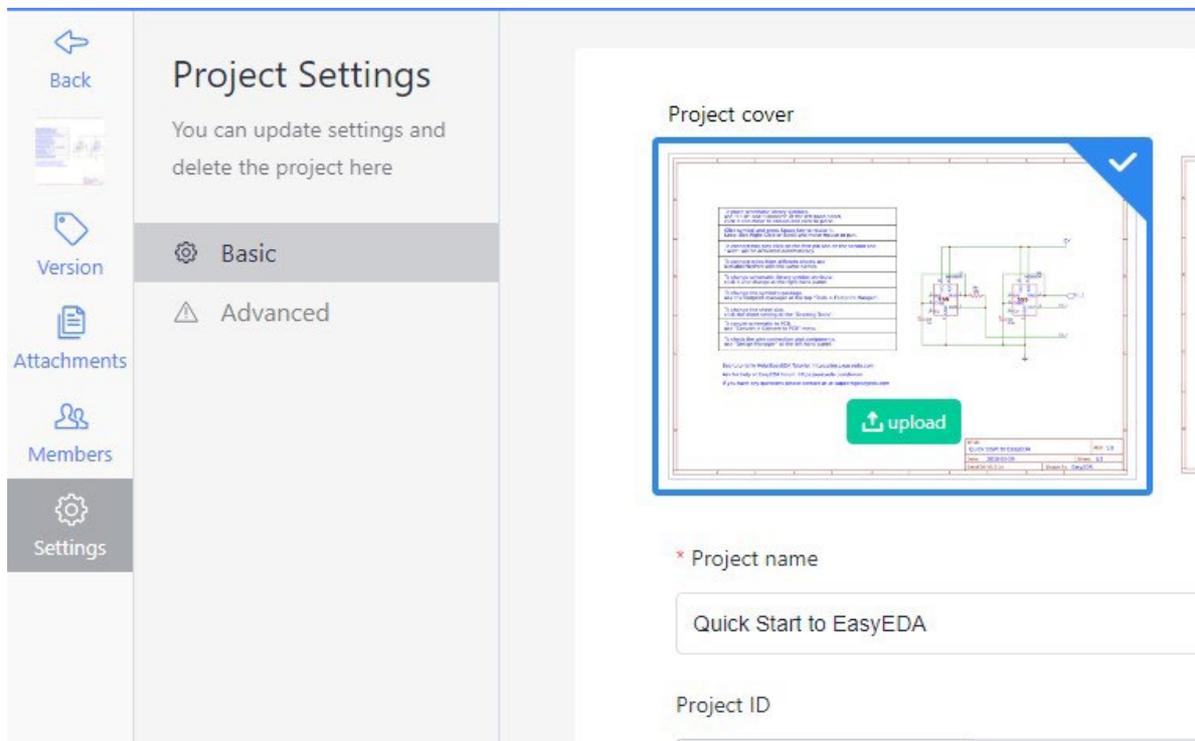
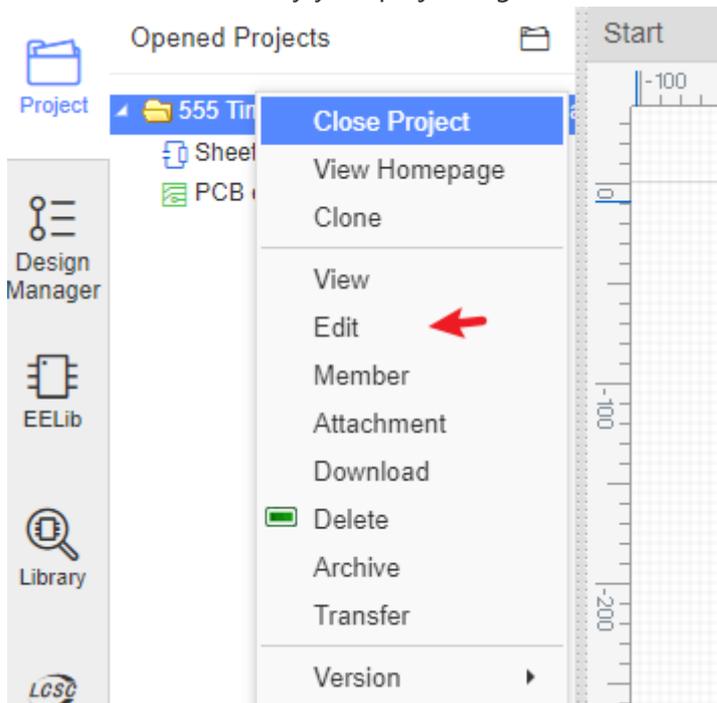
A screenshot of the 'New Project' dialog box in EasyEDA. The dialog has a title bar with a close button. It contains the following fields:

- Owner:** A dropdown menu showing 'UserSupport' and a link to 'Create Team'.
- Title:** An empty text input field.
- Path:** A text input field containing 'https://easyeda.com/UserSupport/'.
- Description:** A large empty text area.

At the bottom right, there are two buttons: 'Save' (with a checkmark icon) and 'Cancel'.

- ◆ **Owner:** You can change the owner of this project; you can change the owner to the team that you have joined.
- ◆ **Title:** Give it a title: this will show in the project tree in the left-hand panel.
- ◆ **Path:** EasyEDA allows you to set the path for the project, this is useful for sharing with your friends. It cannot be edited once it has been 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:

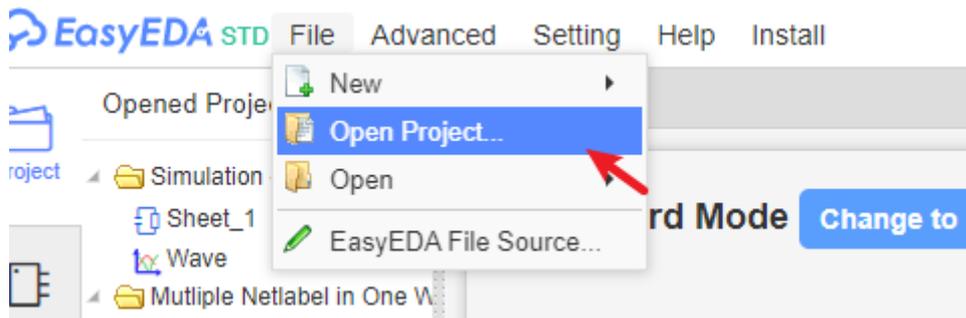


From here, you can change to publish or not, allow other people to comment on your project, or type a more detailed description of the project content. To help you make your project stand out or to maybe simply make a detailed description of your project easier to read, you can use Markdown syntax.

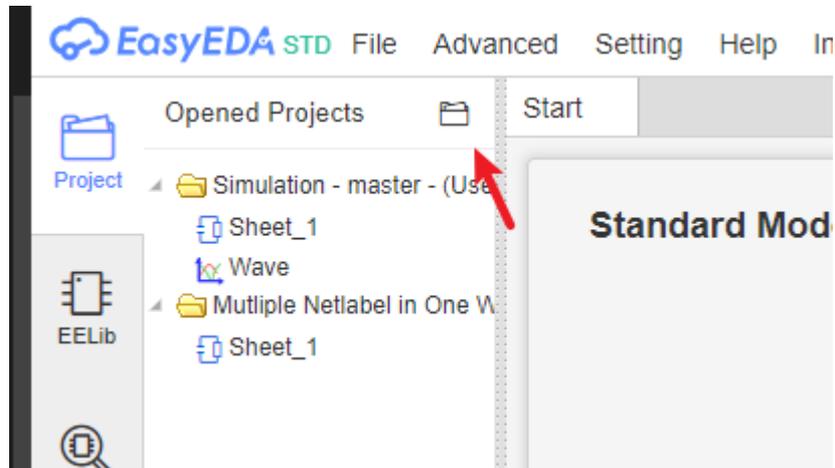
## Open Project

You can open your created project using:

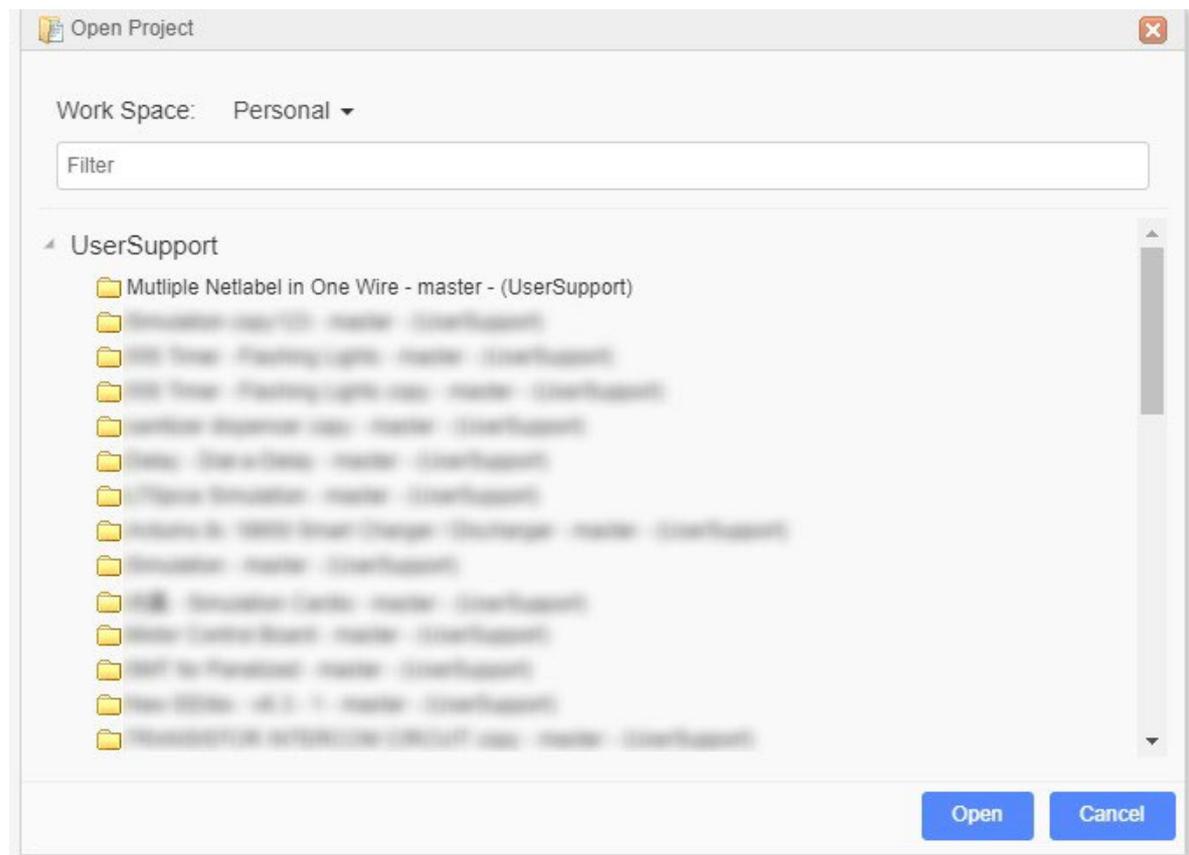
Top Menu - File - Open Project



Or click the Opened Project "open project" icon.

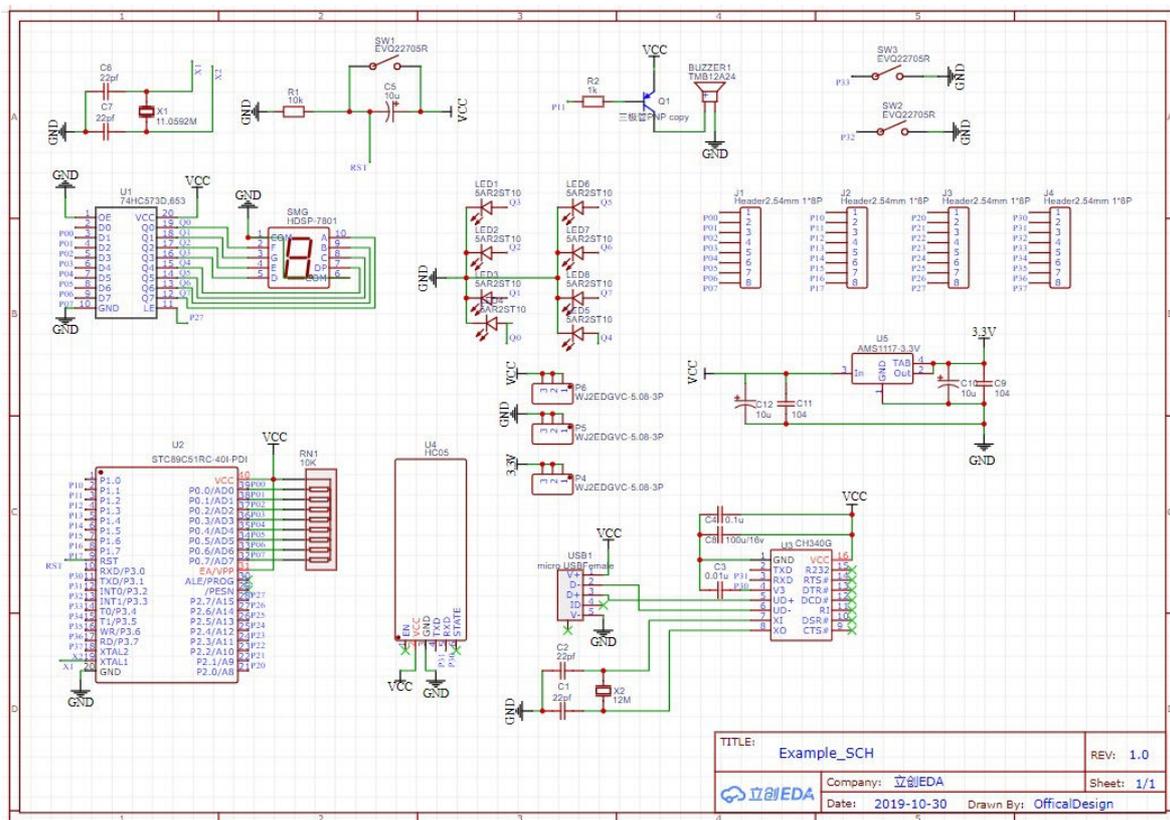


Select the project and open it.



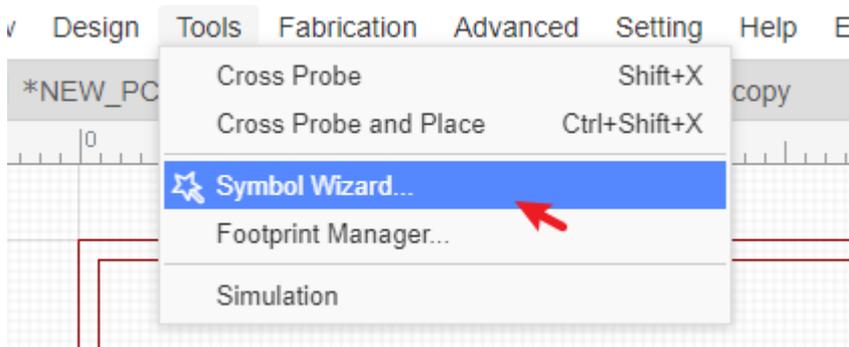
## Schematic Capture

EasyEDA can create professional quality schematics.



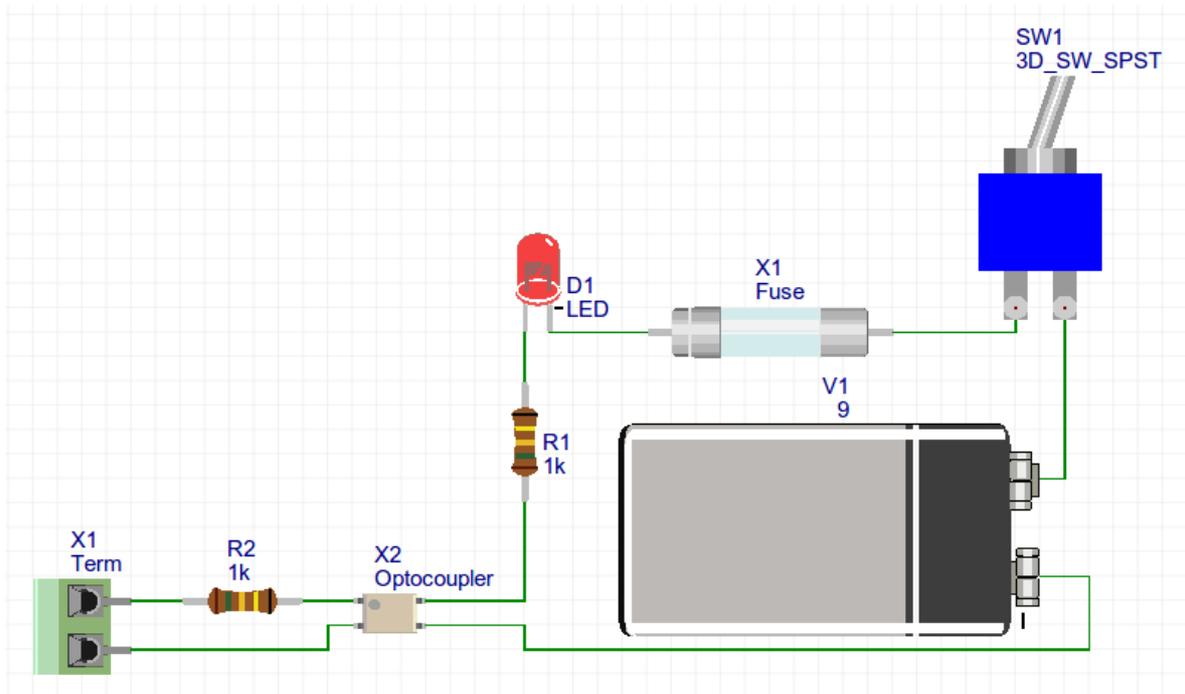
Because EasyEDA has some simple but powerful drawing capabilities, you can create your own symbols either by copying existing symbols into your own library and then editing and saving them, or by drawing them from scratch.

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.

Using the drawing features in symbol creation, your schematic can be built like this:



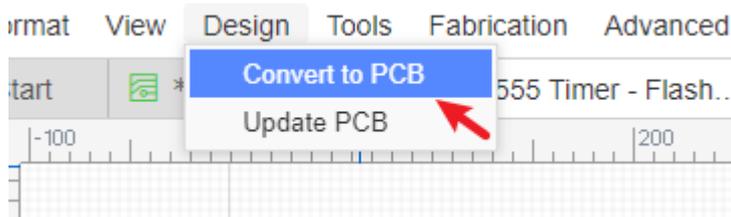
Another powerful feature is that it is possible to import symbols from `KiCad`, `Eagle` or `Altium` libraries.

## 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.

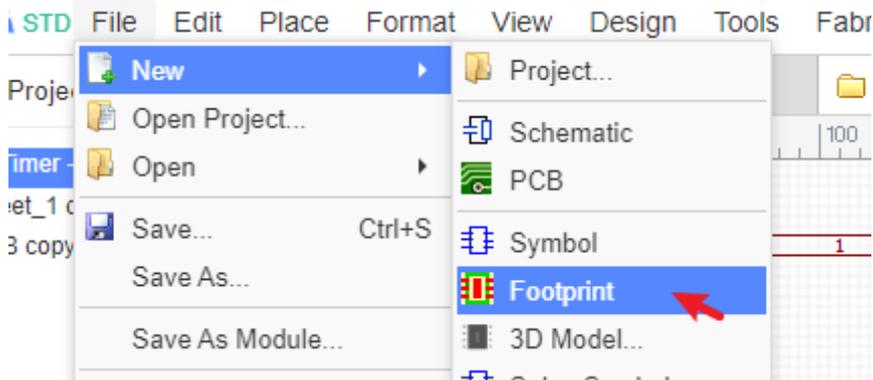
EasyEDA's PCB Design canvas helps you to quickly and easily lay out even complex multilayer designs from schematics you have already created in the Schematic canvas or directly as a layout with no schematic.

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



- ◆ EasyEDA has extensive component 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.

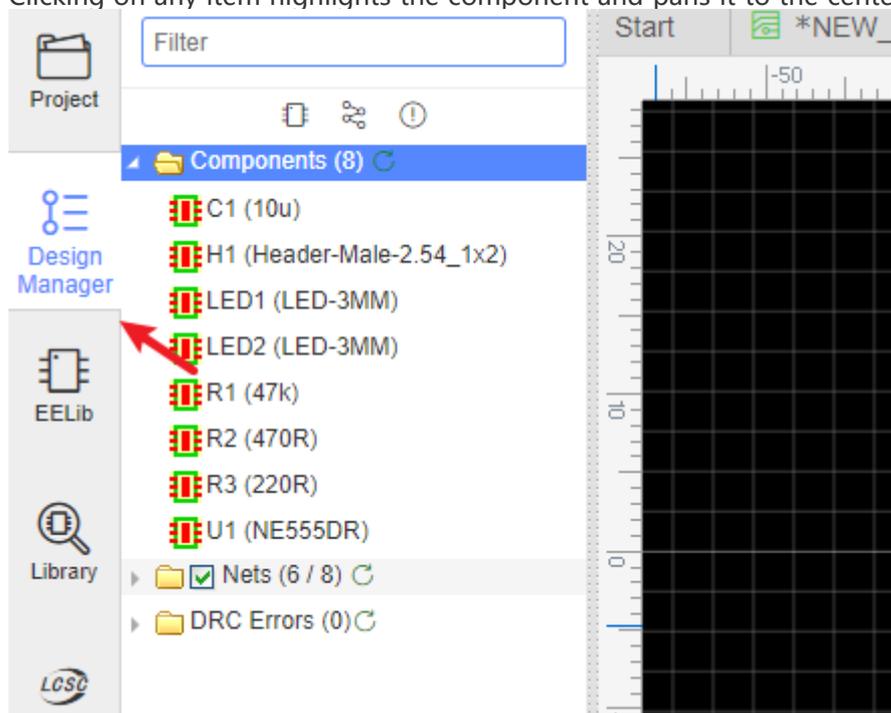


- When working in the PCB Design canvas there is a PCB Design Manager which works in a similar way to the Schematic design canvas, this will help you locate items and navigate your way around.

#### Left Navigation Panel > Design Manager

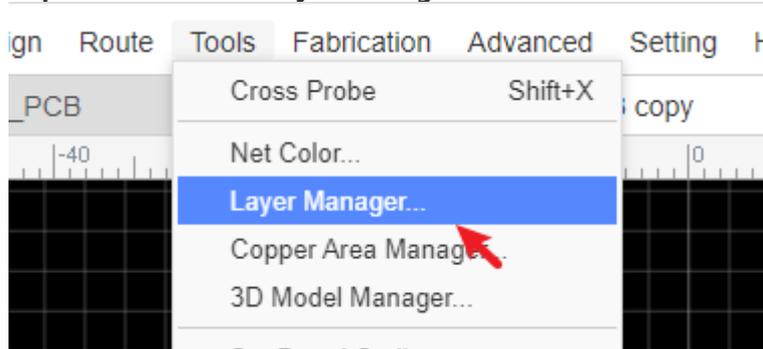
The PCB Design Manager is a powerful tool for finding components, tracks (nets) and pads (Net Pads).

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

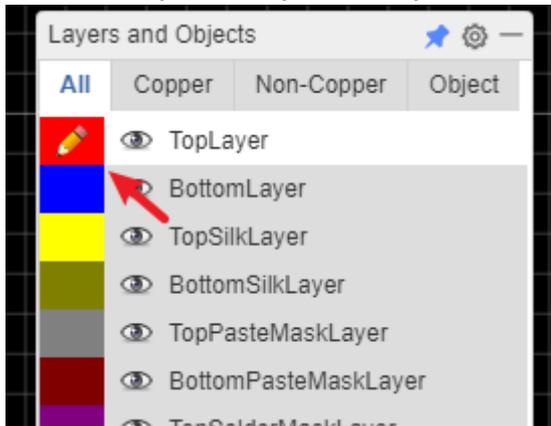


- You can also apply the design rules and display errors and visibility using

#### Top Menu - Tools - Layer Manager...

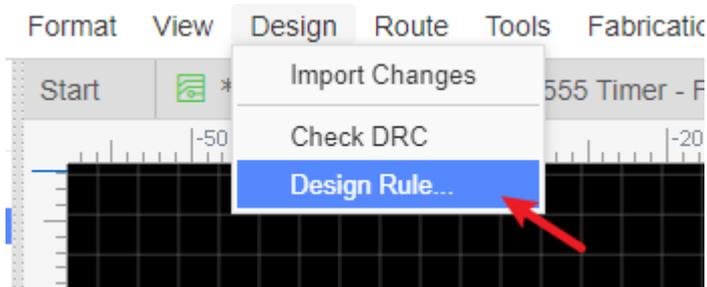


The active layer and layer visibility can be selected using the Layers Toolbar.



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

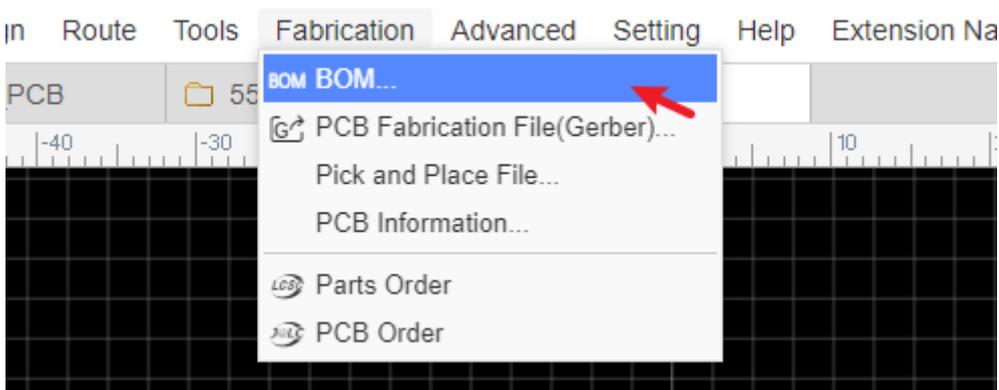
**Top Menu > Design > Design Rule...**



The Design Rule Check (**DRC**) is created when beginning your board layout. It can also be modified at any time. Running a DRC is one of the last steps in checking your PCB design before you generate **Gerber** and **Drill** files for board manufacture and are ready to place your order for a finished PCB.

- ◆ The final step is to check the Gerber and Drill files using a software Gerber viewer. This is an easy to install and use 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 or at any time it is needed, you can create a Bill of Materials (BOM) with:

**File > Export BOM...** or **Top Menu - Fabrication - BOM...**



- ◆ 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 editor.

You can place the selected footprints in the canvas after a successful search.

---

## Libraries Management

---

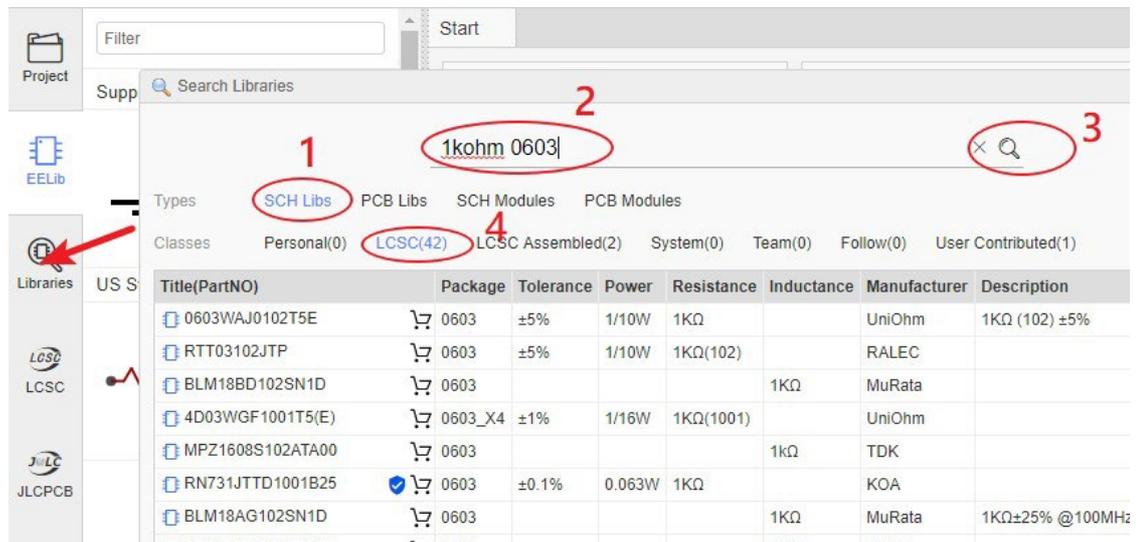
Thanks to the Free and Open Source Kicad Libs and some Open Source Eagle libs, EasyEDA now has 700,000+ components, which should be enough for most projects!

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

- ◆ **Library**

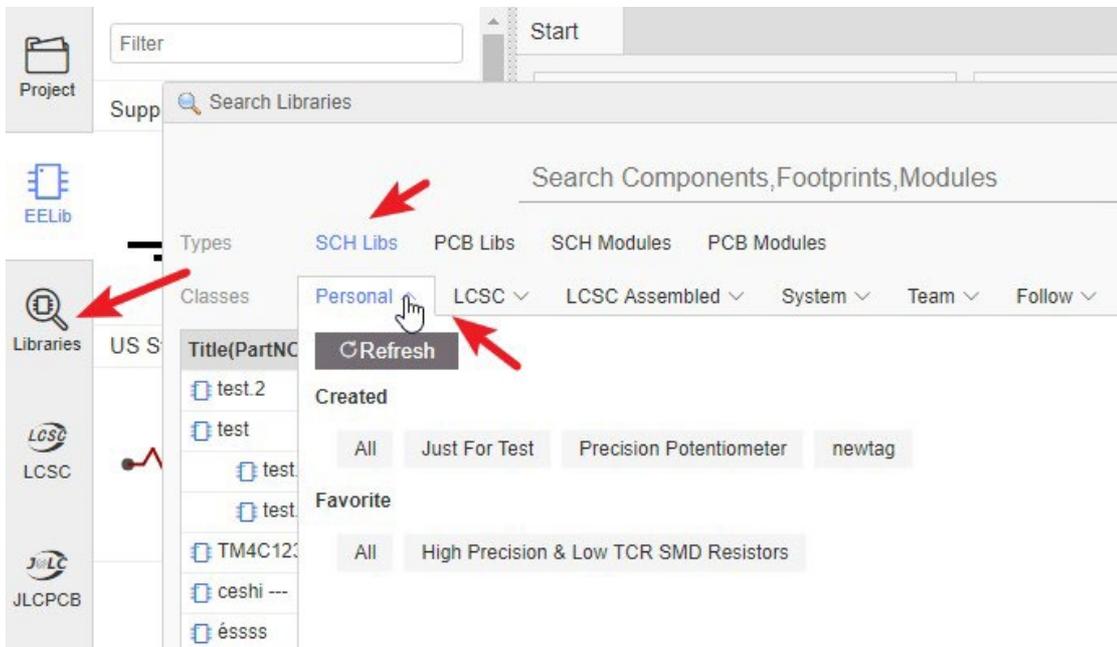
On the left-side Navigation panel you will find "**EELib**" and "**Library**", just type what components you want and search.

At Libraries:



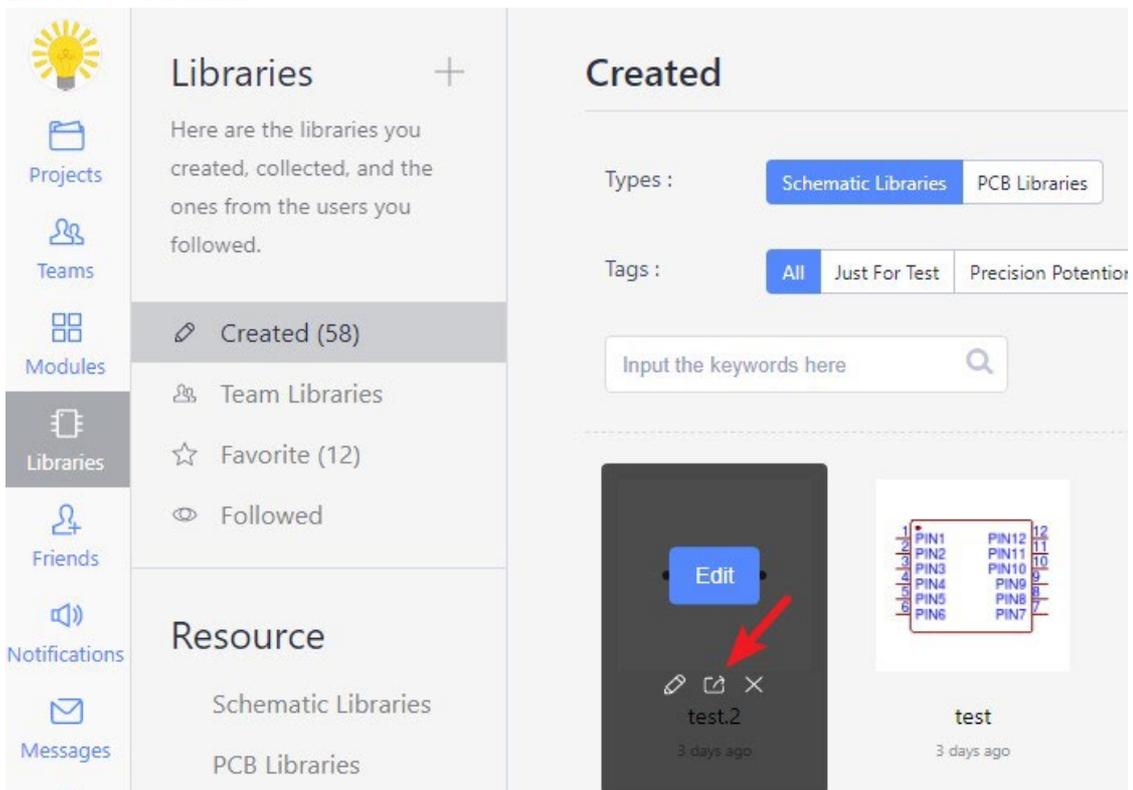
Steps:

- ◇ 1. Choose the library type
  - ◇ 2. Type keywords such as "1k 0603"
  - ◇ 3. Click the search button
  - ◇ 4. Make your choice from the search results
  - ◇ 5. When you are done searching remove all the keywords
- ◆ **Create Library**  
EasyEDA supports creating your own symbols. After creation you can find your own components at **Library > Symbols/Footprints > Workspace**, and it is easy to manage your libraries.



#### ◆ Transfer Libraries

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



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

You can also produce professional quality **SVG**, **.PNG** or **.PDF** output files for your documentation.

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. You will be credited as a contributor. <https://easyeda.com/page/contribute>

# Experiment no:04

## Experiment Title: Version-Control & Member Management

---

EasyEDA provides a simple but powerful version control feature. Each version is independent, you can edit and save each version.

When creating a new project, the default name will be set to "master", you can edit the name using the "Project Manage - Version" page.

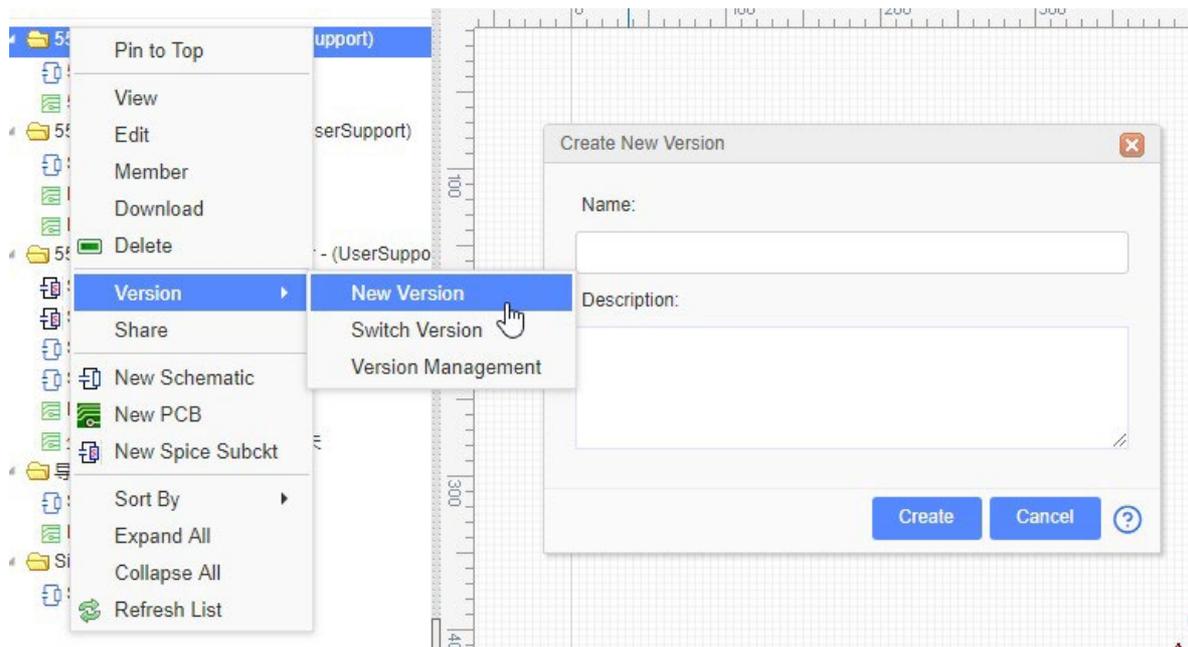
You can create up to 10 versions for every project. To create a new version, you must first delete an older version.

### Create New Version

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

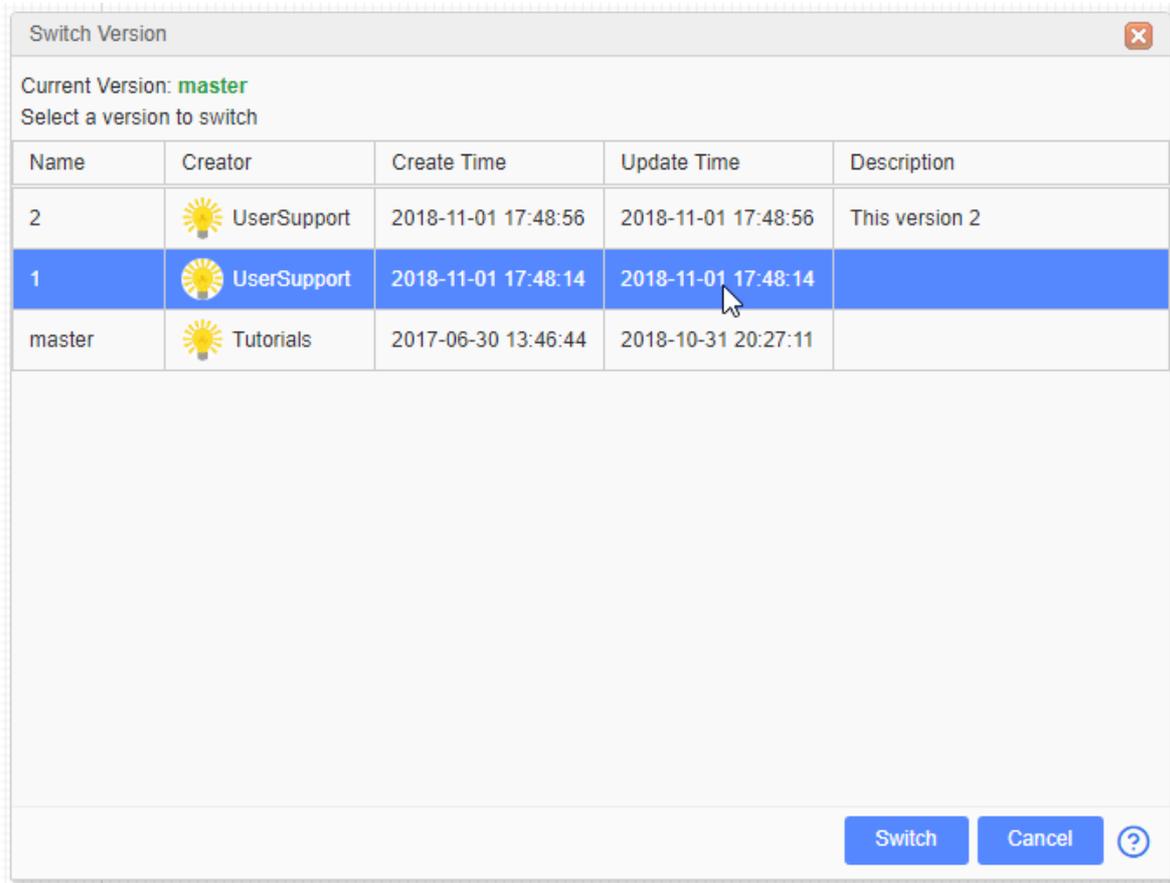
In the new version dialog, you type the version's name and description, and then create it.

To switch to another version use "Version - Switch Version".



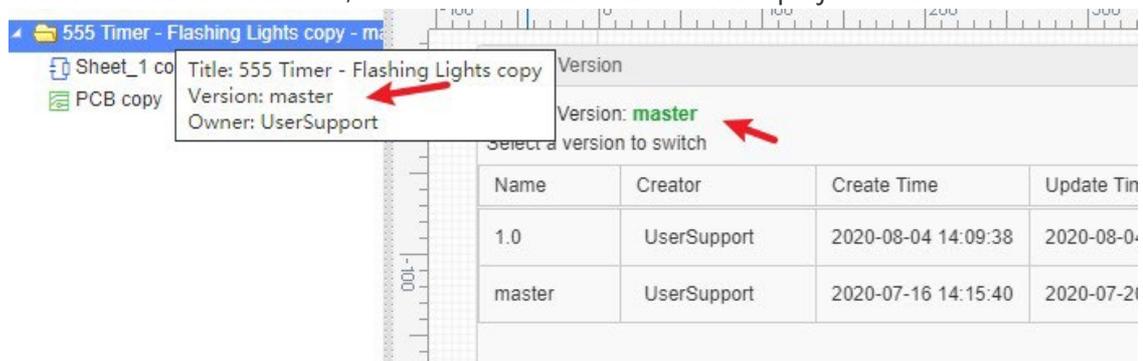
### Switch Version

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



Note:

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



## Version Management

Using "Version Management" will open the "Project Page - Version". which will list all versions. You can edit each versions name and description or delete them. The current version cannot be deleted.

Name	Creator	Create Time	Update Time	Description	Operation
master	Tutorials	2017-06-30 05:46:44	2018-10-31 12:27:11		
1	UserSupport	2018-11-01 09:48:14	2018-11-01 09:48:14		 
2	UserSupport	2018-11-01 09:48:56	2018-11-01 09:48:56	This version 2	

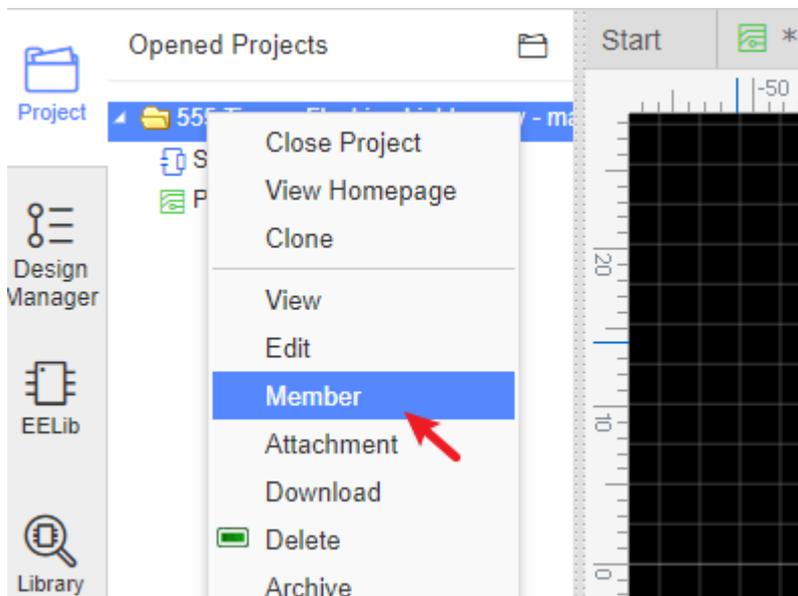
## Project Member

How to share project with selected people.

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

If the answer is yes, you can use **Member** to do this.

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



To share a project with someone,

1. You need to know the E-mail address they used to create an account with EasyEDA.
2. The project member is set as "Developer", "Manager", or "Observer".

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

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

# Share with Public

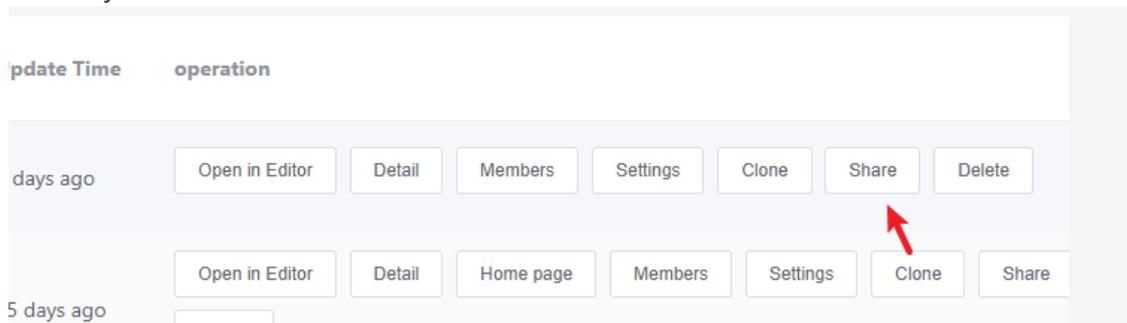
Sharing your work with others is a big feature of web based EDA tools and EasyEDA is no exception in offering great features.

Did you create a 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.

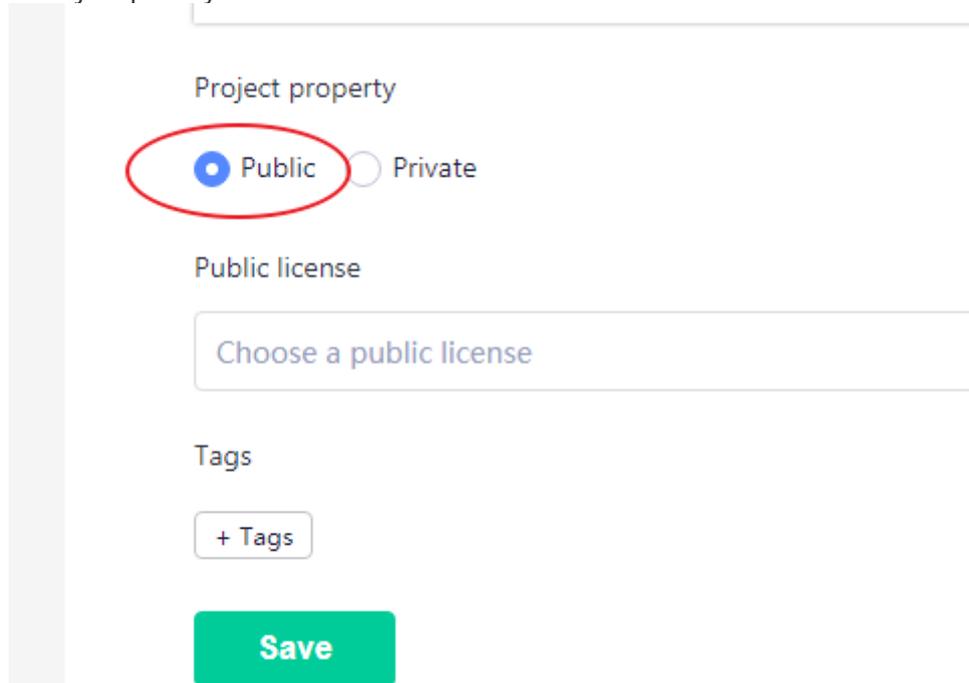
All projects in EasyEDA are set to private by default, your private project is not shared with anyone.

To make it public, you should right click and edit your existing project to make it a Public project:

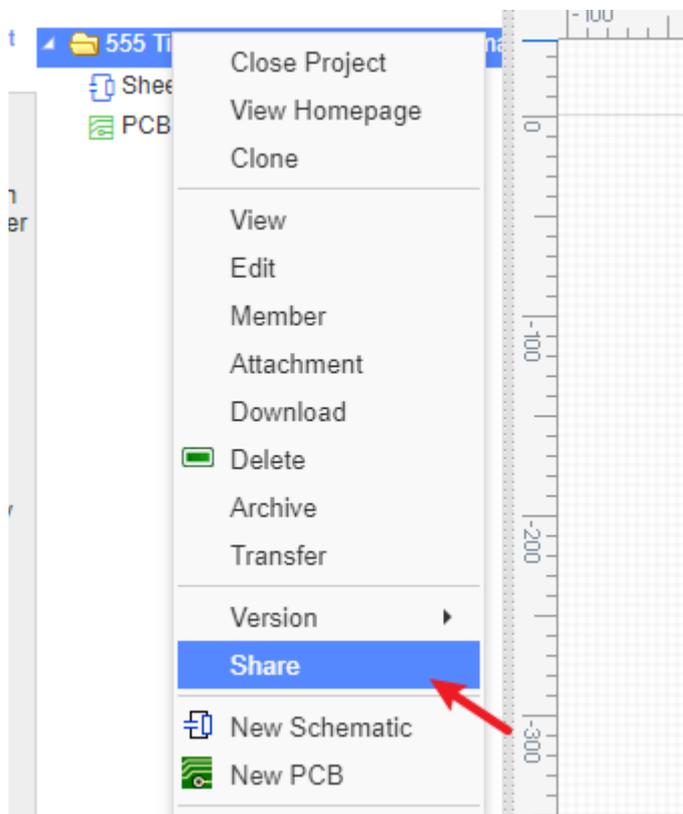
- ◆ In the Workspace, click the **Share** icon when the mouse hovers over the project cover, it will ask you to confirm.



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



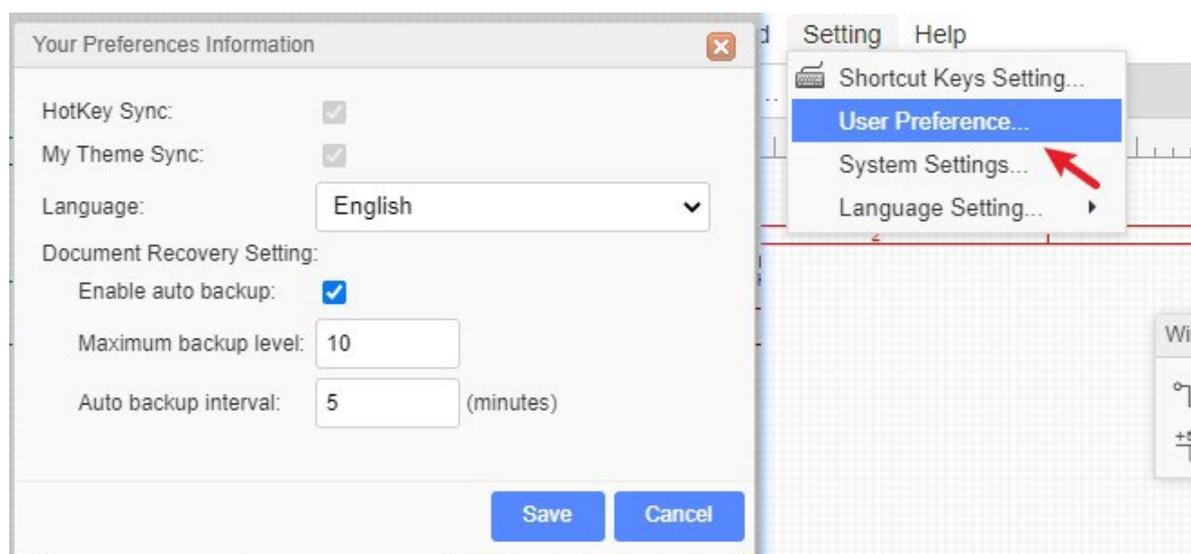
- ◆ ng the  
project as public.



## User Preference

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

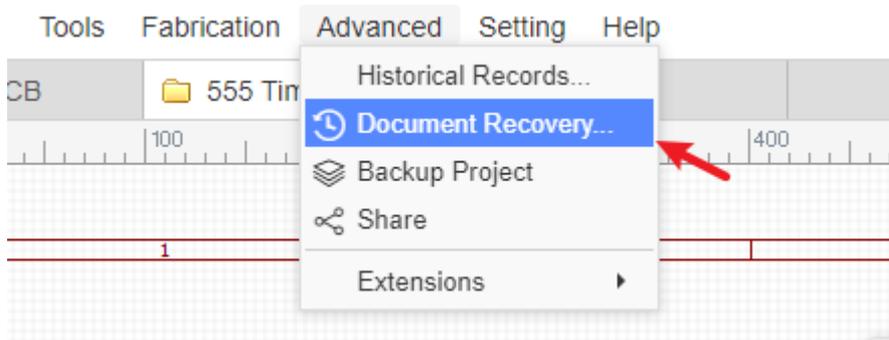
Click on **Top Menu - Setting - User Preferences**,



### Document Recovery Setting:

- ◆ **Maximum backup level:** Every opened document can be saved as a backup, up 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:



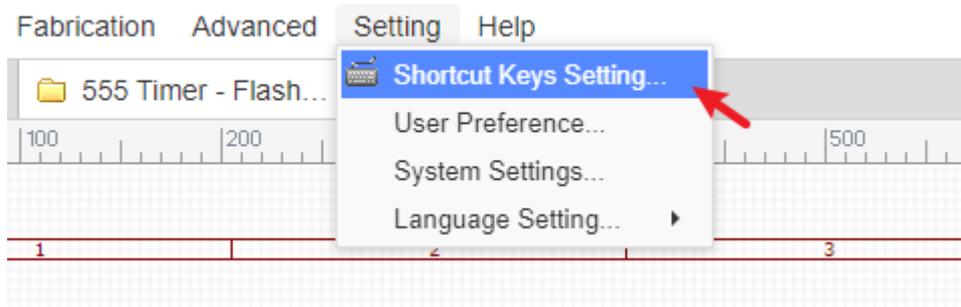
# Experiment no:05

## Experiment Title: Shortcut Keys of EasyEDA

---

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

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 changing any hotkey, you must click the 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 selecting an 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	ALT+W	Close current tab
All	SHIT+ALT+W	Close all tabs
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

DocType	Shortcut	Function
All	CTRL+S	Save
All	CTRL+F	Find Component
All	CTRL+D	Design Manager
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
Schematic	SHIFT+T	Open symbol wizard
Schematic	ALT+F	Open footprint manager

## **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

<b>DocType</b>	<b>Shortcut</b>	<b>Function</b>
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
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
PCB	CTRL+ALT+L	Enable all layers
Footprint	CTRL+SHIFT+ALT+D	Open custom data dialog

---

# Experiment no:06

## Experiment Title: Basic Skills of EasyEDA

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, and create a new schematic:

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

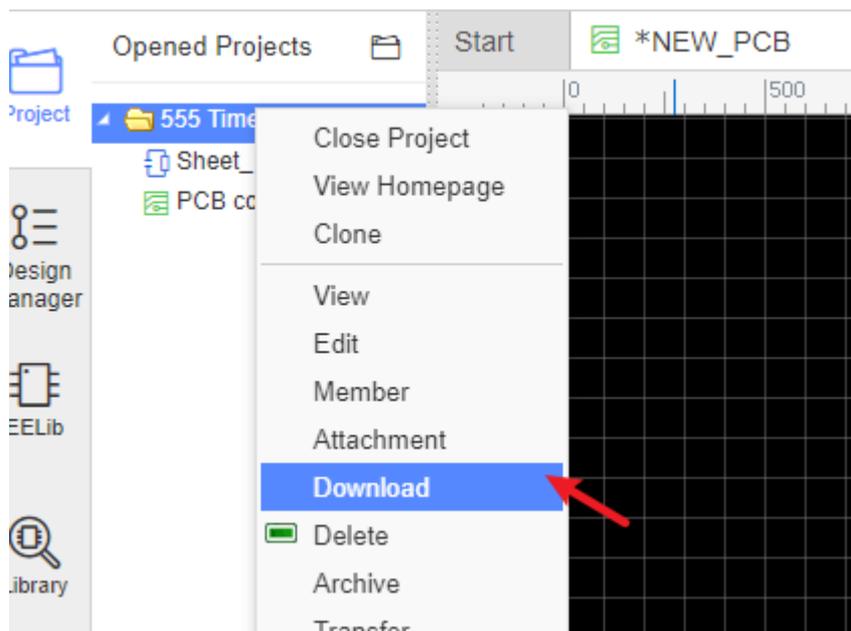
### Saving Your Work Locally

Although EasyEDA saves all your files on our Server, sometimes you may want to save your work locally and EasyEDA does provides a 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".

For more detail you look here: [Export EasyEDA Source](#) section.

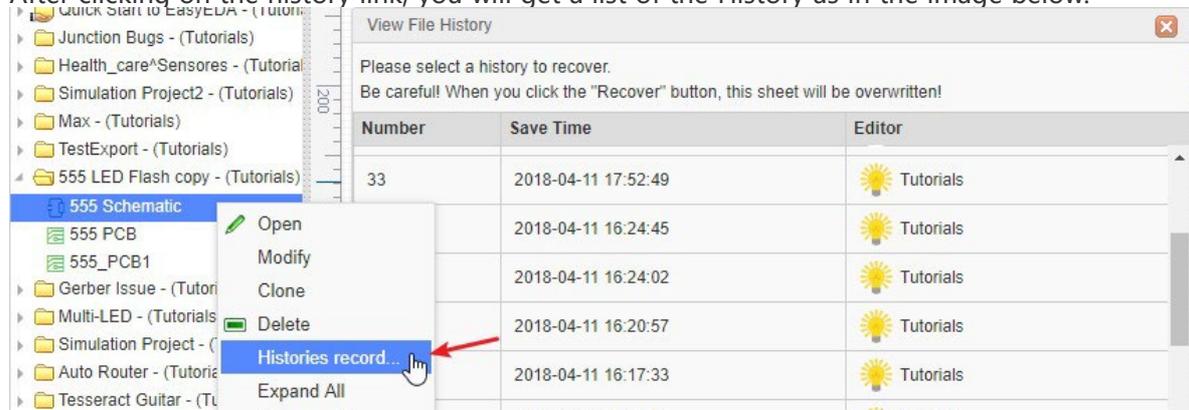
To download your project:



### History Record

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

After clicking on the history link, you will get a list of the History as in the image below.



Click the History number to 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 the History entries are tagged with a number. An upcoming feature will allow you to add a custom tag.
2. Saving your files too frequently will create many History entries and it will become more difficult to find the exact one you want.

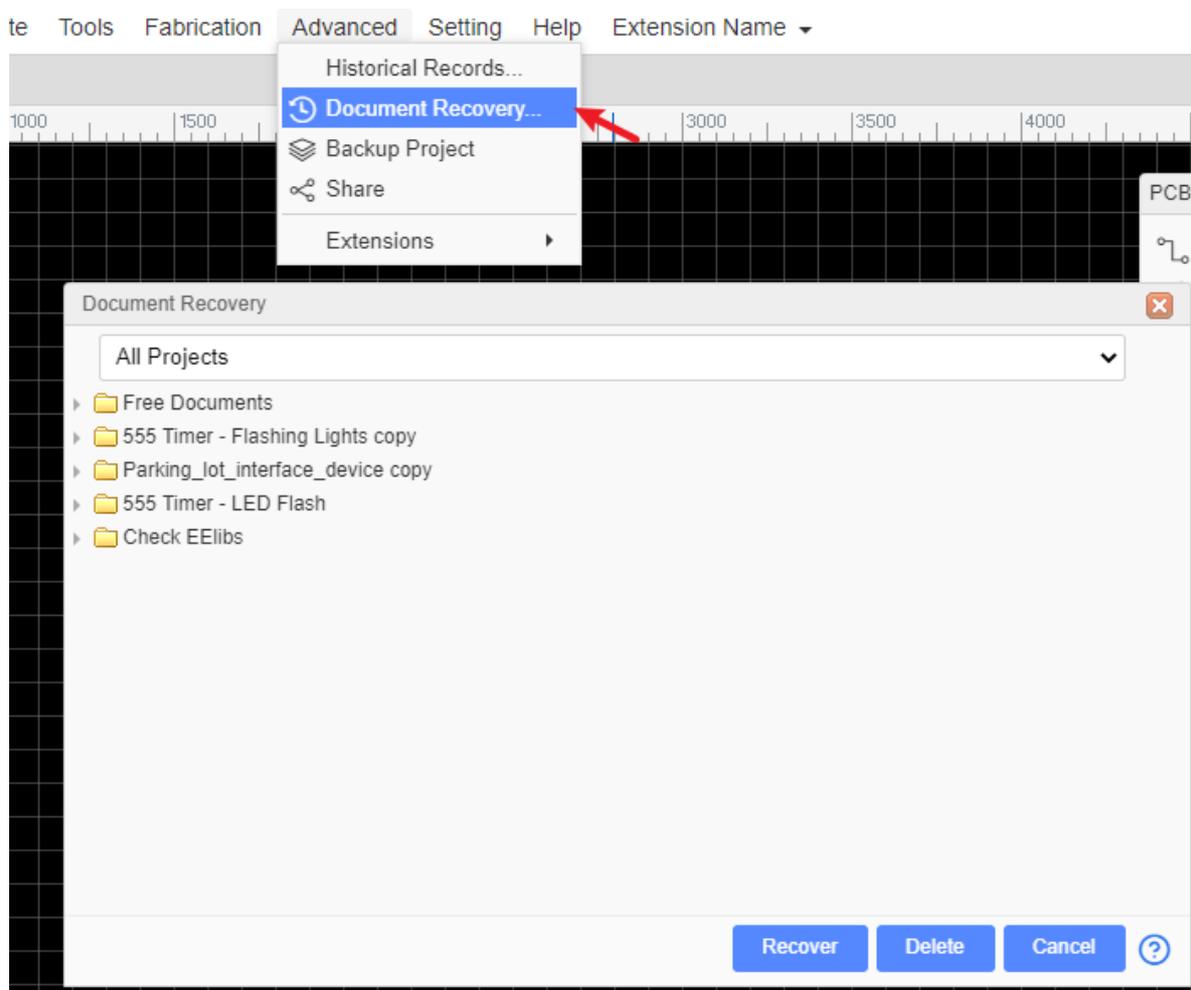
## Document Recovery

No operating system, software or network is perfect, so sometimes things 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, crash recovery is built into EasyEDA.

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



Expand the folder to the latest version, 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 this opened file.

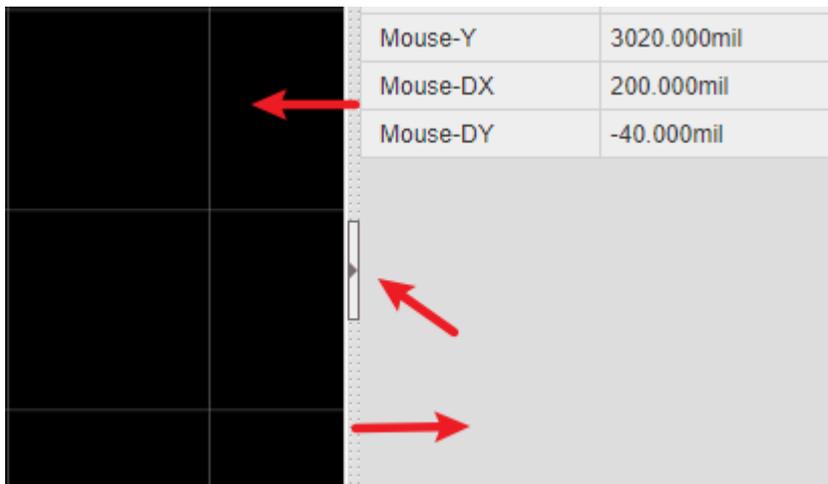
**Please note:**

- 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.

- If you cleaned your browser's cache, the recovery files will disappear.
- If you made a mistake deleting a file and removing the cache, you might be able to find your document in the recycle bin: <https://easyeda.com/account/user/recycles/personal>.

## Resizing the canvas area

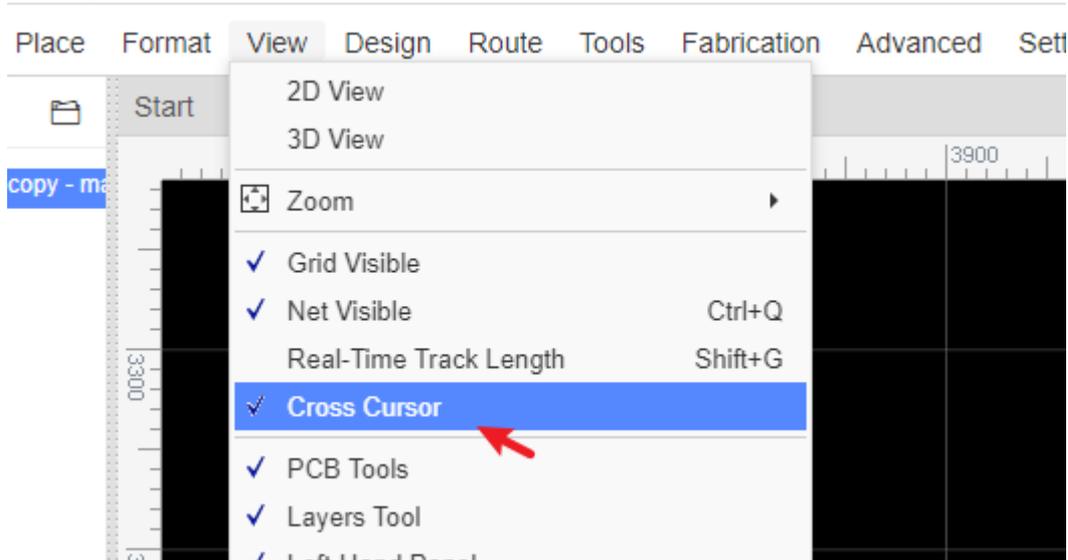
Hovering the mouse cursor over the areas indicated by the three green ellipses will bring up the blue sidebar toggle lines. Clicking on them will toggle the visibility of their associated right and left areas to expand the canvas area. The vertical lines can be dragged horizontally to resize the panels.



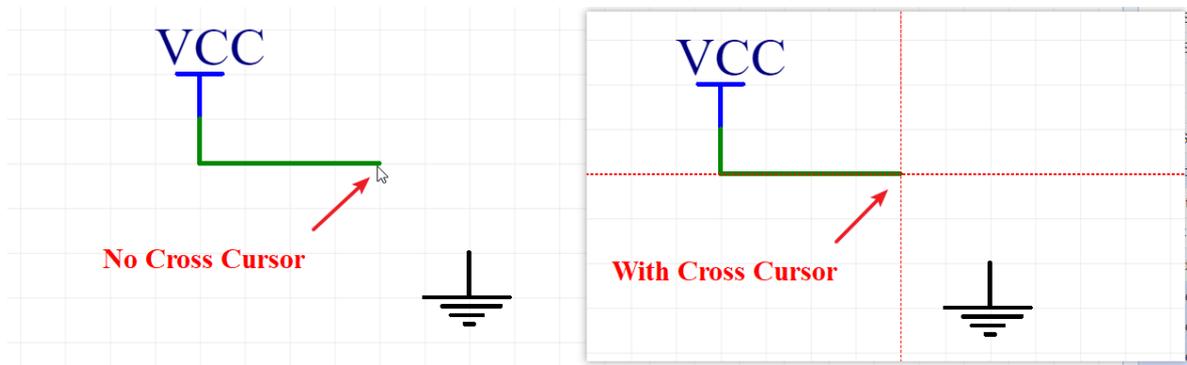
## Cursor Style

Some users do not like the cross cursor, so you can change it to an arrow cursor like in the image below.

Via: Top Menu - View - Cross Cursor



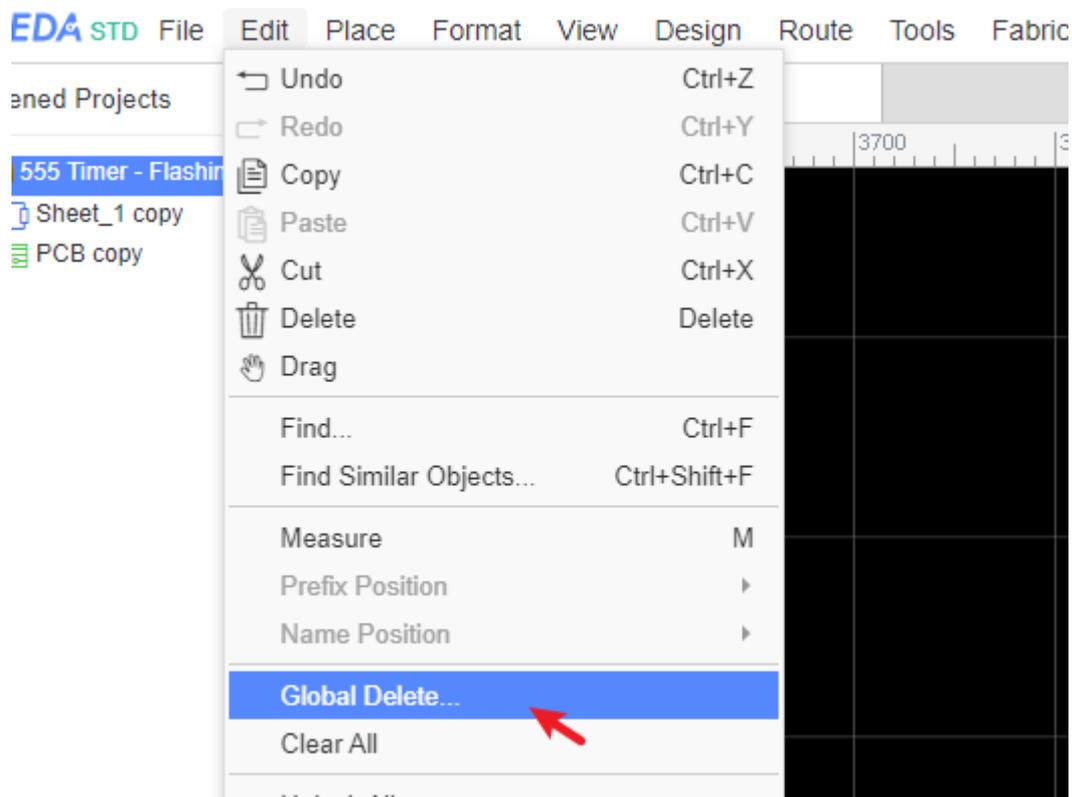
The difference between the two options is shown below.



## Clear and delete

If you think your schematic or PCB looks terrible, and you want to redraw everything, you can do this:

- ◆ **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 the text;
- ◆ The exact left click functionality depends on what item is being selected and in what Canvas the item exists (Schematic or PCB).

## Right clicking

In EasyEDA, right-clicking opens a context sensitive menu:

- ◆ When you are placing a symbol, right-clicking will stop placing and return to select mode. This is the same as the ESC key.
- ◆ 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 or press ESC (sometimes twice).

**Right-Click and drag** Right-clicking and holding the button anywhere in the Schematic, Waveform or PCB Canvas while dragging the mouse will move the canvas around within the EasyEDA window. Holding the middle button and dragging performs the same operation.

## 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). Pressing ESC again returns to select mode.

## Select more shapes

- ◆ CTRL+Left Clicking 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 put your cursor onto the canvas. If the “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 that allows you to edit the text.



Press the enter key to save your changes. Click outside the box or press ESC to discard your changes.

## 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.

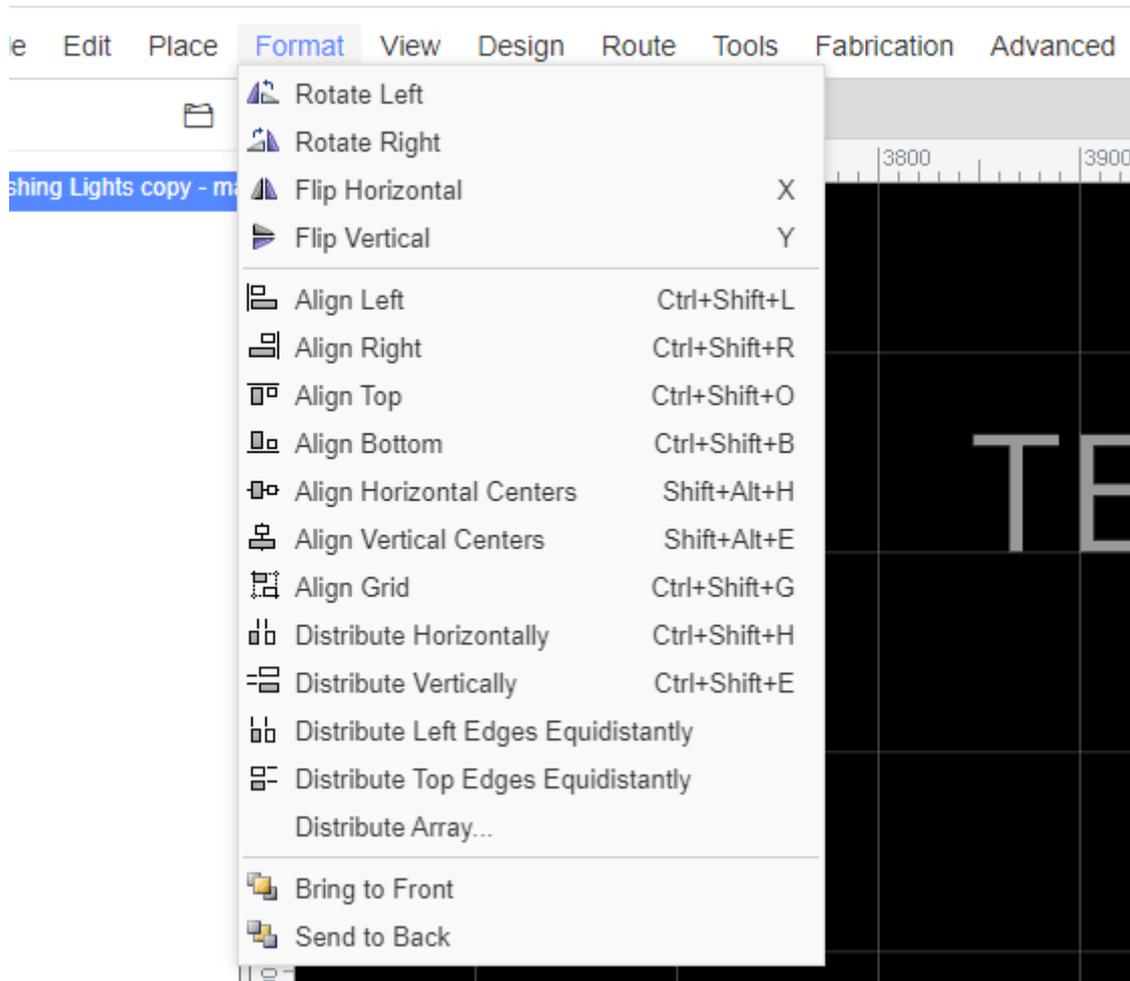
### Tip:

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' .



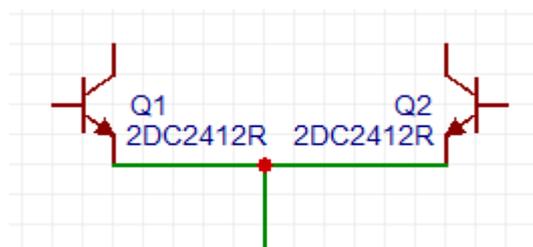
When in PCB view mode you can click the footprint and change its rotation in the right property panel.

**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:

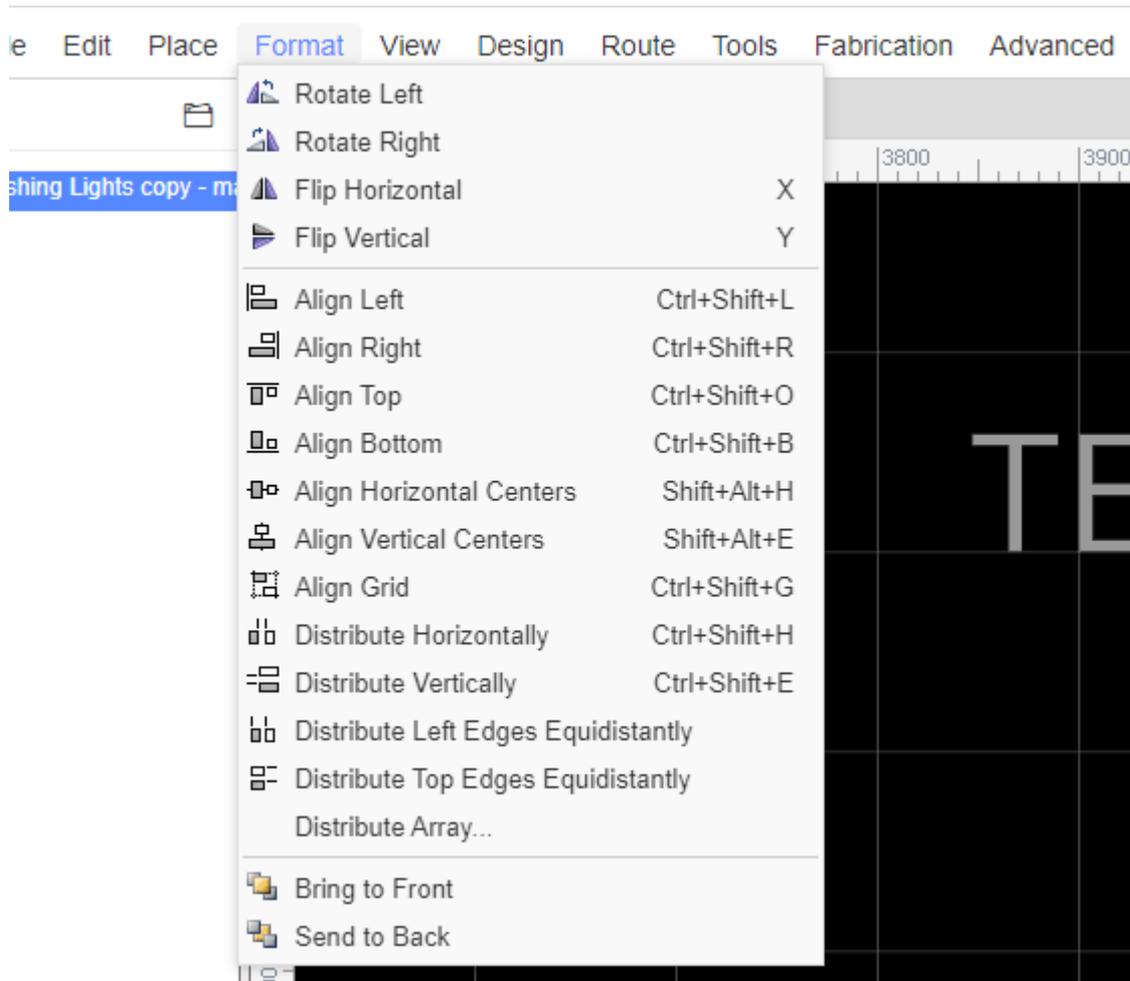
**Rotate and Flip > Flip Horizontal or Flip Vertical** from the toolbar,

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

Notice: Footprint does not support the flip command.

## Align

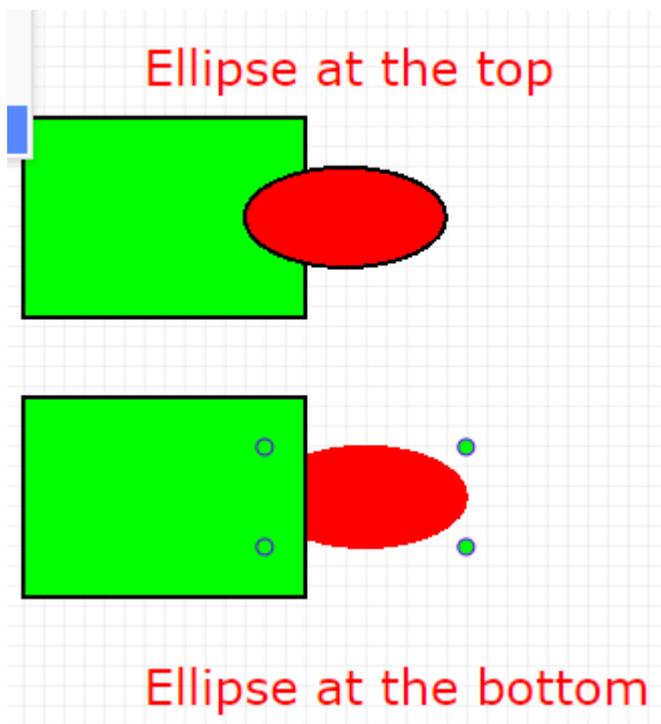
EasyEDA provides many align option features, you can align your symbols or footprints very easily using: Top menu - Format - Align. There are also icons on the toolbar for this.



## Bring to Front and Send to Back

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

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

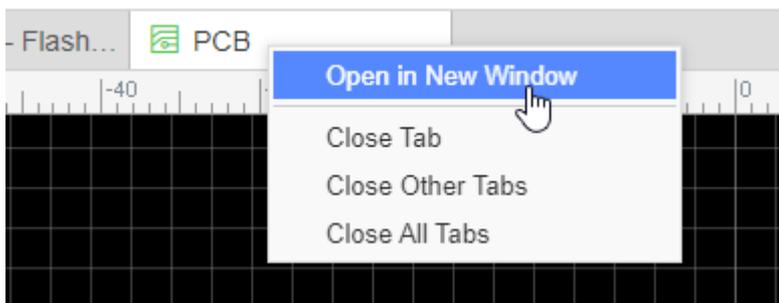


## Multiple Windows

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"

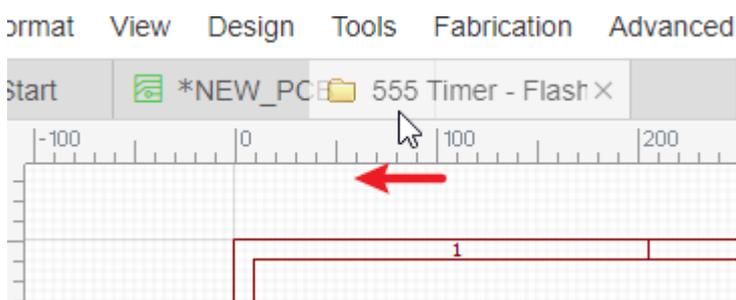


3. This will open the document in a new window, then you can do the cross probe: Click the component, pads, click the Design Manager list, the "Cross Probe and Place" also works.

## Documents Tab Switch

It is easy to modify the tab positions of your documents.

Simply drag the tab location, or use the hotkeys SHIFT+1 and SHIFT+2

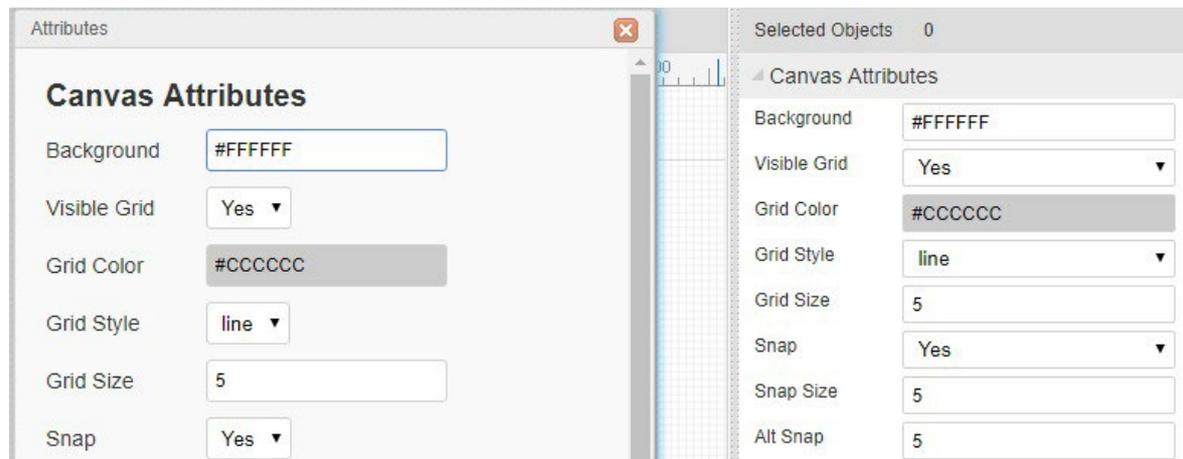


# Experiment no: 07

## Experiment Title: Basics of Schematics

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.

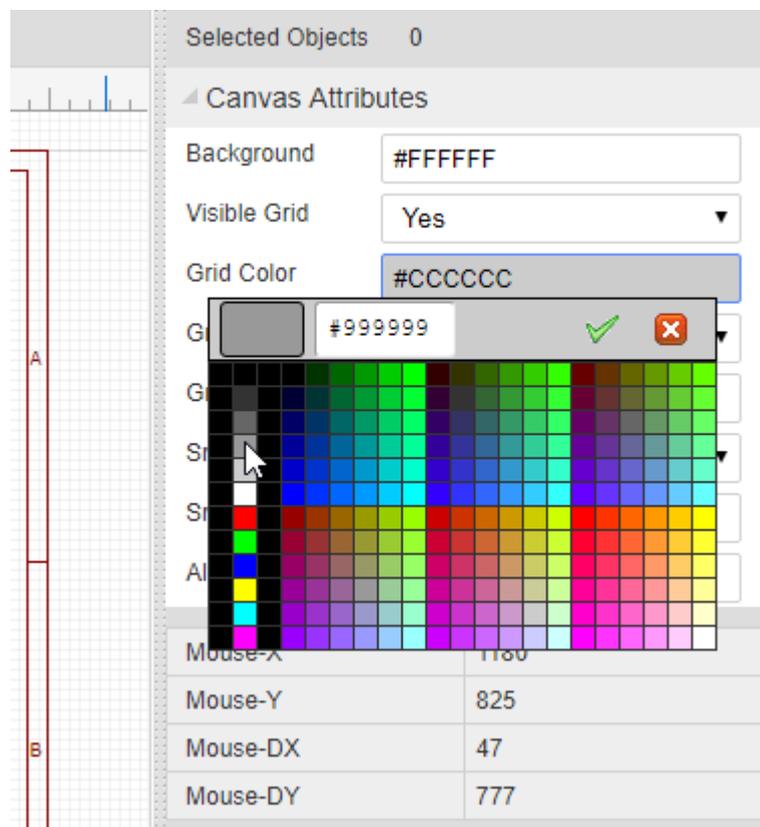


As described earlier, background and grid colours and the style, size, visibility and snap **attributes** of the grid can all be configured.

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.

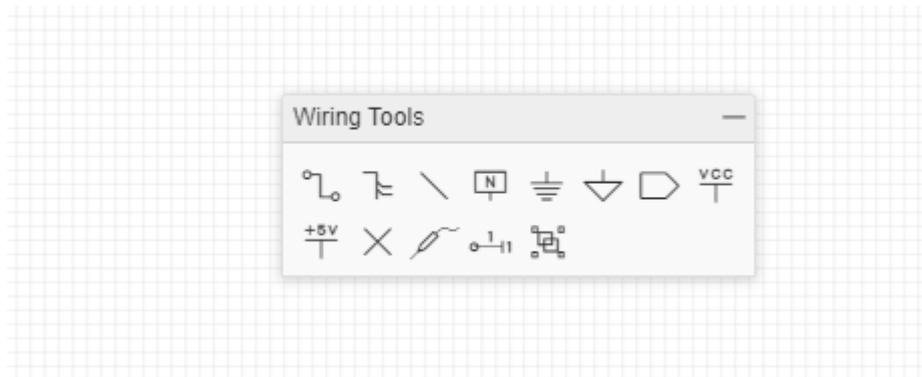
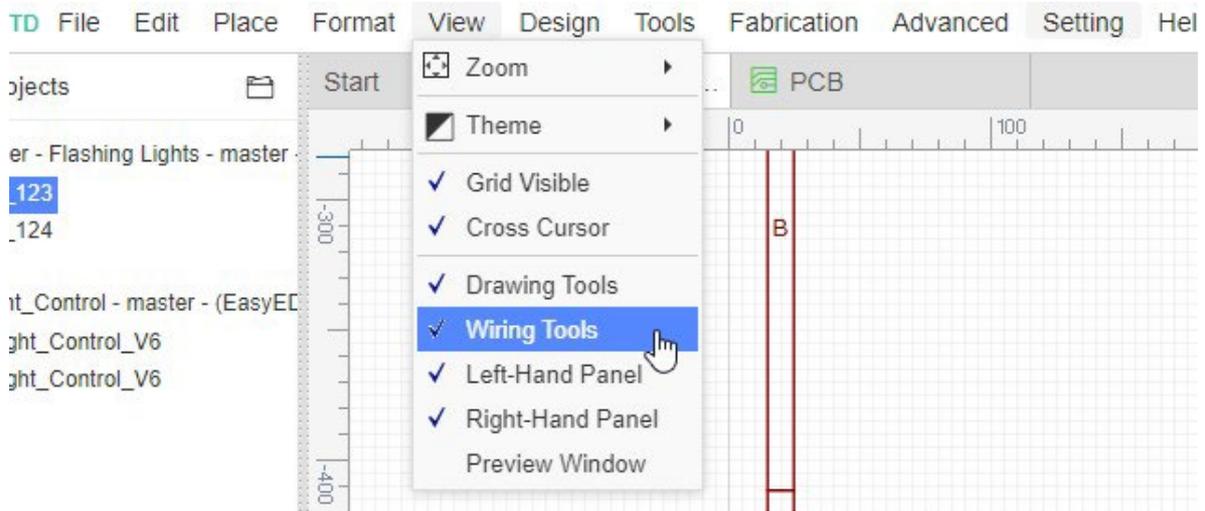
- **ALT Sanp:** Snap size when pressing the `ALT` key.

---

## Wiring Tools

---

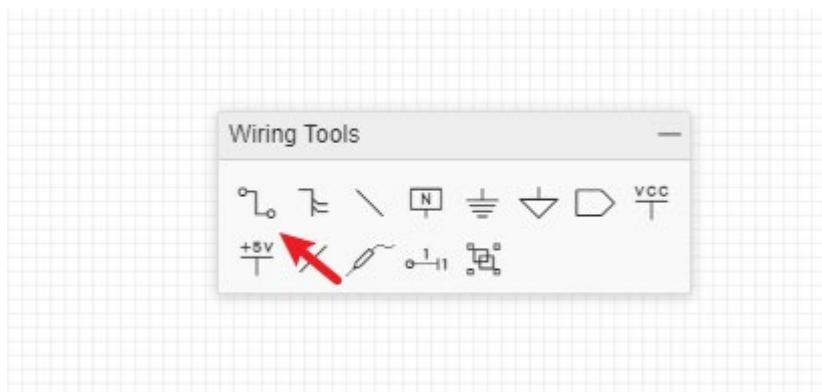
If you have hidden your tools , you can open them from here:  
 Top toolbar **Top Meun > View > Wiring Tools...**



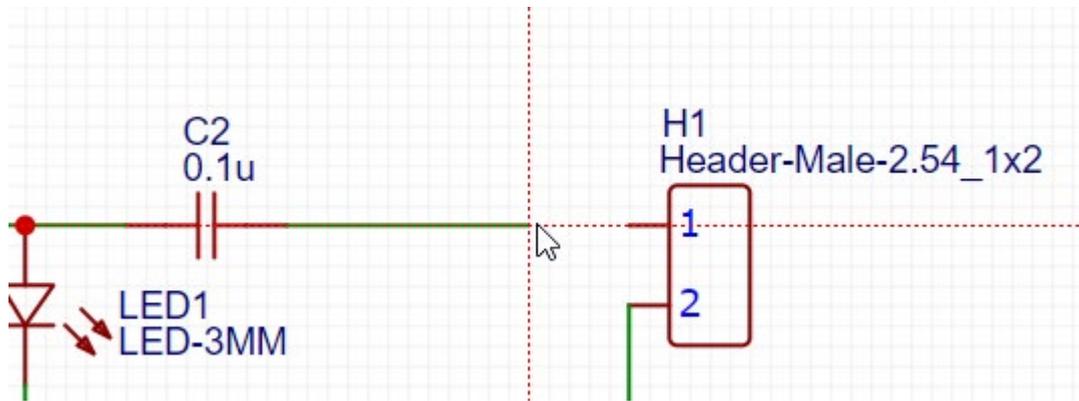
**Note:** All of the commands in Wiring Tools are electronics related. Don't use a wire when you just need to draw a line, shape or an arrow: use Drawing Tools instead.

## Wire

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

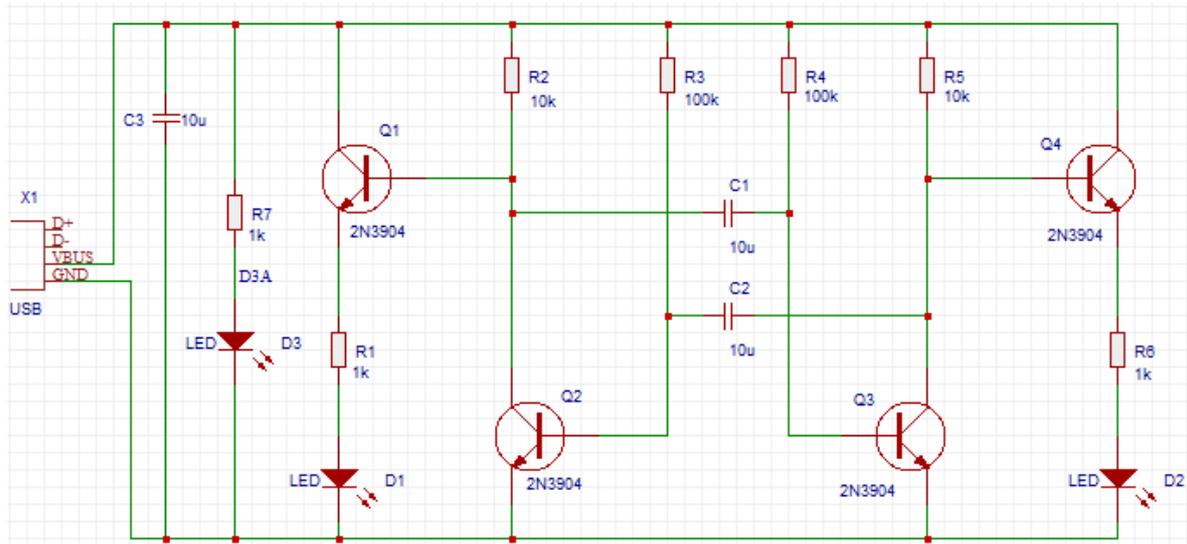


1. Click the **Wire** button from the **Wiring Tools** palette.
2. Press the **W** hotkey.
3. Click on the end of a component pin (where the grey pin dot appears if you select the component):



EasyEDA automatically enters **Wire** mode.

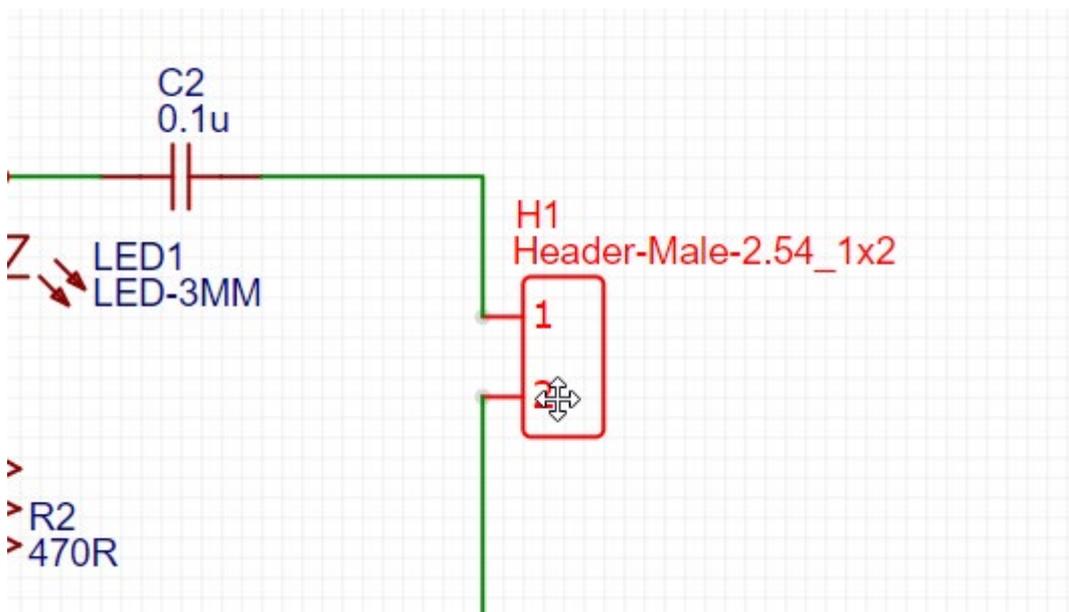
Here is a screenshot of the **Astable Multivibrator LED** project schematic after wiring:



### 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.

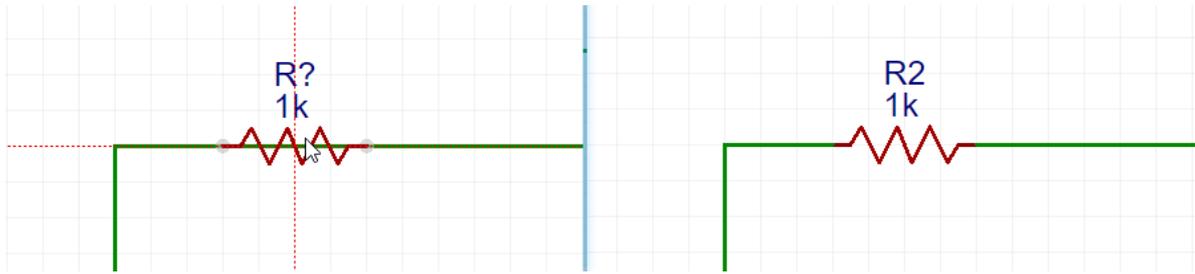
When moving selected components using the mouse, they will drag attached wires with them ( "rubber band" ) to some extent but please be aware that the rubber banding feature has some limitations. When moving selected components most wire will move vertically and horizontally. Using the arrow keys will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



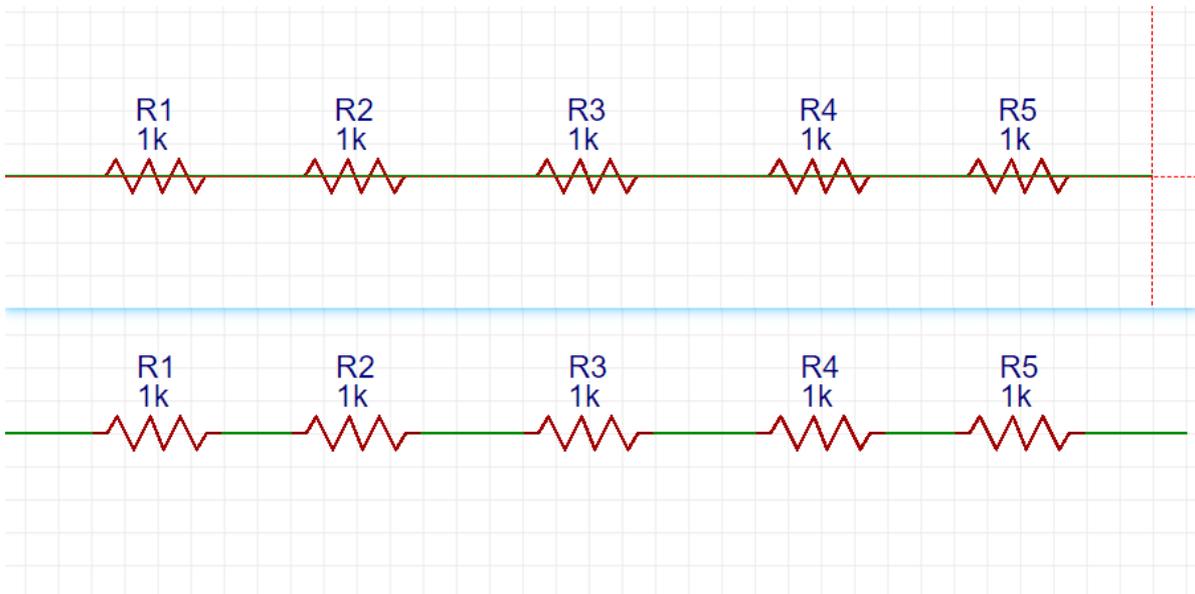
A selected wire can be moved directly by clicking on it using the mouse or by the arrow keys. If a wire is selected by clicking on it using the mouse then green grab handles will appear at the ends and vertices.

### Auto adjust connection

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

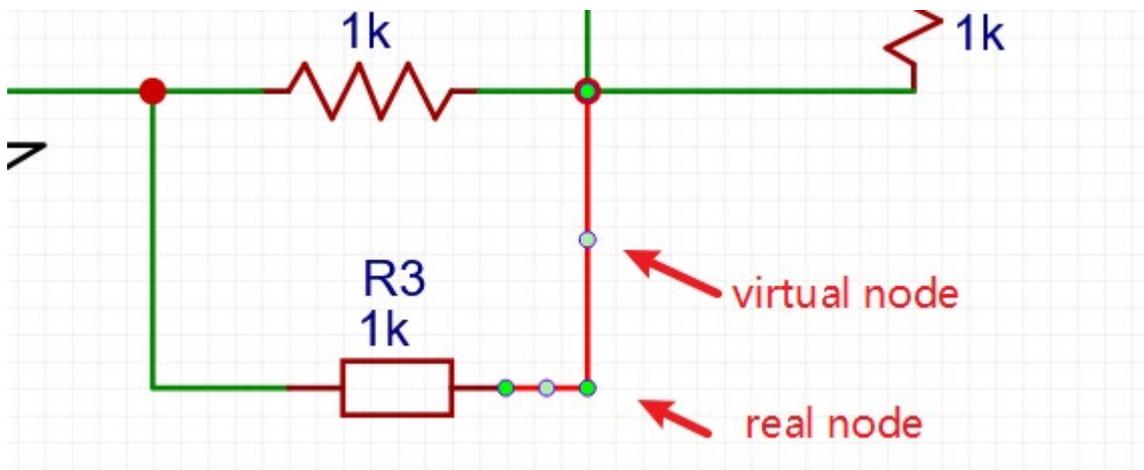


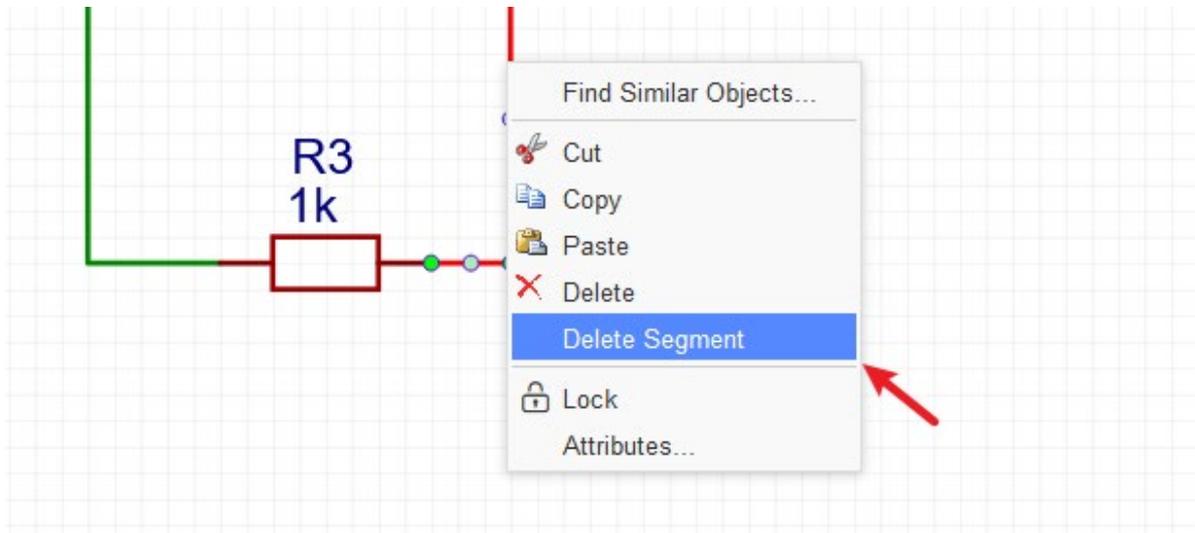
When you want wiring a series of resistors which are in a row, you can just wire through them, and then you will find they all be connected.



### Wire Node

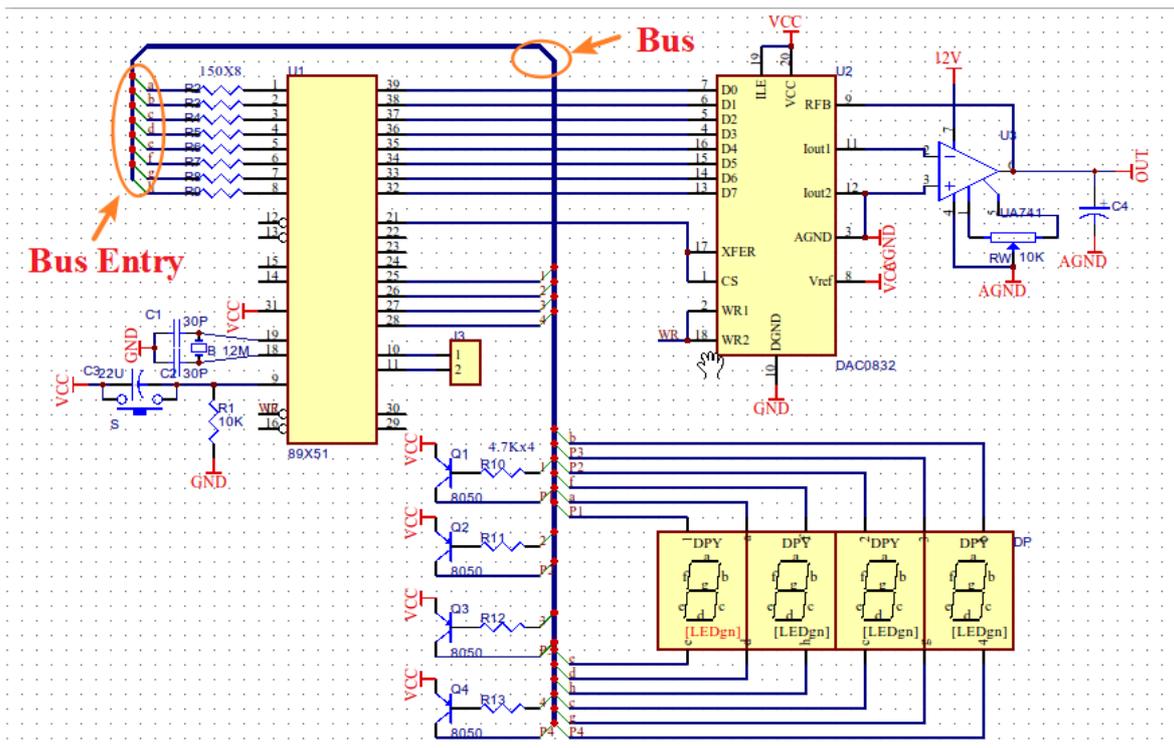
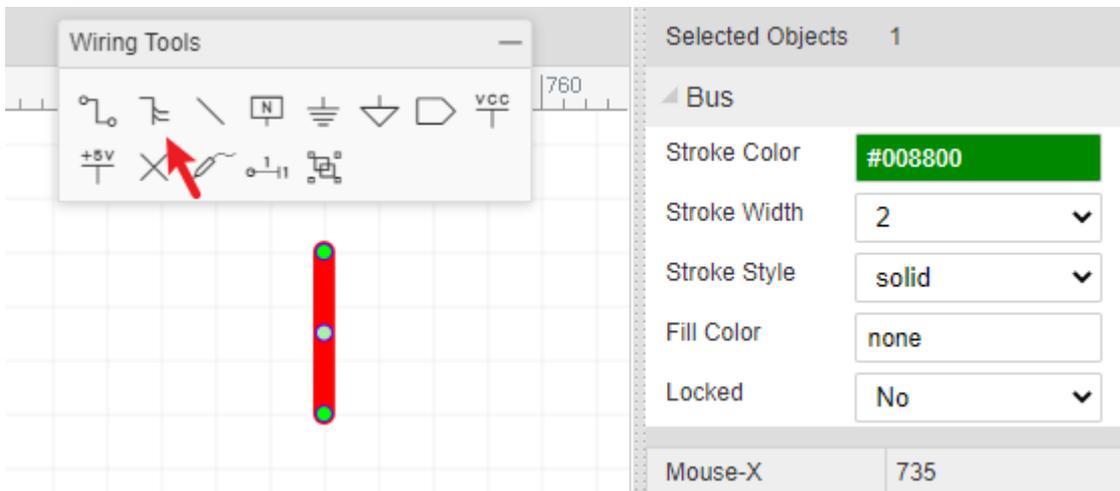
When you click on the wire, you can see the nodes on the wire, where the white is the virtual node, the red is the real node, drag the virtual node to generate the real node, and right-click delete the line segment is to delete the line segment between the real nodes.





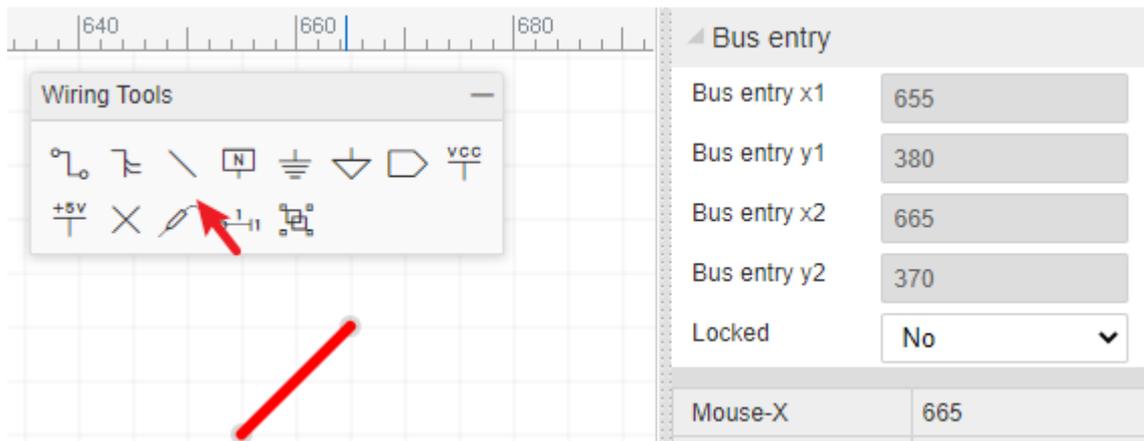
## Bus

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

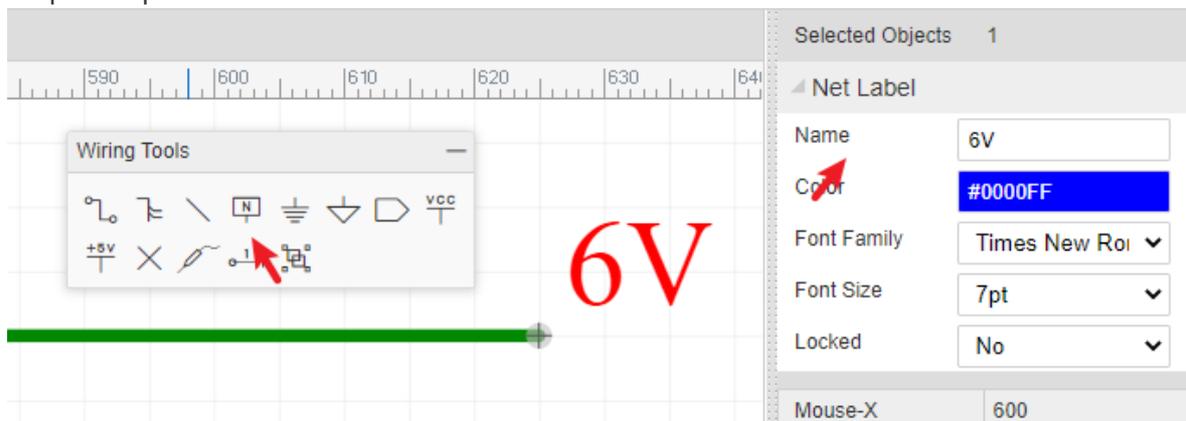
If you decide to wire with `Bus`, the `Bus Entry` must connect to Bus and other nets with wires, such as in the above image.



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:

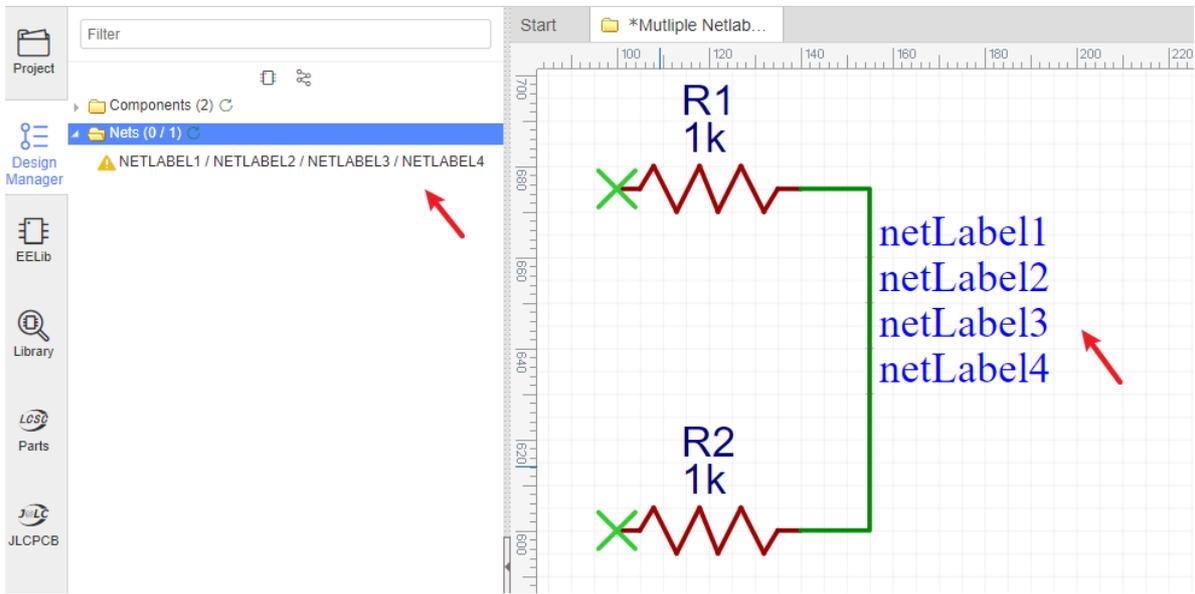


You can change its name and colour. If you only want to change its name, it may be easier to just double click the netlabel.

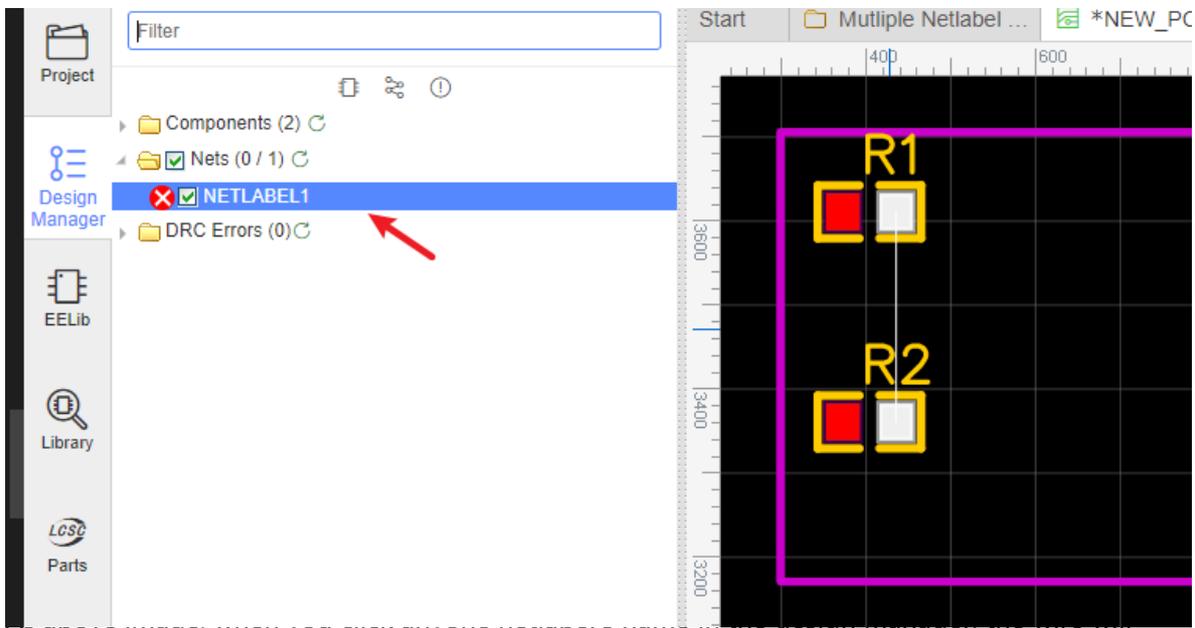
### Multi-NetLabels in One Wire

EasyEDA support mutil-netlabel in one wire now.

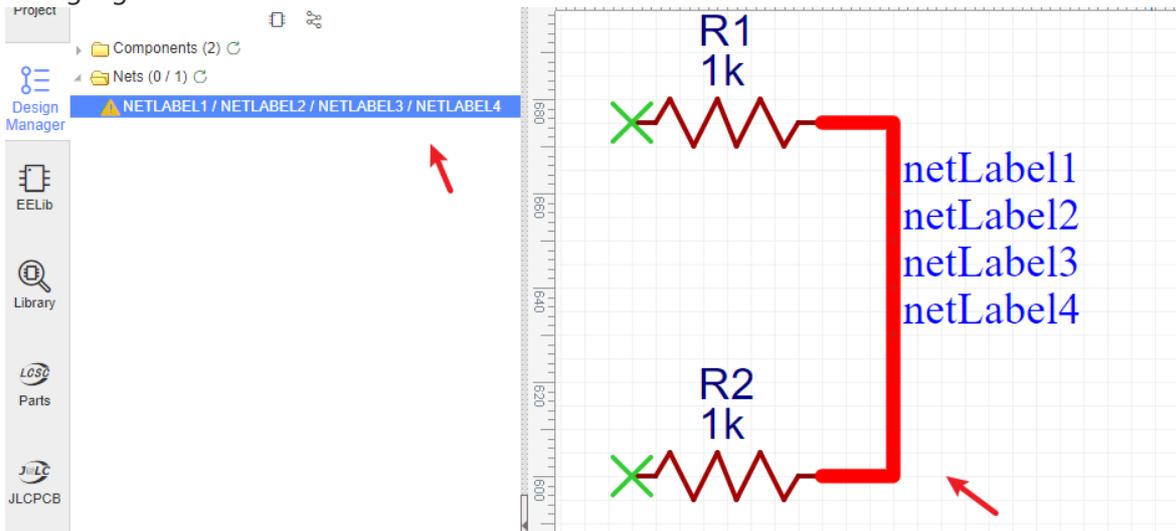
When you convert the schematic to the PCB, the editor will choose the first netlabel you placed as the net name for this wire, as below NETLABEL1.



Convert to PCB:



be highlighted.



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

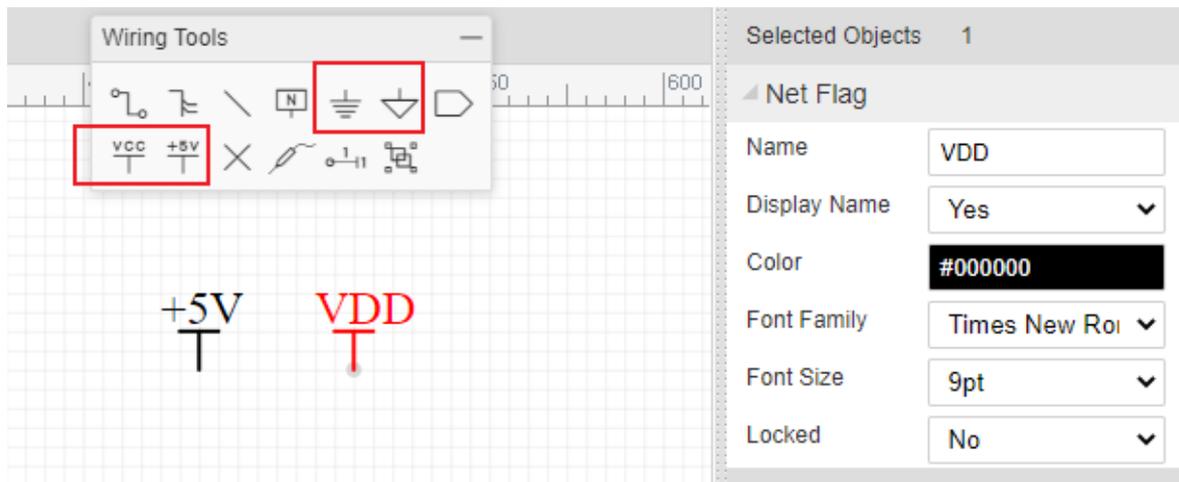


#### Notice:

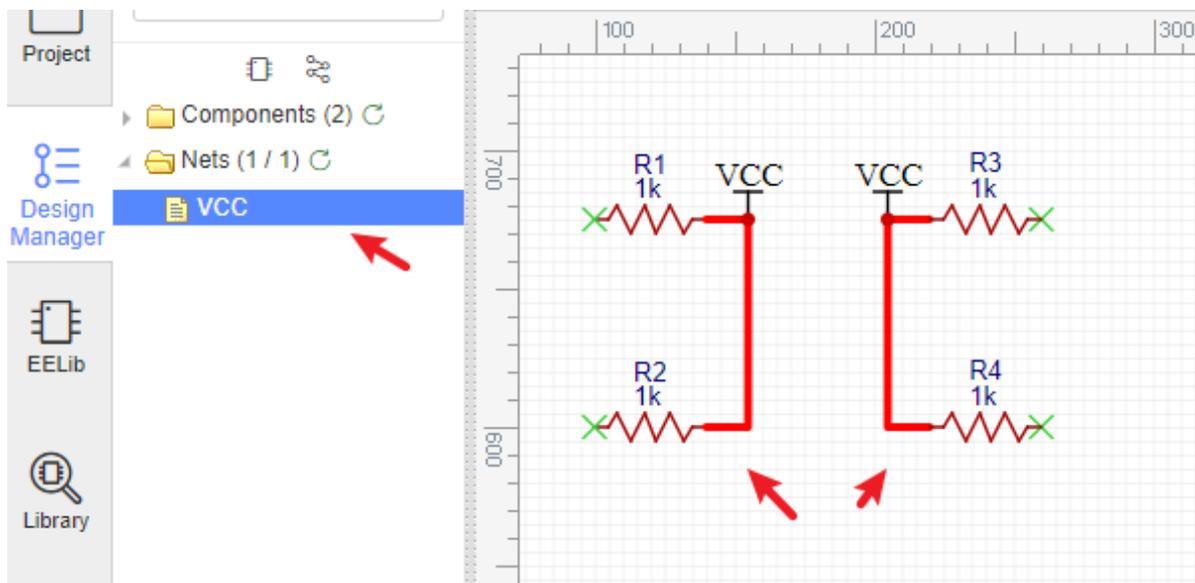
- If wire 1 has 3 netlabels A B and C, and wire 2 has netlabel A, then wire 1 and wire 2 are the same net.
- Netlabel/Netflag/Netport/volprobe only support English characters and letters, and Arabic numerals.
- If a part prefix is P1, which has two pins, it will have two nets "P1\_1" and "P1\_2" by default, 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**:



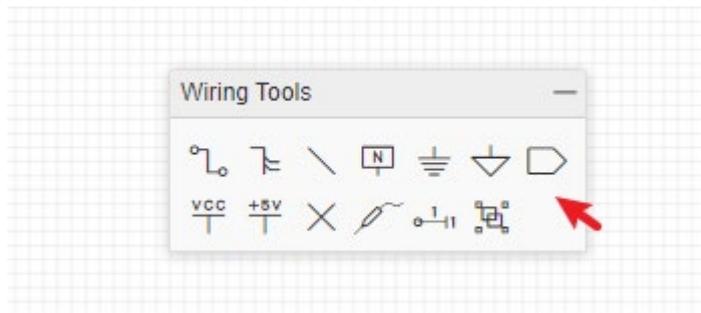
When appear two and more Netflag or Netlabels which are the same name, they will connected with each other.



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

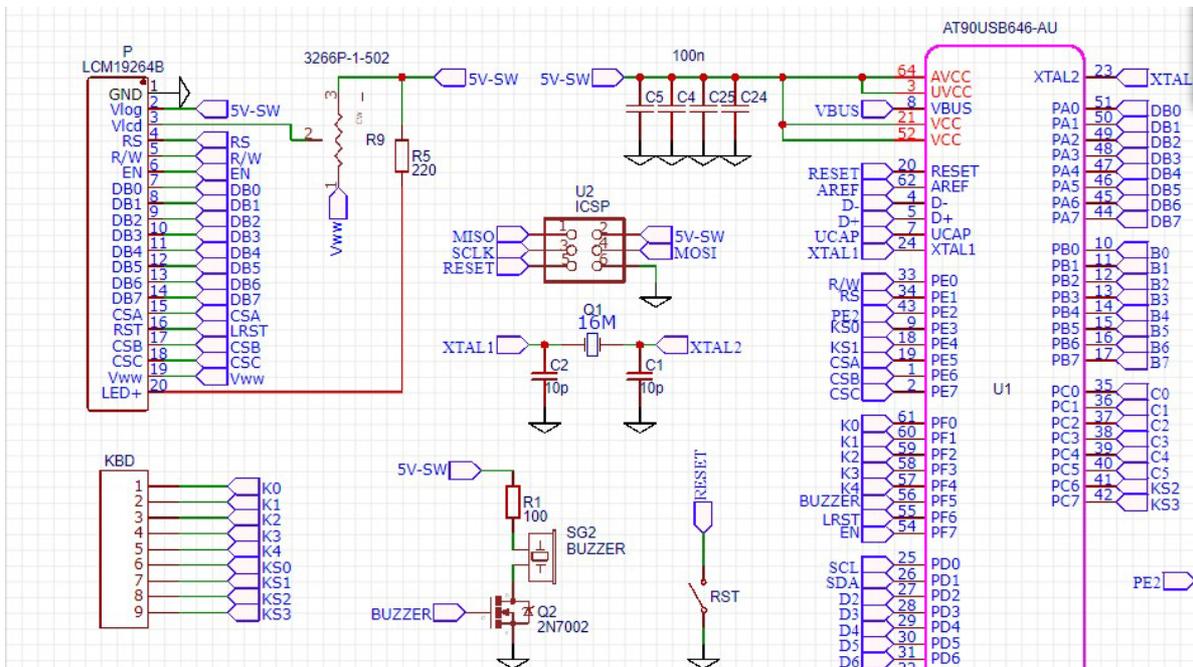
## Net Port

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` :



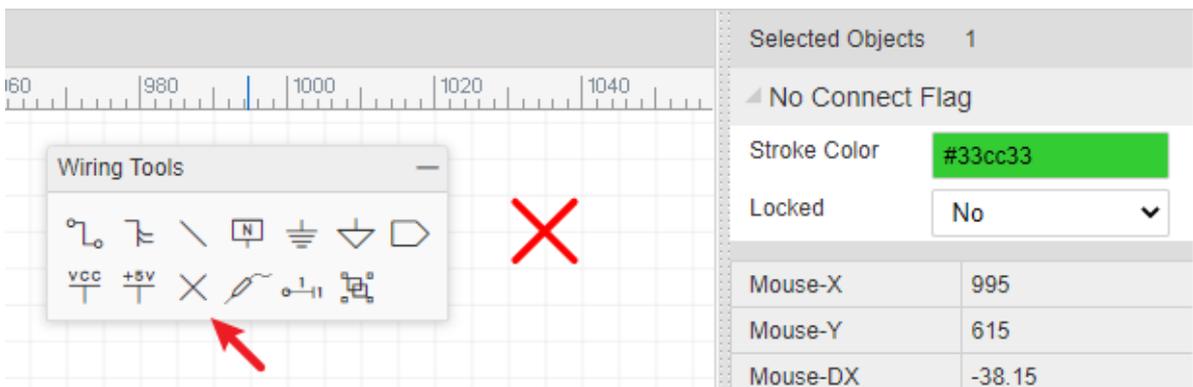
Selected Objects	
Net Port	
Name	netPort1
Display Name	Yes
Color	#0000FF
Font Family	Times New Ro
Font Size	8pt
Locked	No
Mouse-X	975

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

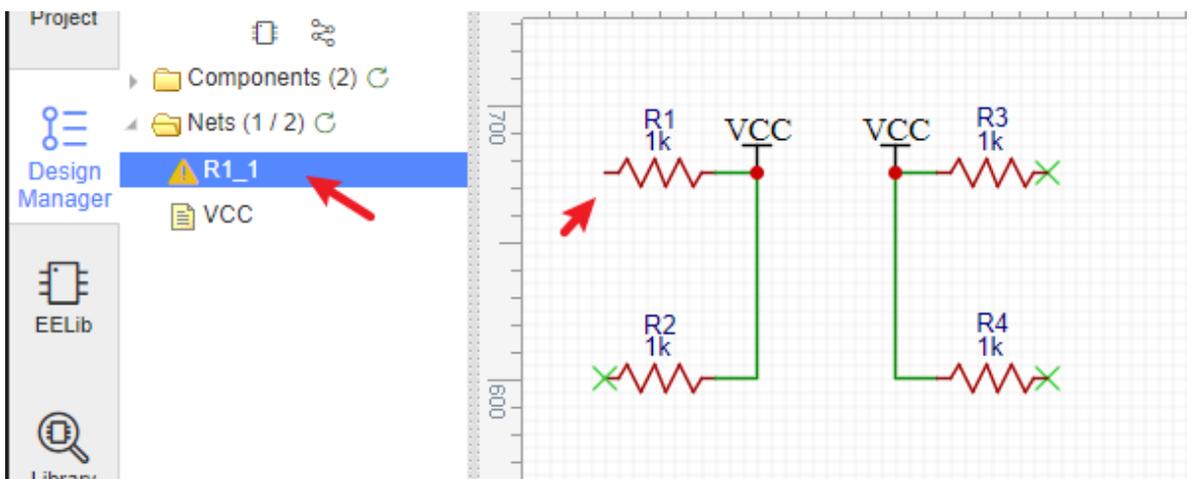


## No Connect Flag

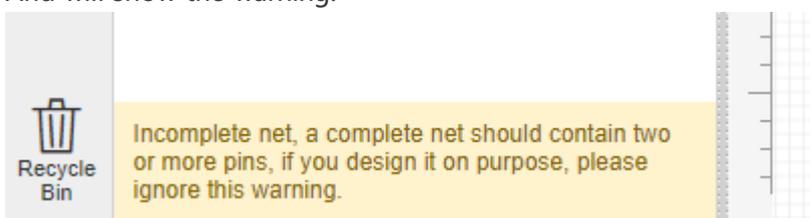
You can find the **NO Connect Flag** via wiring tool,



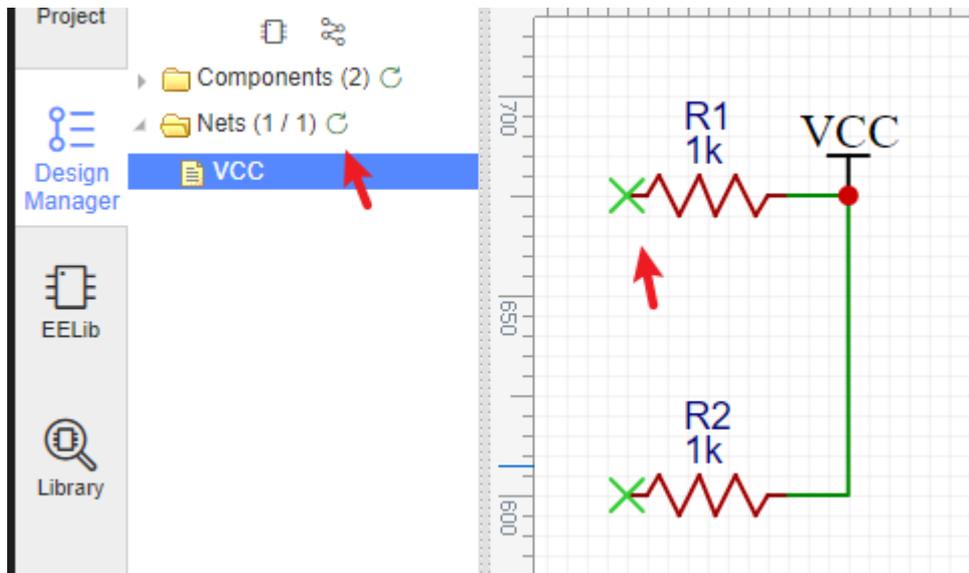
In the below schematic, if you don't add a **NO Connect Flag**, there is an error flag in the nets collection of the design manager.



And will show the warning:

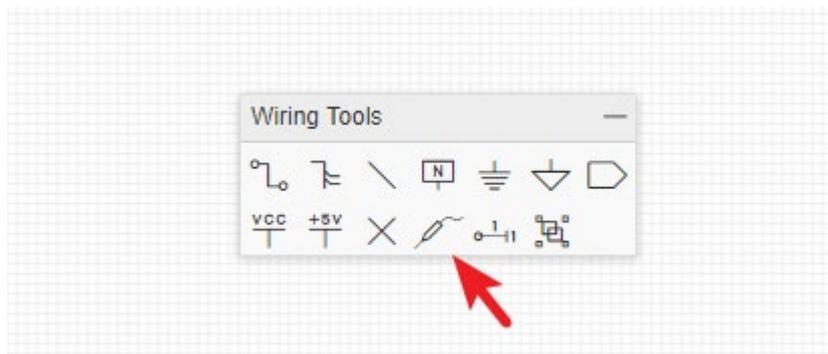


After adding a `NO Connect Flag`, and then refresh the Nets folder, the error disappears.

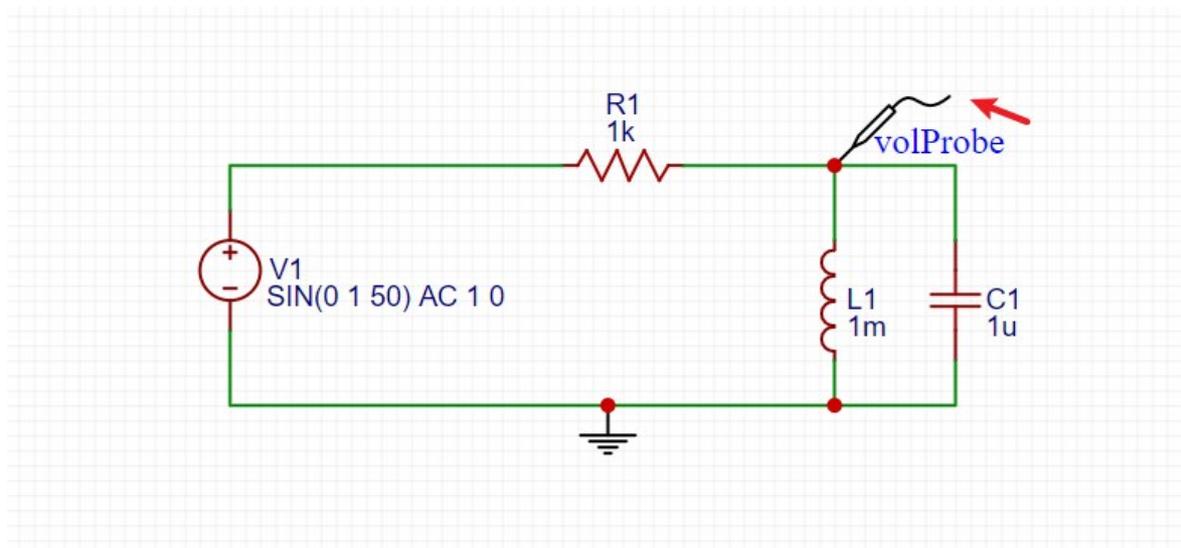


**Note:** `NO Connect Flag` only works on the symbol's pin directly.

## Voltage Probe



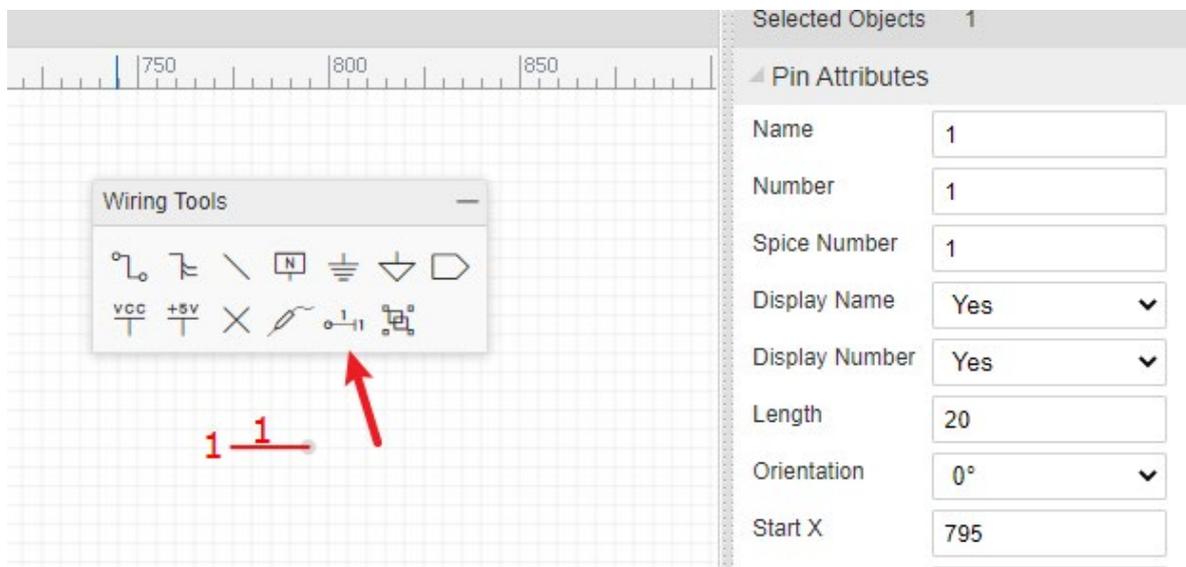
EasyEDA provides a simulation feature for the schematic. After the simulation is running, you will see the waveform where you placed the voltage probes in the circuit.



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

## Pin

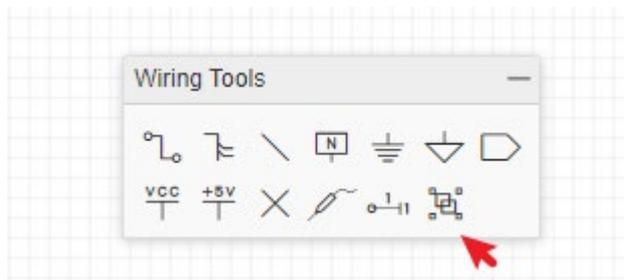
When you create a new symbol in schematic and schematic lib, you must use `pin` to create pins for the new symbol, otherwise your symbol can't be wired with wires.



For more information please refer to the **Symbol Library - Create Symbol** section.

## Group/Ungroup Symbol

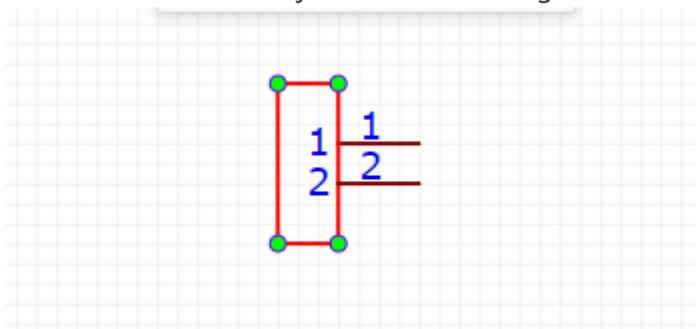
On the **Wiring Tools** palette there is the **Group/Ungroup Symbol...** button.



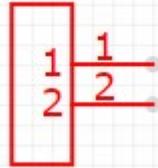
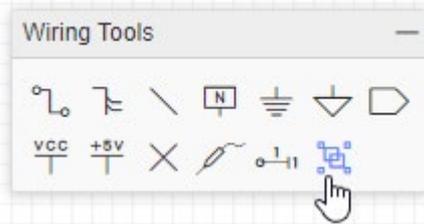
Just like the **Symbol Wizard**, this tool is also for you to quickly create schematic library symbols.

Here's how.

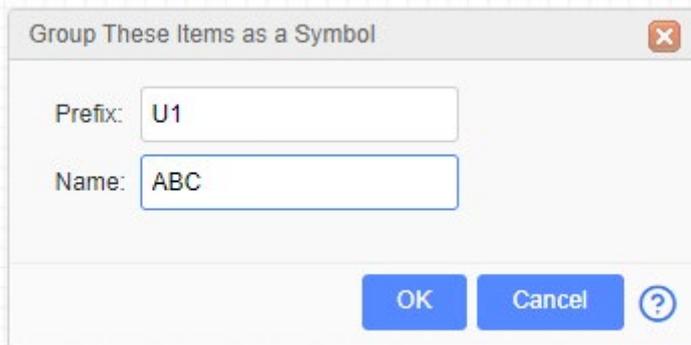
- ◆ Place Pins and other objects such as rectangle



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



- ◆  ed.



So w  
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.

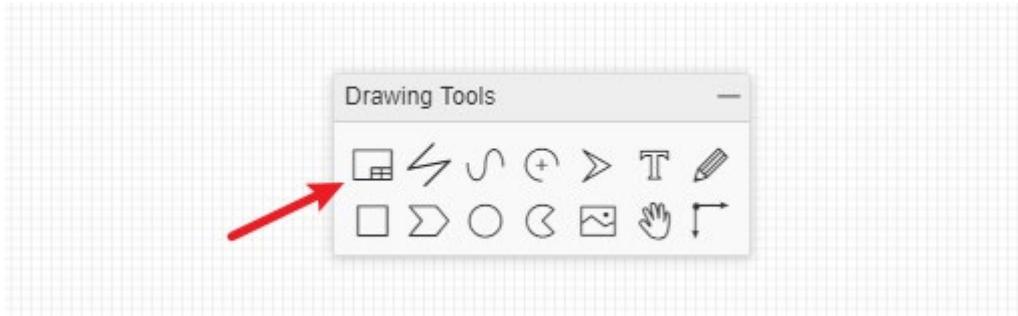
# Experiment no: 08

## Experiment Title: Drawing Tools of EasyEDA

### 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.

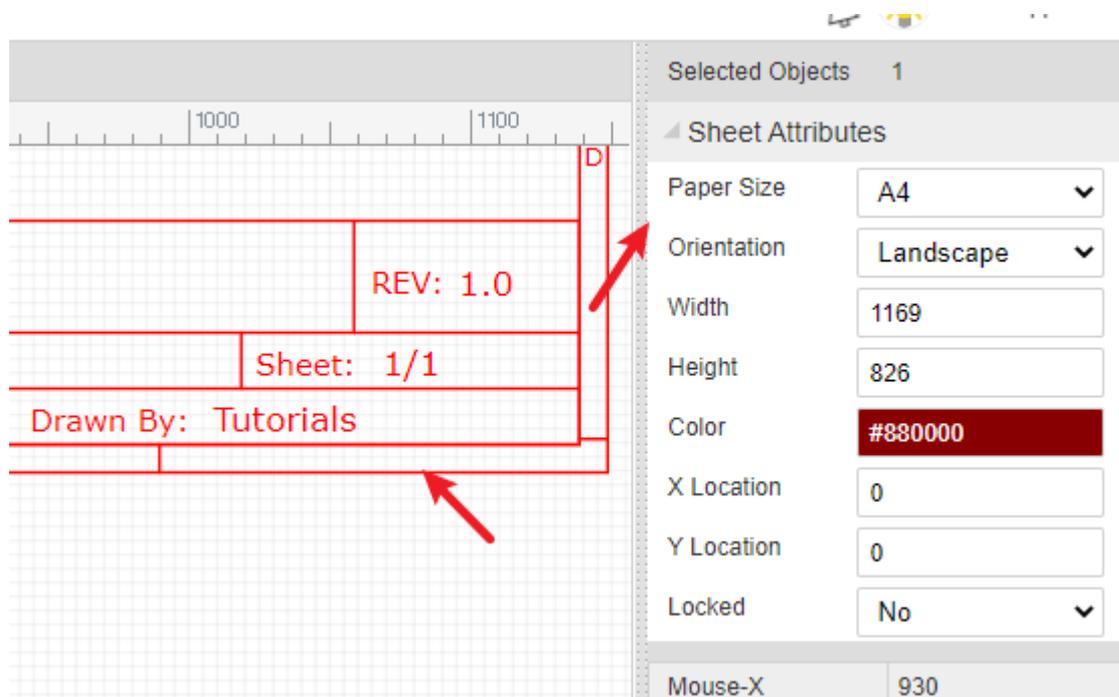
Click the frame/drawing/document button like in the image below:



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:

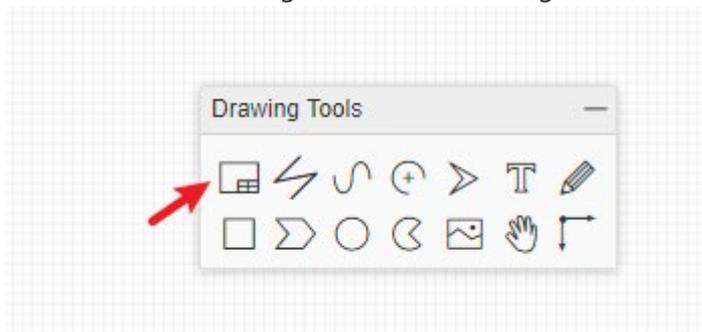


### Custom Sheet

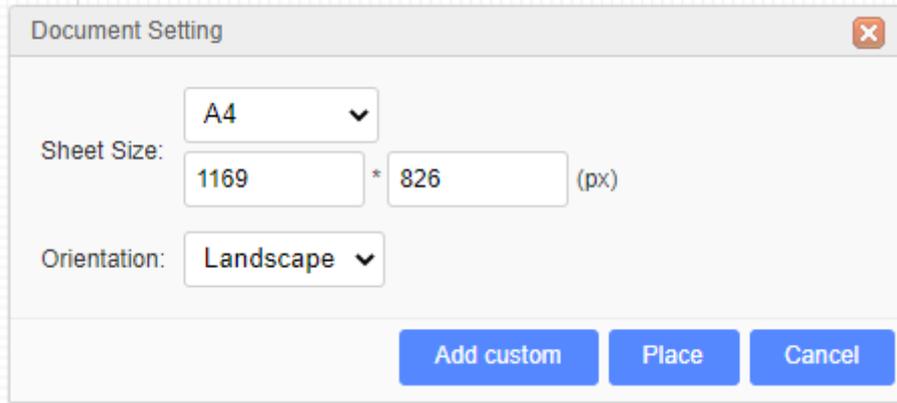
EasyEDA supports the schematic diagram drawing frame required by custom. At present, custom drawings need to be placed manually, and automatic reference of custom drawings is not supported when creating new schematic diagram.

How to create:

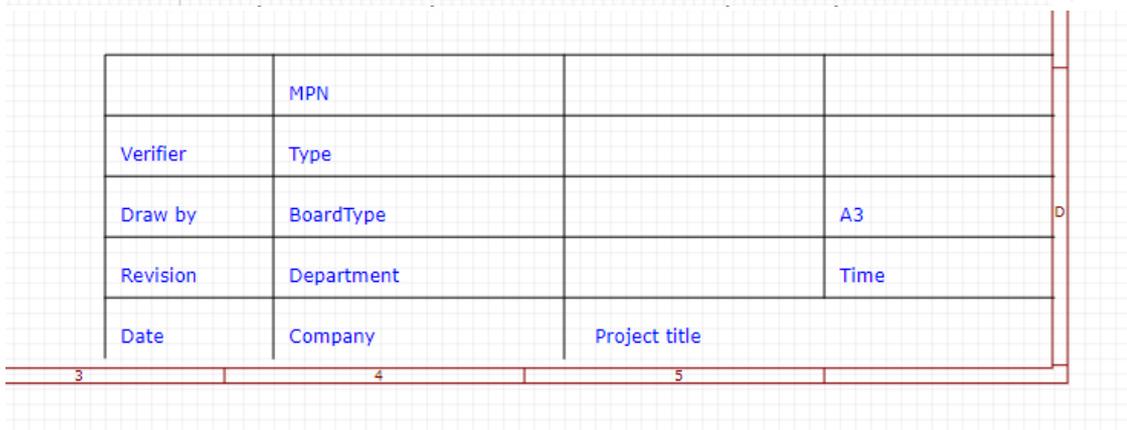
1. Click the "Sheet Setting" button at "Drawing Tool".



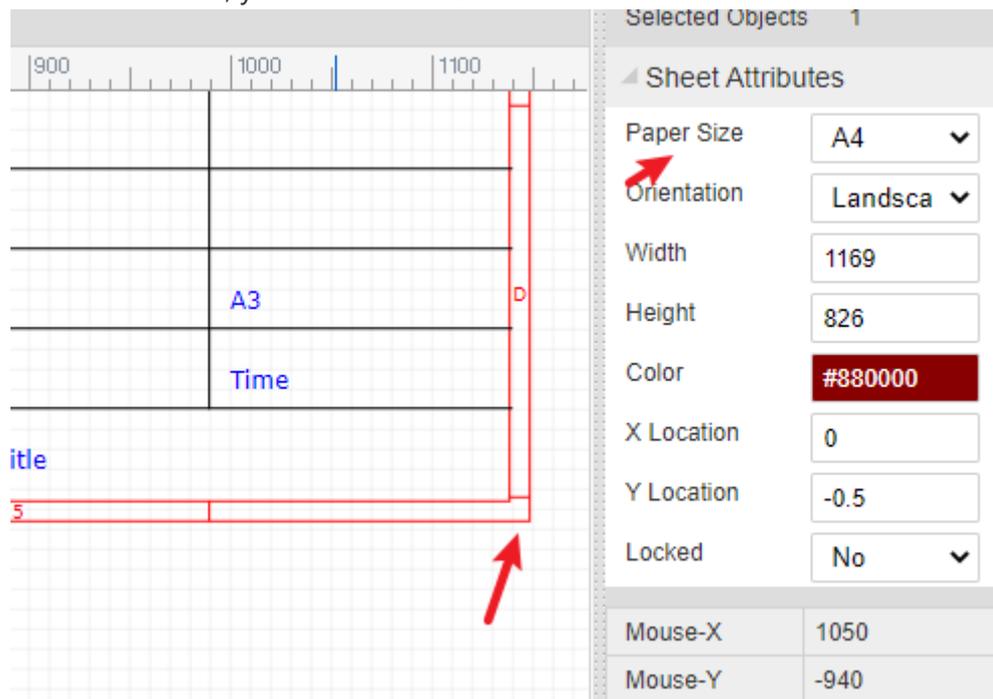
2. Click the 'Custom' button.



- 3.



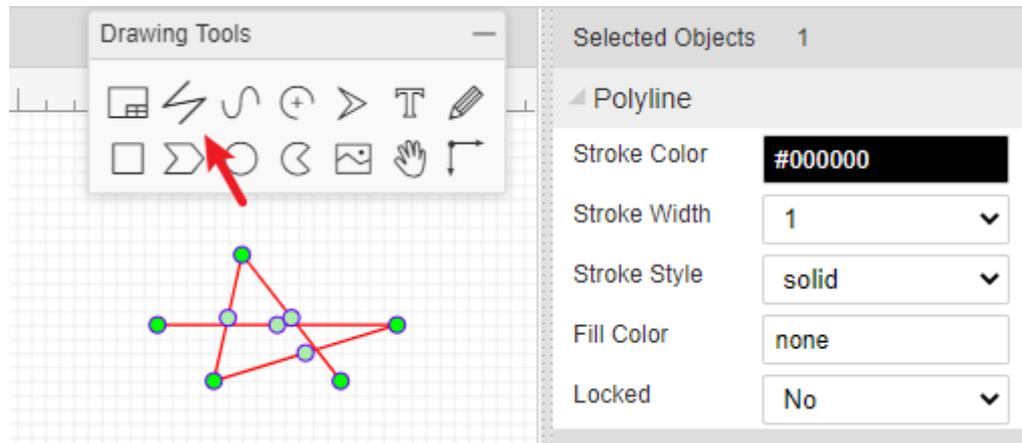
4. Select the outline, you can edit its size.



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

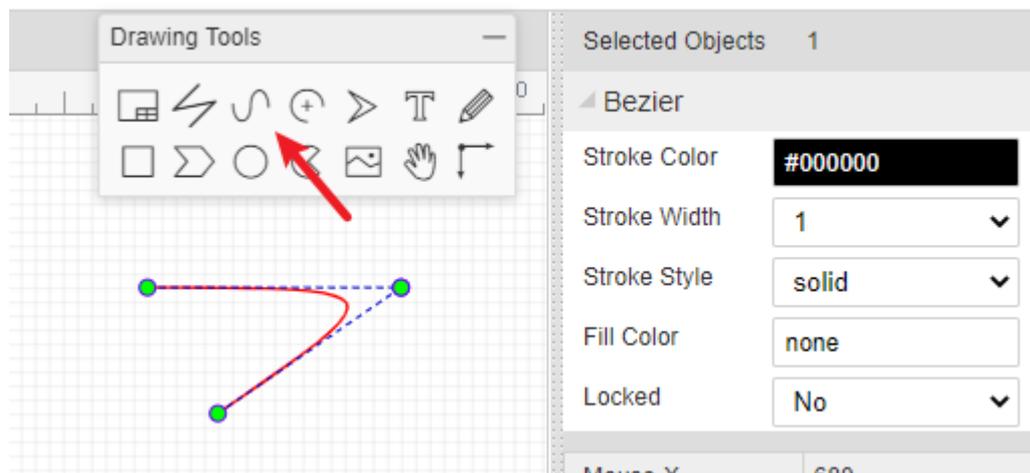
## Line

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



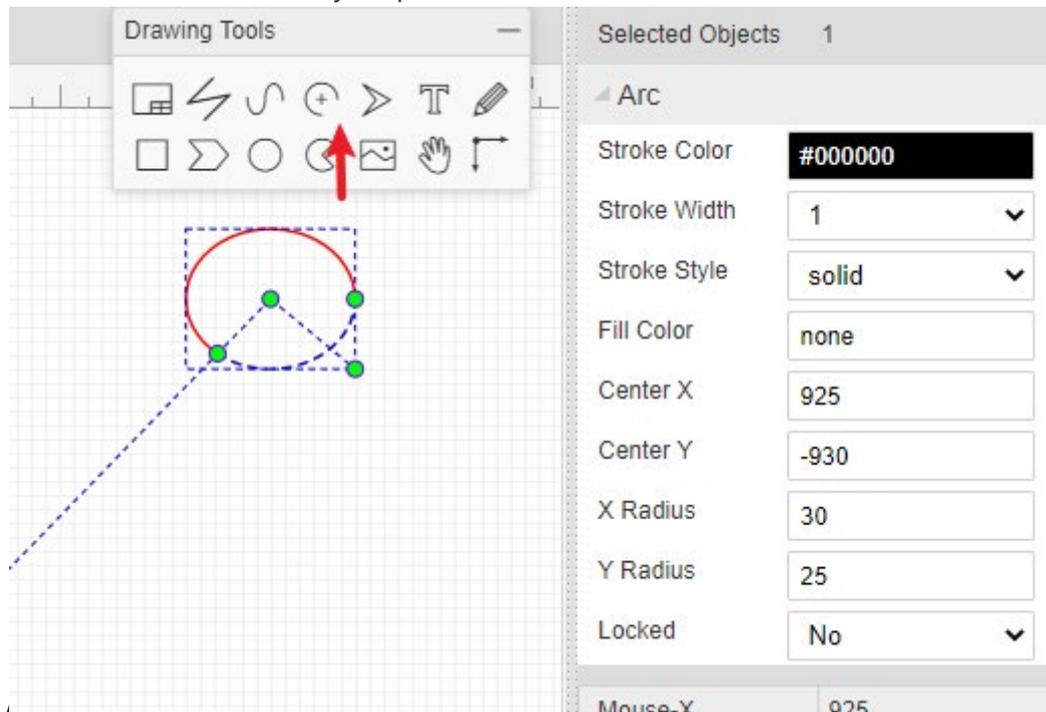
## Bezier

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



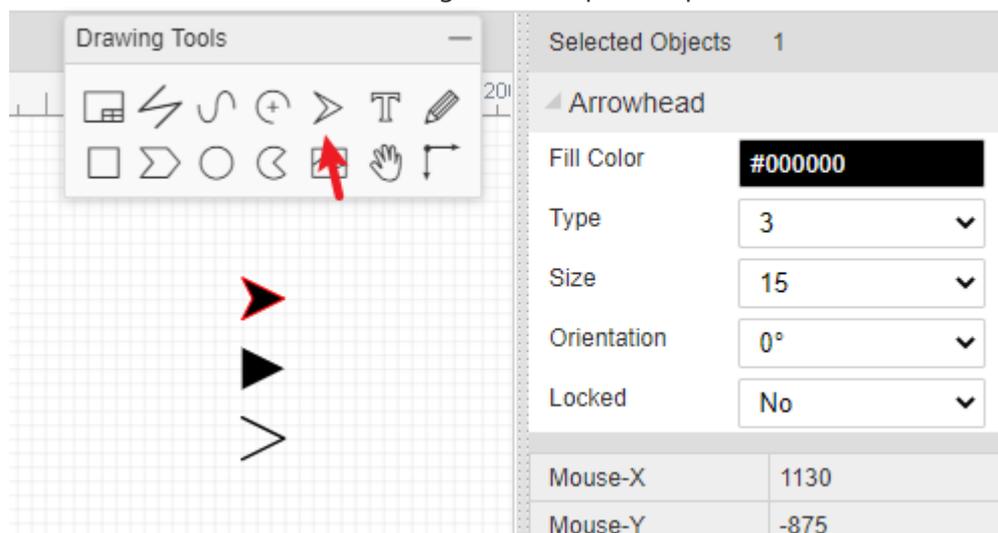
## Arc

You can draw the arc of any shape.



## Arrow Head

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

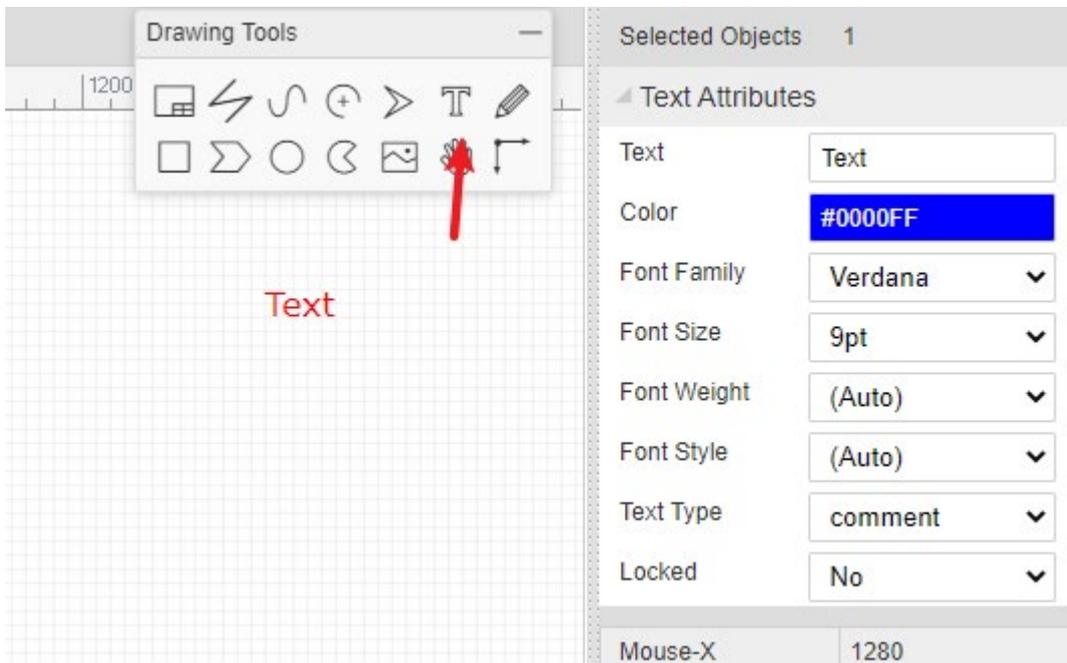


## Text

Text attributes provide many parameters for setting:

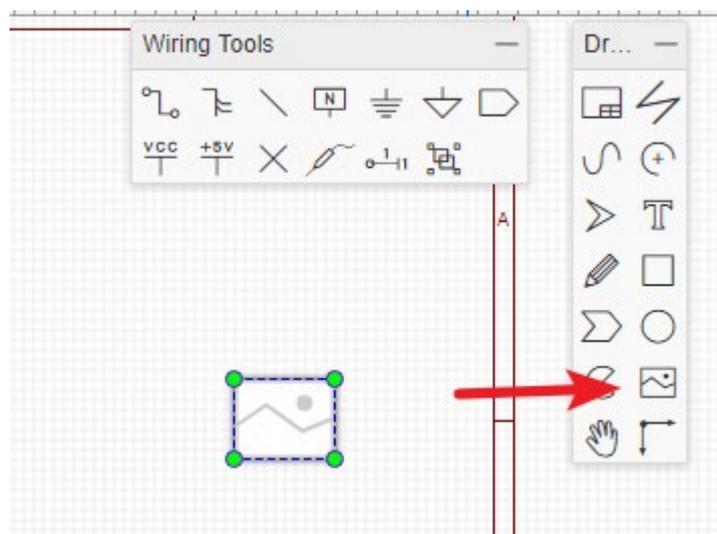
- **Text:** You can change text in inner box or double click the text. For every new text, the default text is `Text`.
- Color:** Defines text color.
- Font-family:** It provides 12 fonts for choosing.
- Font-Size:** Defines Text size.
- Font-weight:** Defines Text weight.
- Font-Style:** It contains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

The editor will remember your last text parameters.

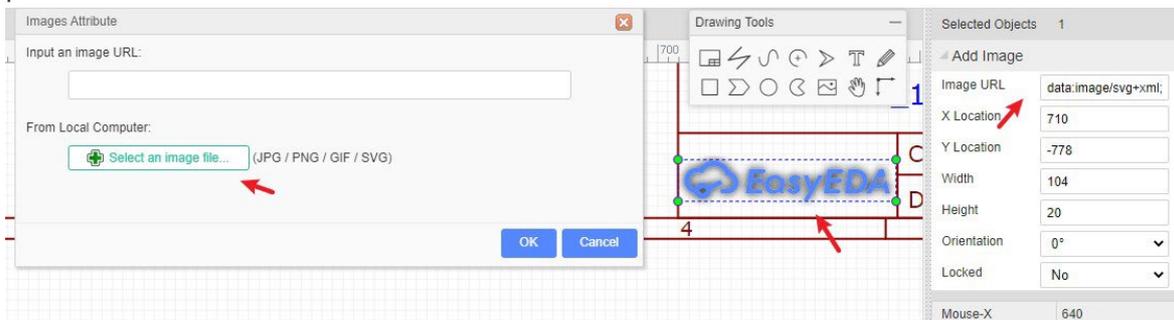


## Image

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



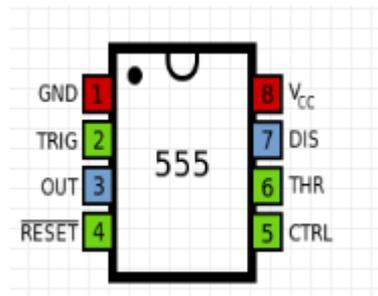
Select the place holder, so you can see the image's attributes in the right hand Properties panel:



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)

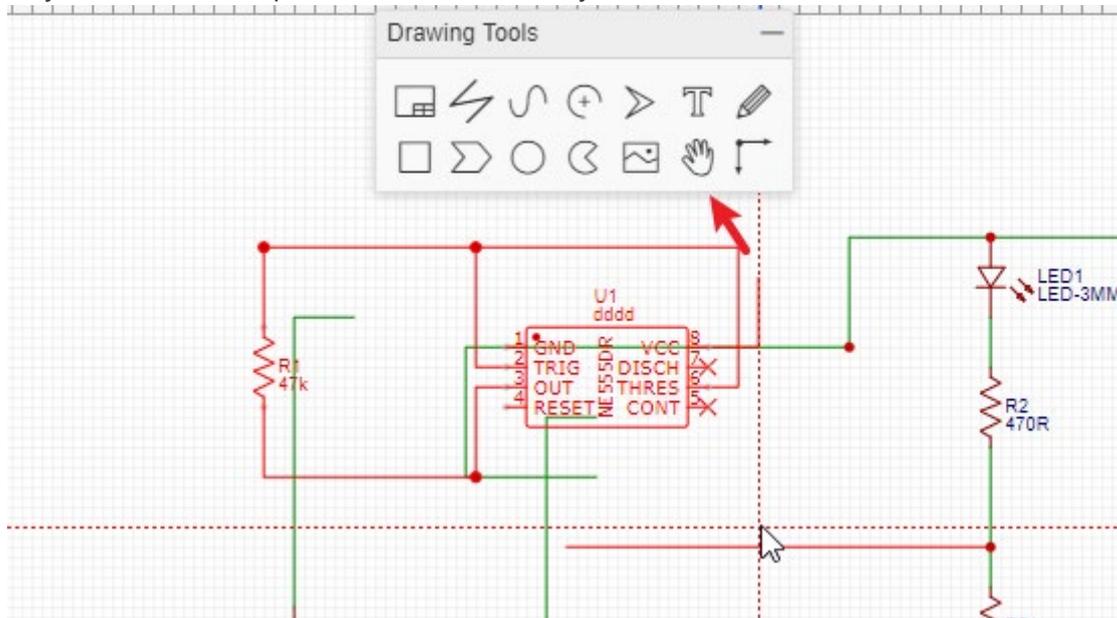
will make your image look like this:



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

## Drag

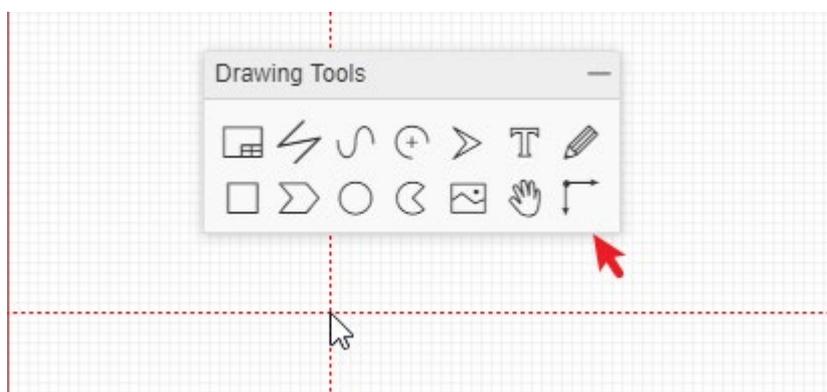
If you want to move some kind of parts and wires, you can use drag, hotkey D. Or you can select the parts and wires area firstly and move them.



## 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**.

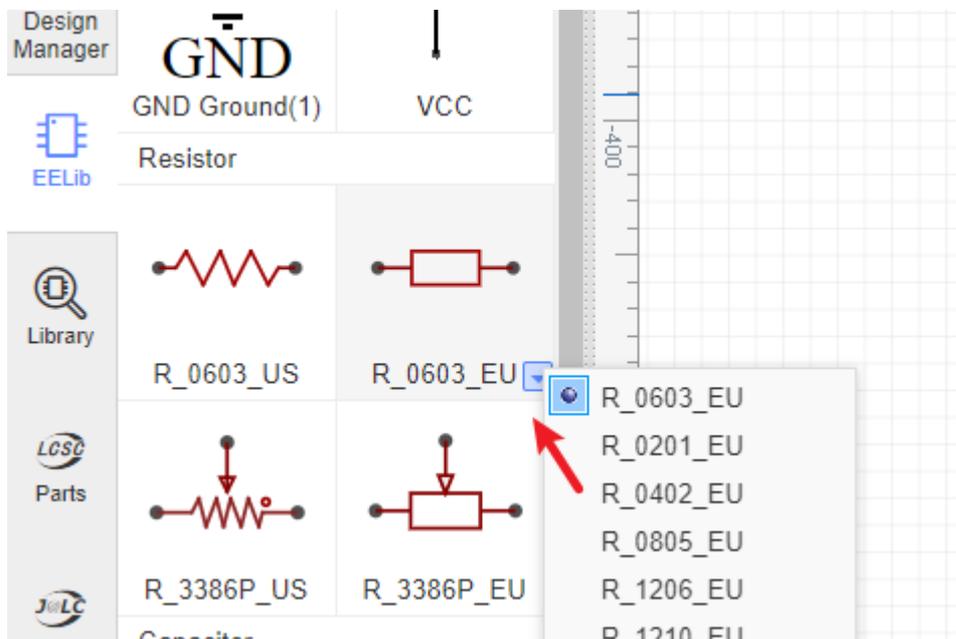


# Experiment no: 09

## Experiment Title: Library Management in EasyEDA

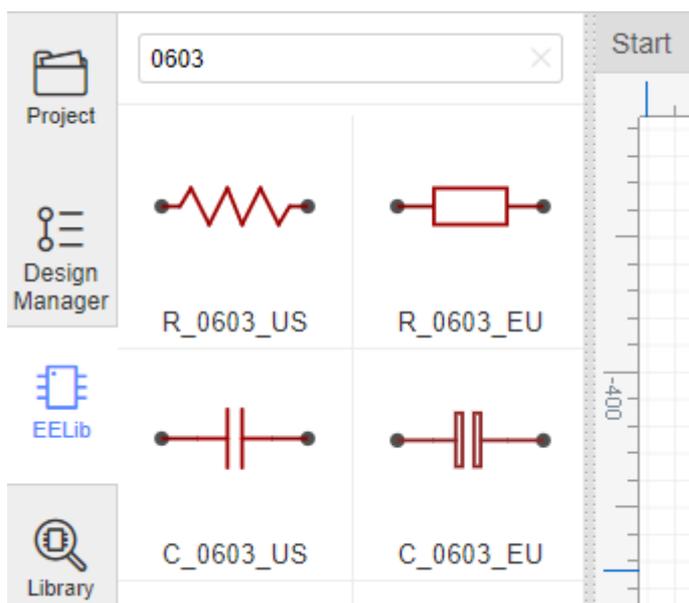
### EELib

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



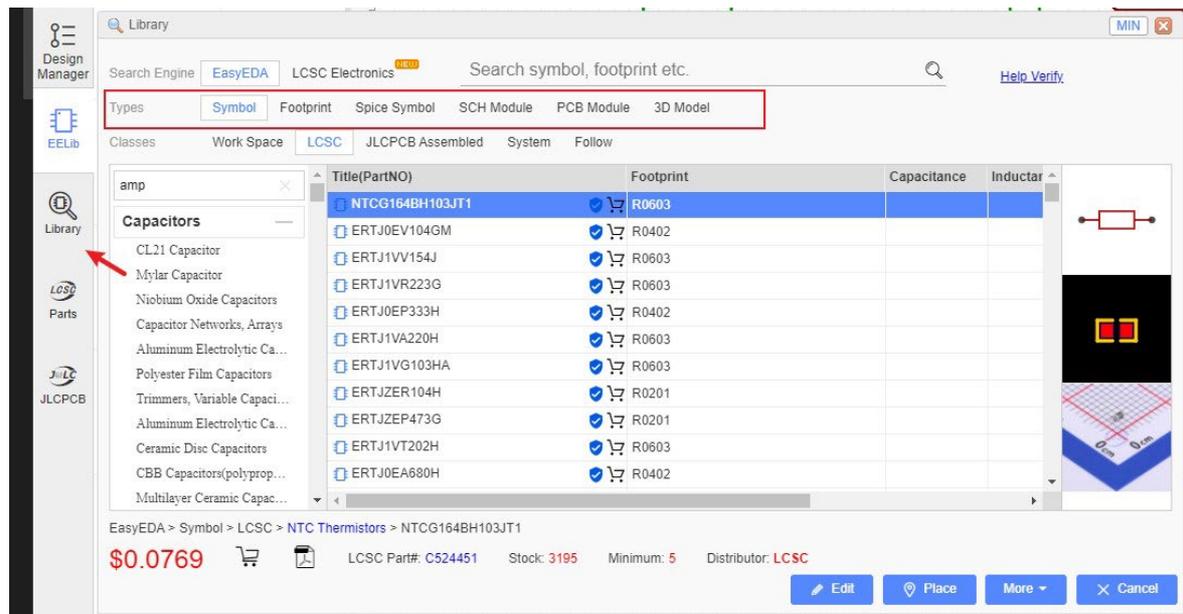
Many of these components have optional US and EU style symbols, we split them, so you can select those you like. Click on the drop down list or right click to popup the context menu, it contains many footprints or parameters. EasyEDA will remember your choices for the next time.

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.



## 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](https://www.lcsc.com) parts(Official Parts). It will add new libraries everyday
- LCSC Assembled: JLCPCB Assembled parts. All JLCPCB assembly parts will contain a SMT icon, that means this part can be JLCPCB 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](#)

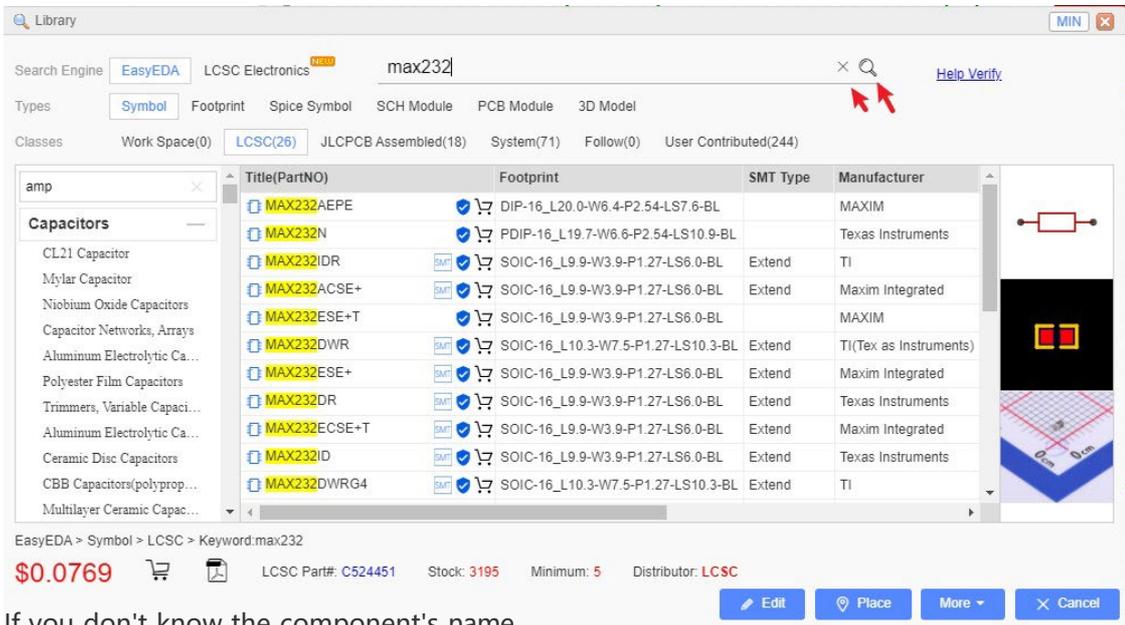
We add an "JLCPCB Assembled" Components option of the Parts, It's easy to choose which component can be assembled by JLCPCB. Yes, JLCPCB will provide the assembly service. the more information please refer at: [How to order a SMT order](#)

## Search Engine - EasyEDA

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

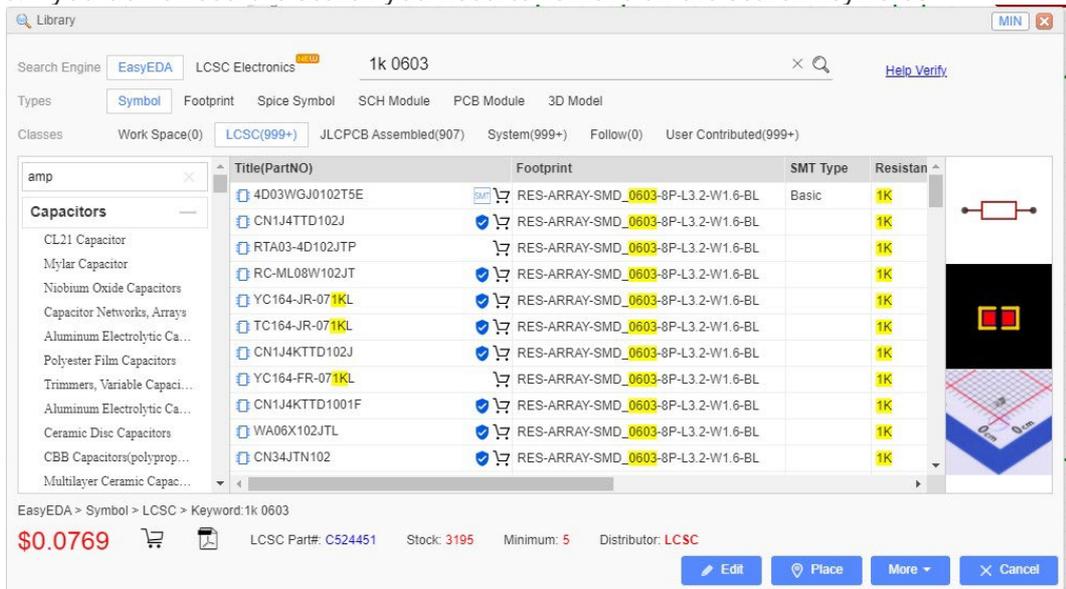
and then click the "Table of contents" to open the categories list to choose your components. From there you can scroll up and down to browse parts from each category.

- 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:



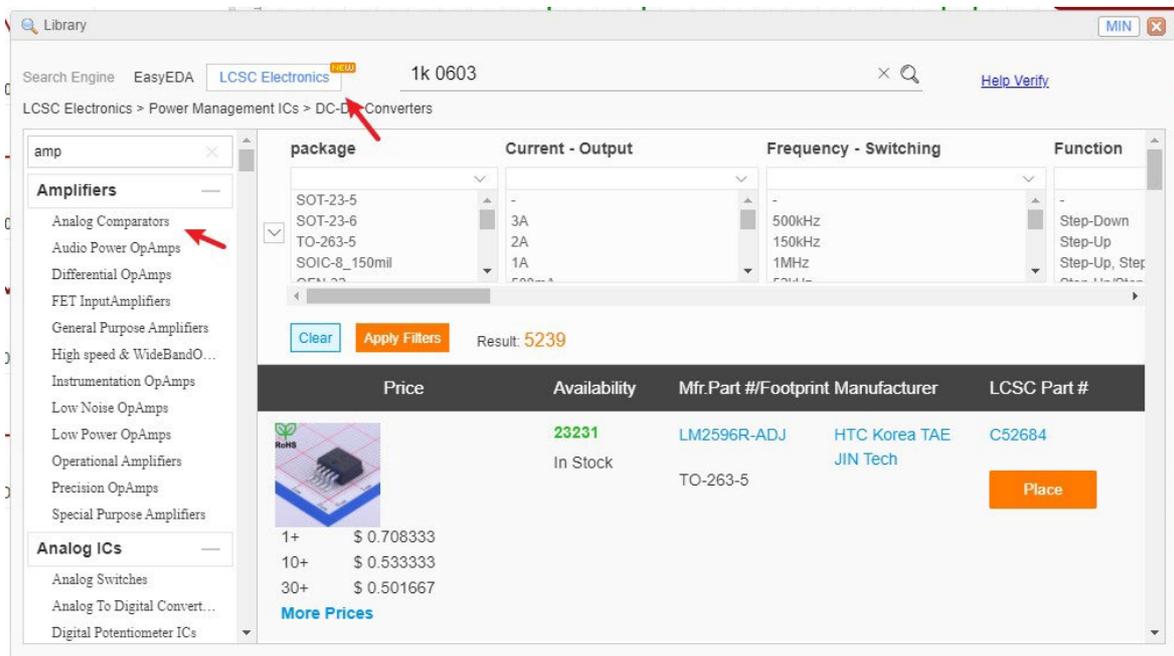
- If you don't know the component's name  
For example, you want to find a resistor which value is 1kohm, footprint is 0603, at Libraries you can follow below steps:

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

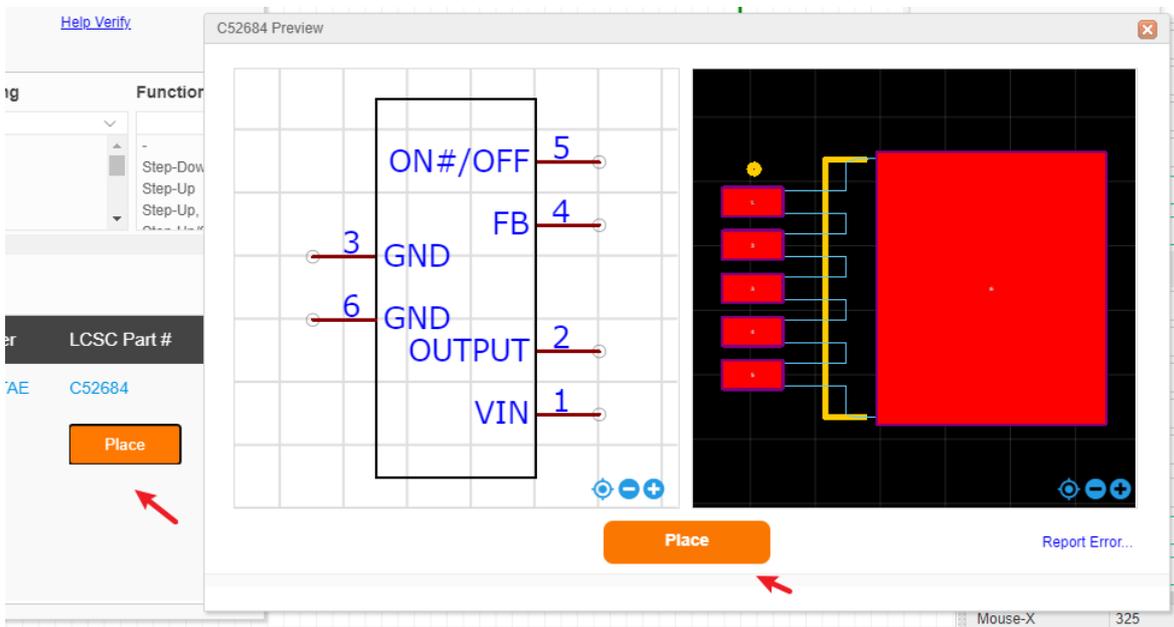


## Search engine - LCSC Electronics

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

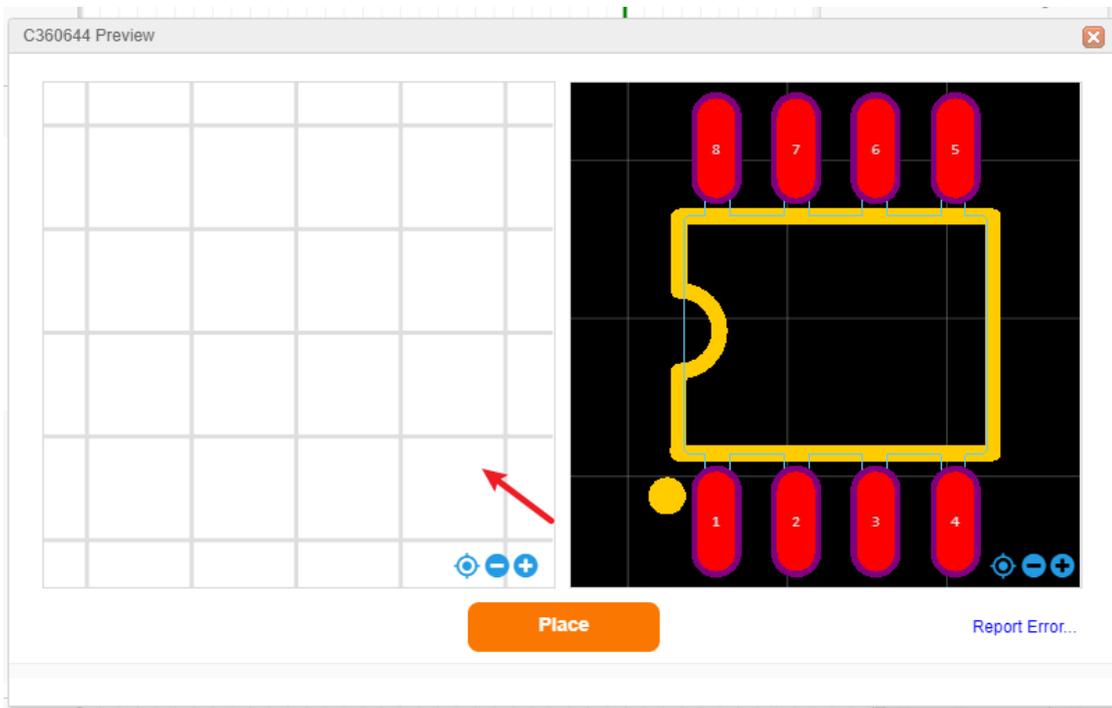


When you find out part, and you can place into the schematic:



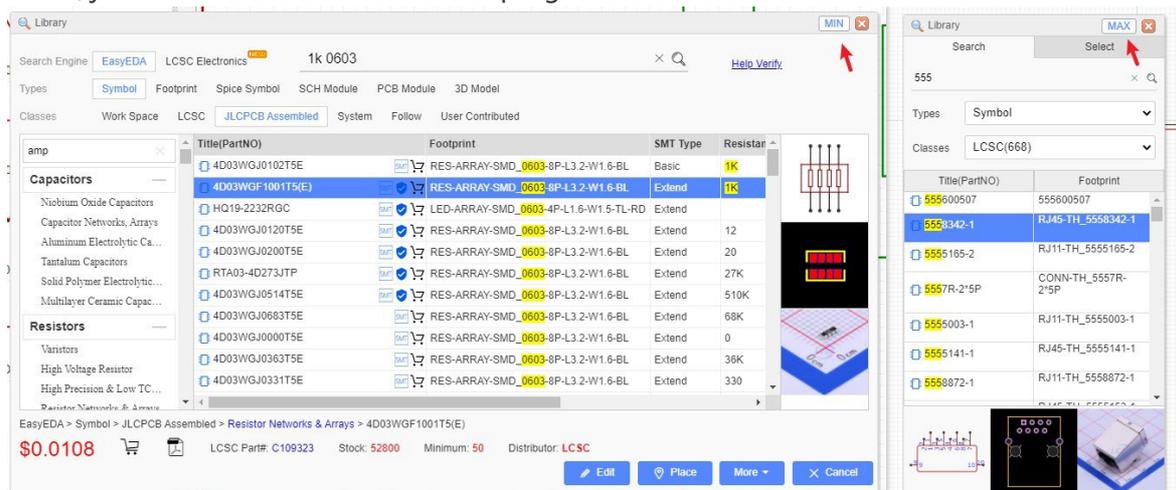
Notice:

- 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 know. 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.

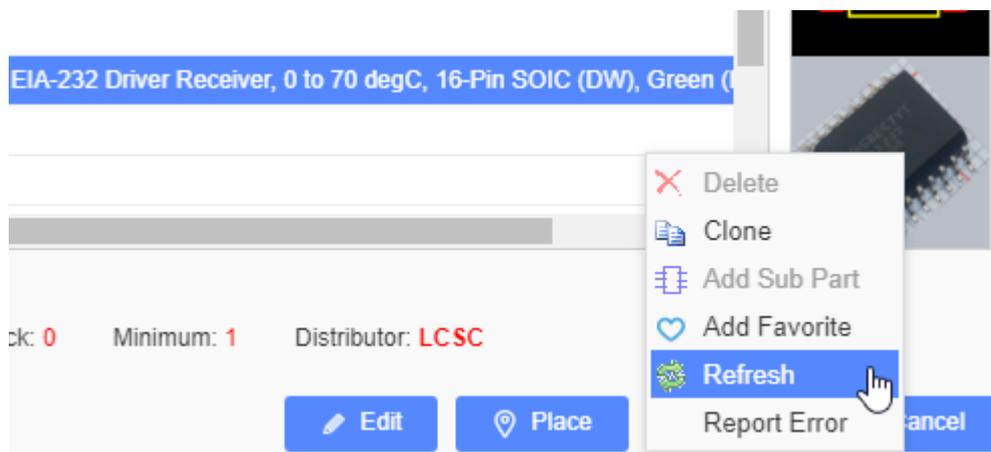
### More:

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.

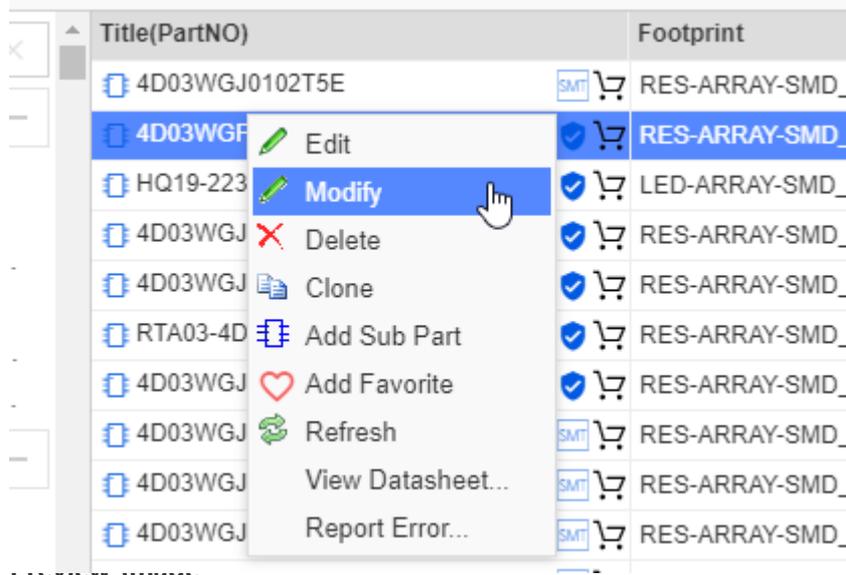
Components with sub parts (multi-device footprints).

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.

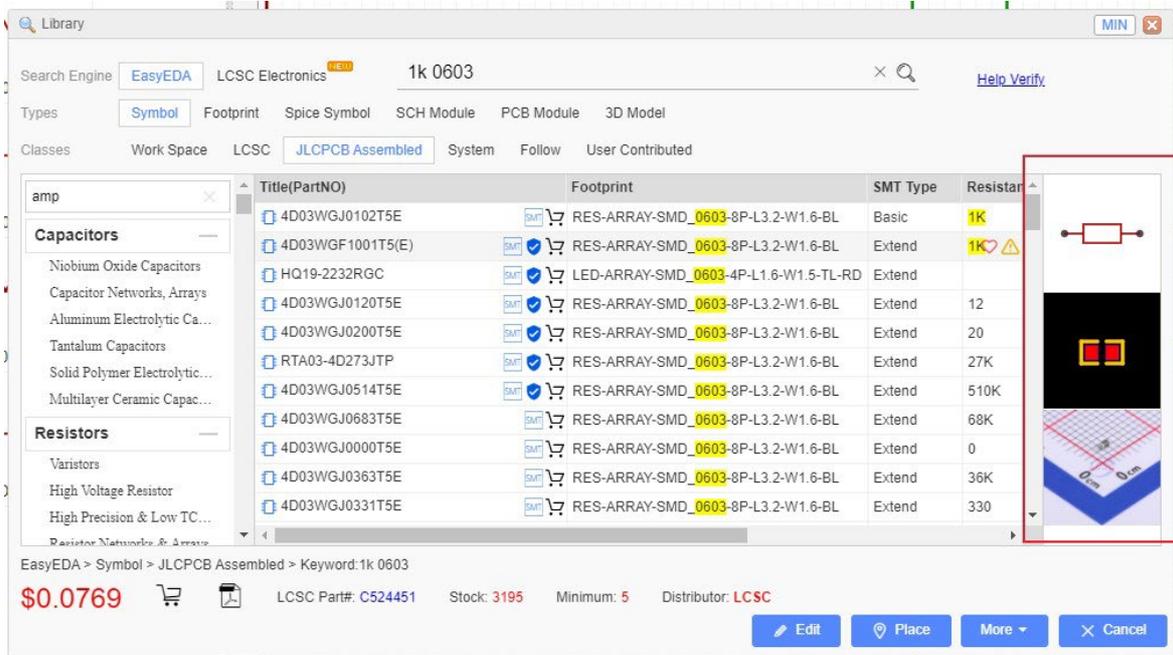


### Right-Click

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

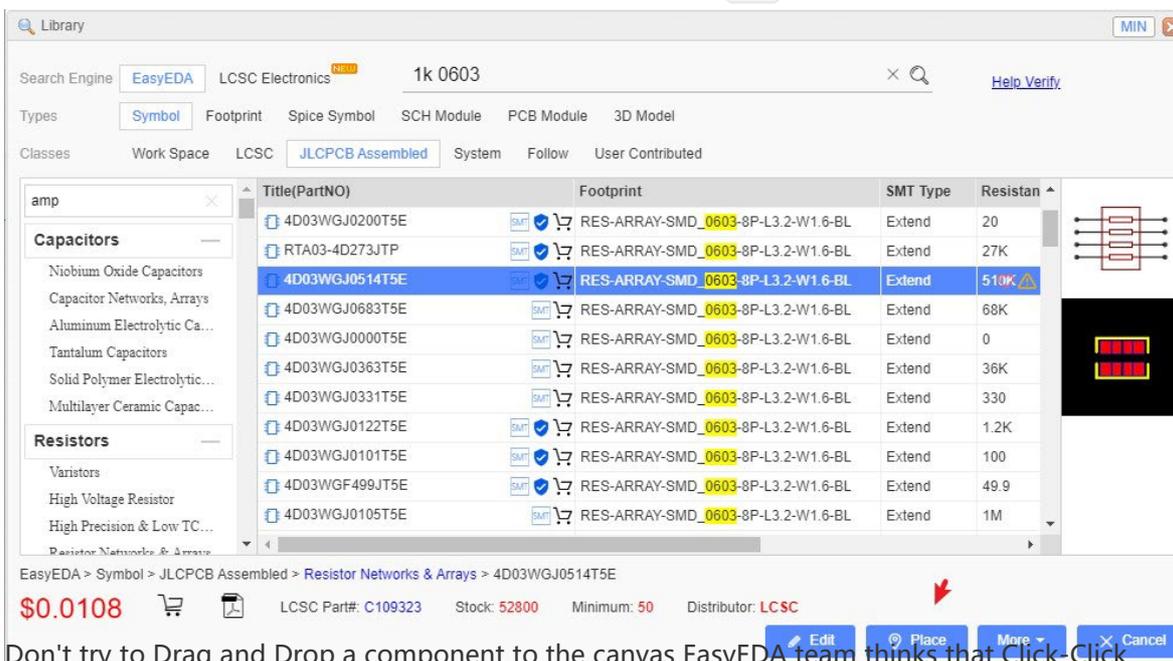


Every library when you click, you can check its preview image, such as symbol, footprint, production picture. Click the the image you can open it quickly.



## 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`.



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.

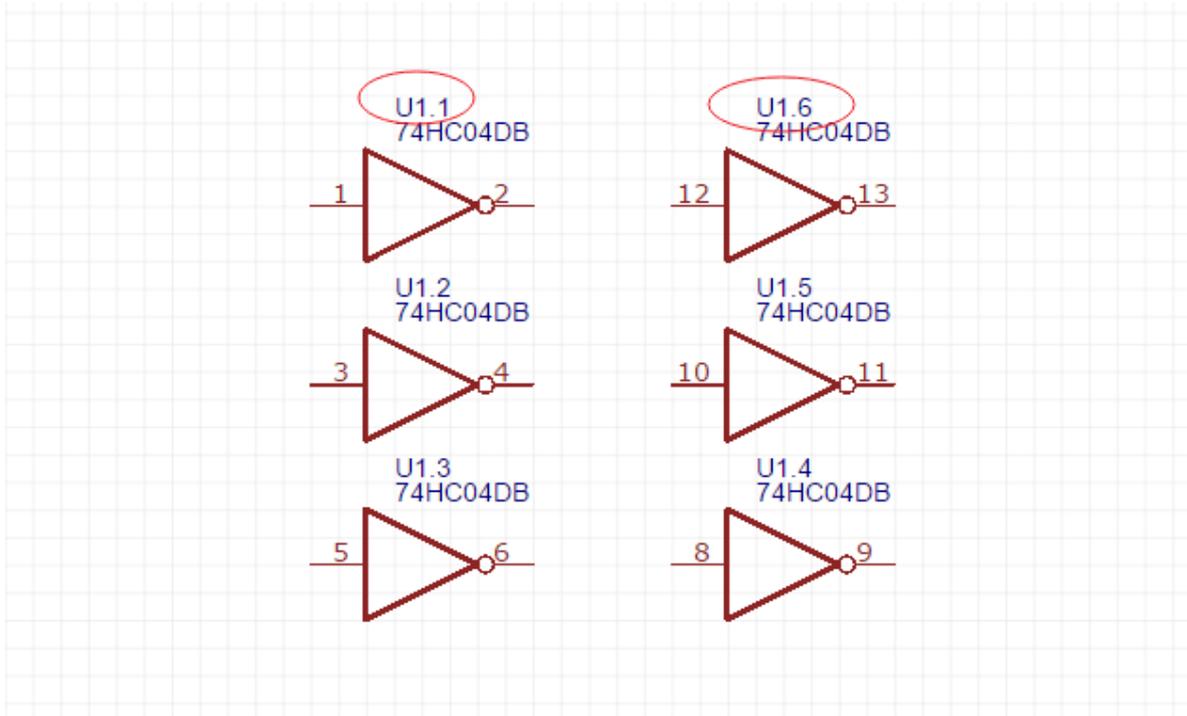
## Multi-part Components

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, it calls multi-parts or subparts.

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.



If you click the father-part and place on the schematic, the remaining subparts will be placing one by one, if you click the one of the subpart, you will placing U1.1, U2.1, U3.1 etc.

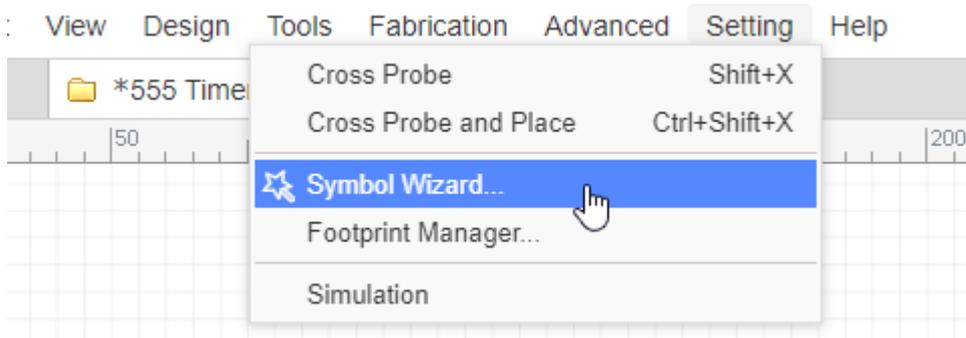
How to create multi-part(subpart) please refer [Create Symbol](#)

## 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.



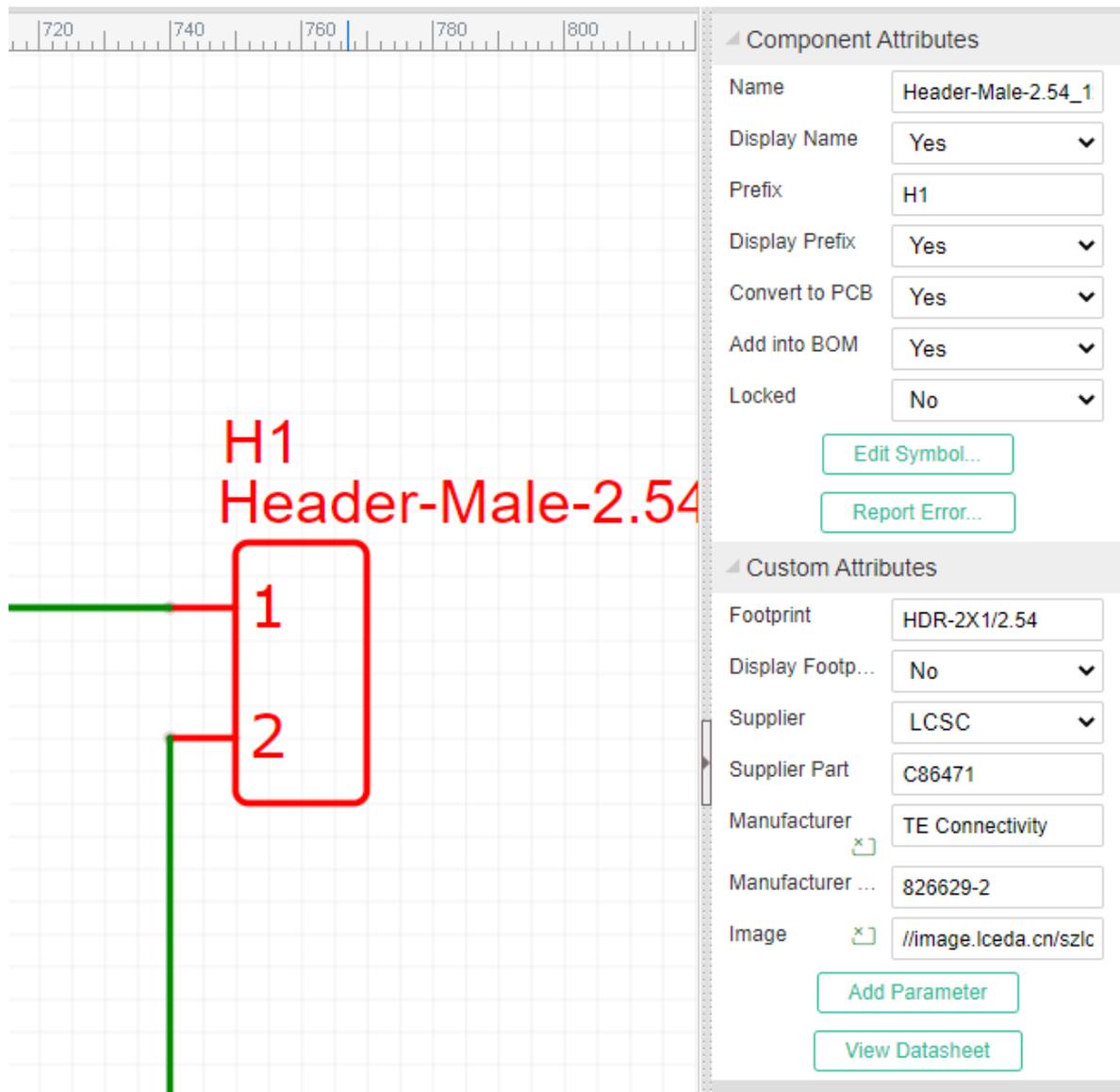
The professional function please refer at [Schematic Symbol Wizard](#)

# Experiment no: 10

## Experiment Title: Component Management in EasyEDA

### Component Attributes

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

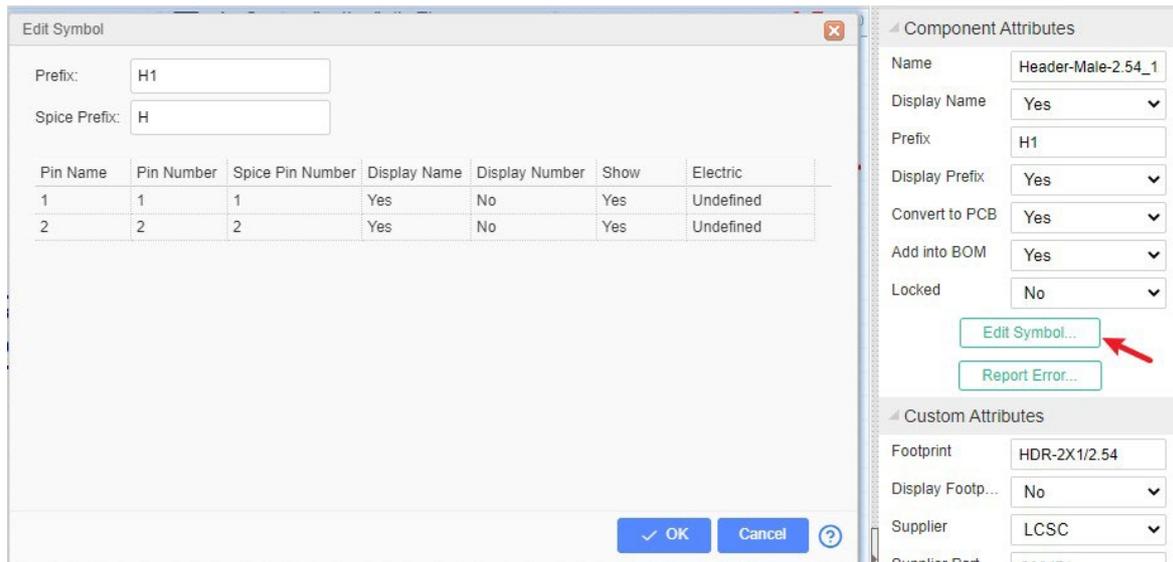


The screenshot displays the EasyEDA interface. On the left, a component symbol is shown on a grid. The symbol is a red rectangle with two pins, labeled '1' and '2'. Above the symbol, the text 'H1' and 'Header-Male-2.54' is displayed in red. The right-hand side of the image shows the 'Component Attributes' panel. The panel is divided into two sections: 'Component Attributes' and 'Custom Attributes'. The 'Component Attributes' section includes fields for Name, Display Name, Prefix, Display Prefix, Convert to PCB, Add into BOM, and Locked, each with a dropdown menu. Below these fields are two buttons: 'Edit Symbol...' and 'Report Error...'. The 'Custom Attributes' section includes fields for Footprint, Display Footp..., Supplier, Supplier Part, Manufacturer, Manufacturer..., and Image, each with a dropdown menu or a text input field. Below these fields are two buttons: 'Add Parameter' and 'View Datasheet'.

Attribute	Value
Name	Header-Male-2.54_1
Display Name	Yes
Prefix	H1
Display Prefix	Yes
Convert to PCB	Yes
Add into BOM	Yes
Locked	No
Footprint	HDR-2X1/2.54
Display Footp...	No
Supplier	LCSC
Supplier Part	C86471
Manufacturer	TE Connectivity
Manufacturer ...	826629-2
Image	//image.lceda.cn/szlc

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**.



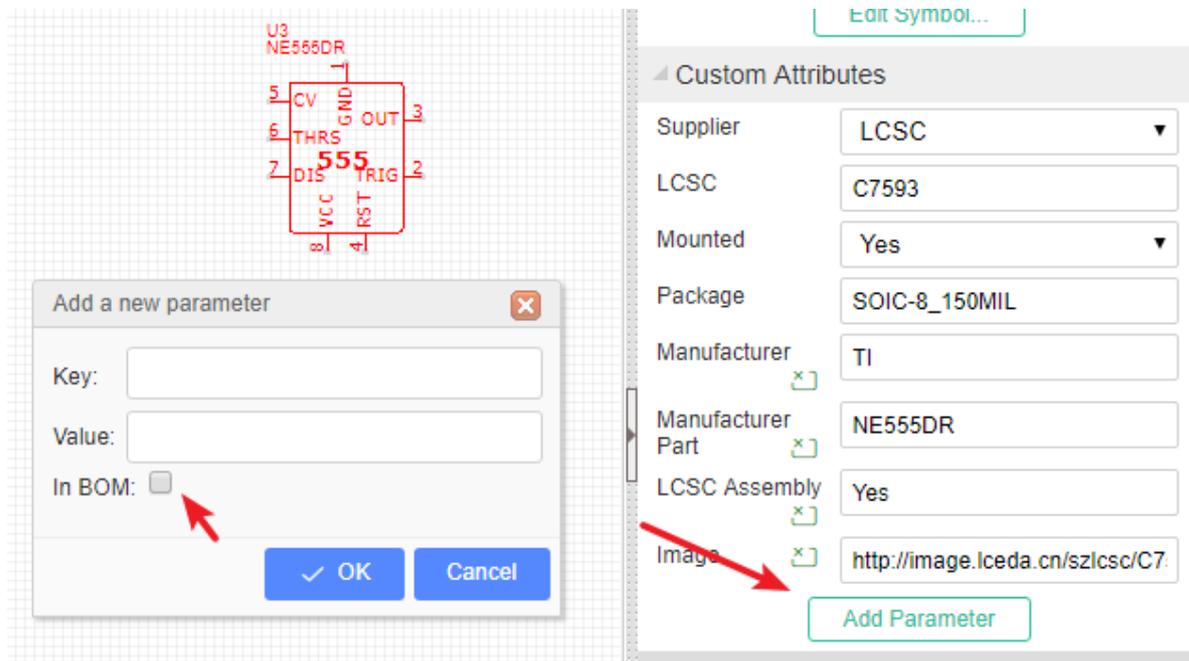
If the component's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

## 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.

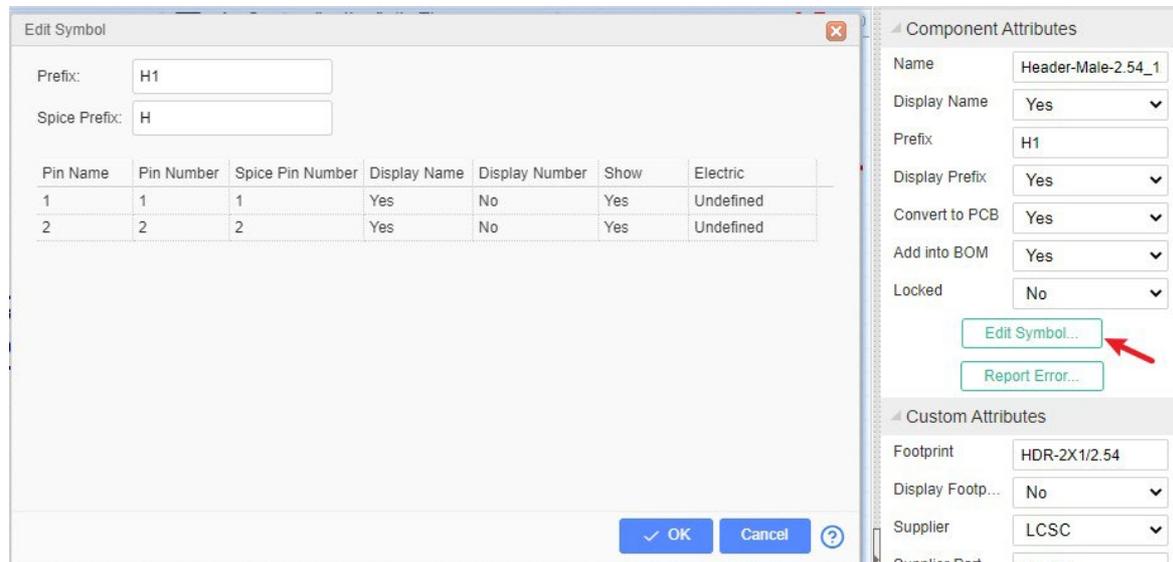


## Modify Symbol Pinmap Information

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

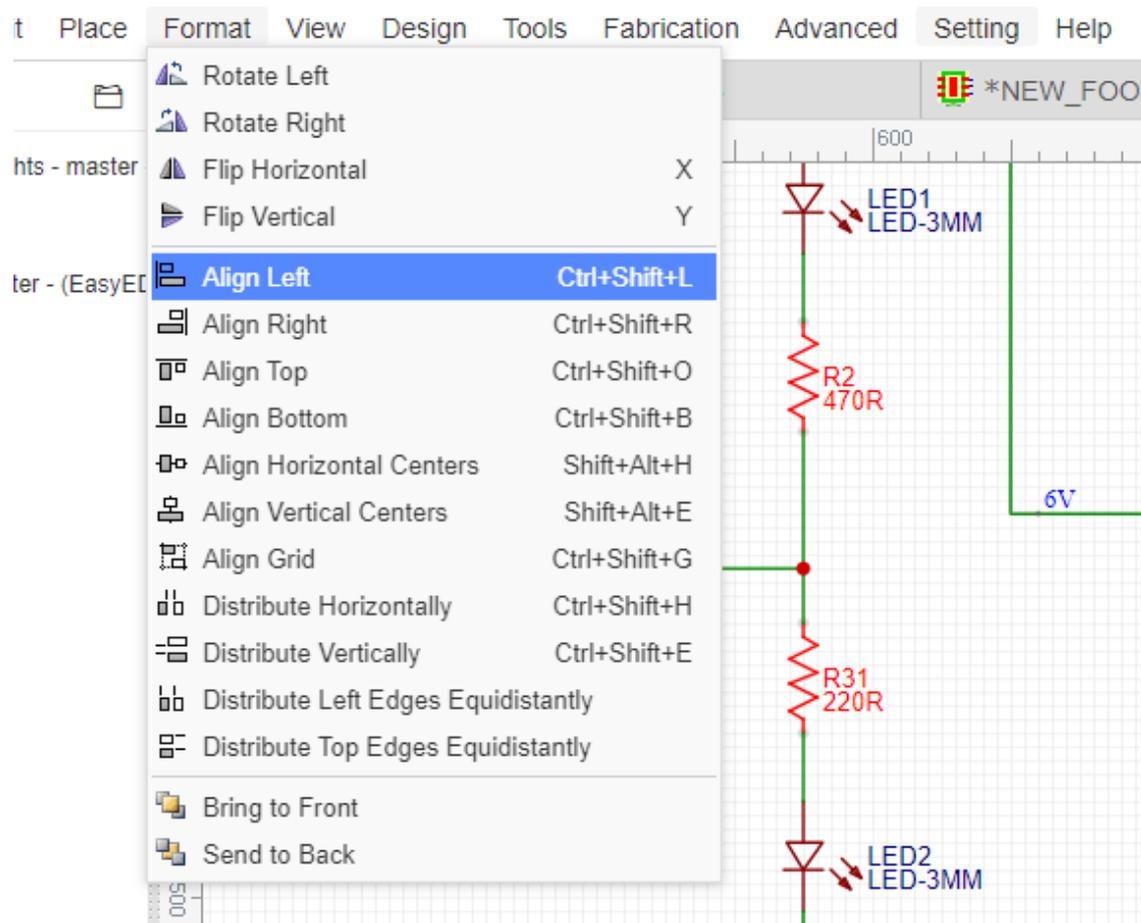
---

## 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

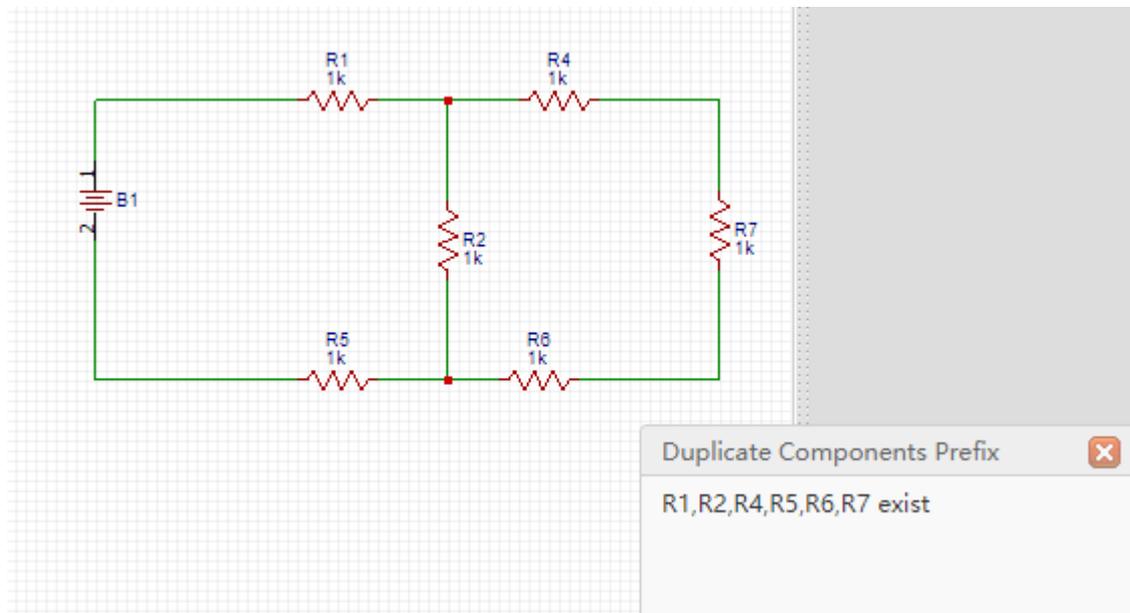
---

### 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.

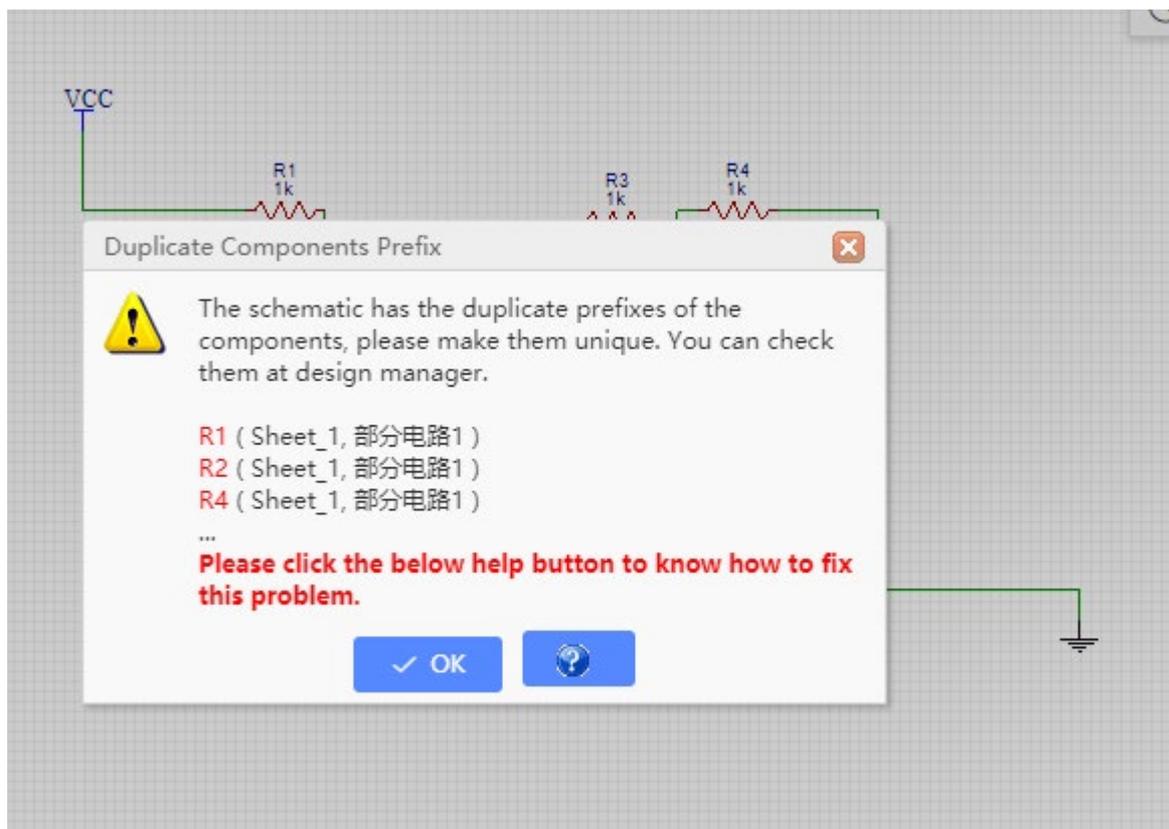
### Prefix Conflict Error

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.



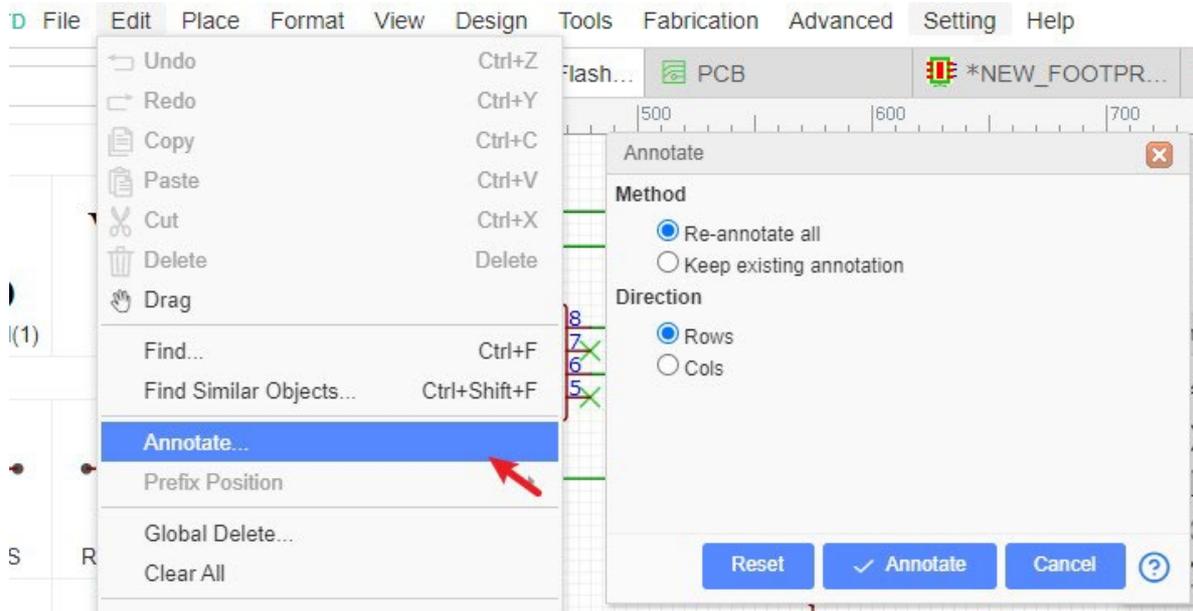
In the above image, you can find the two identical copies of the same schematic, which when you Convert Project to PCB, EasyEDA will try to merge into a single schematic, so every item will have 2 copies.

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.

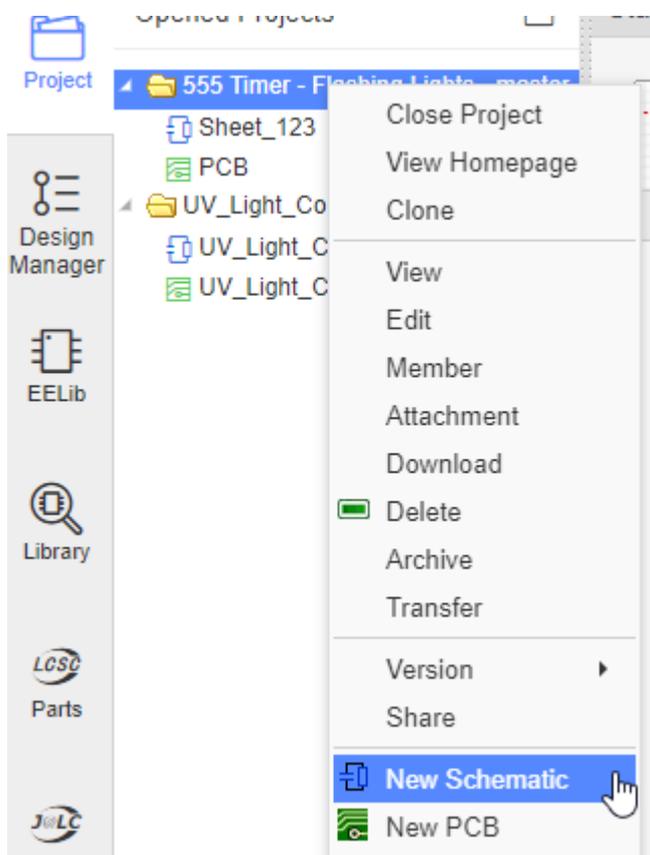
---

## Multi-Sheet

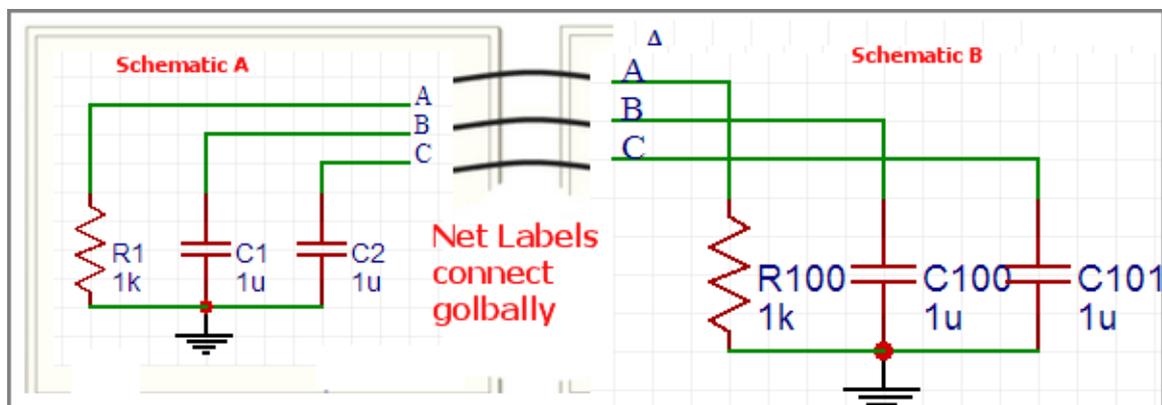
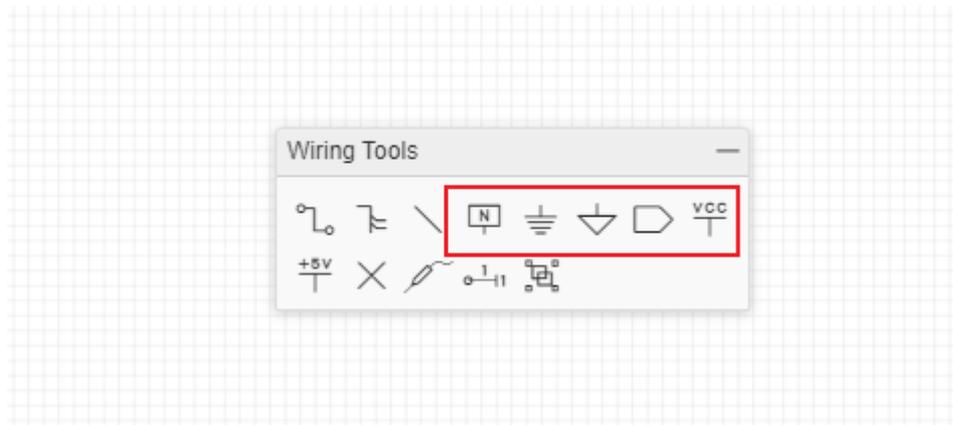
---

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

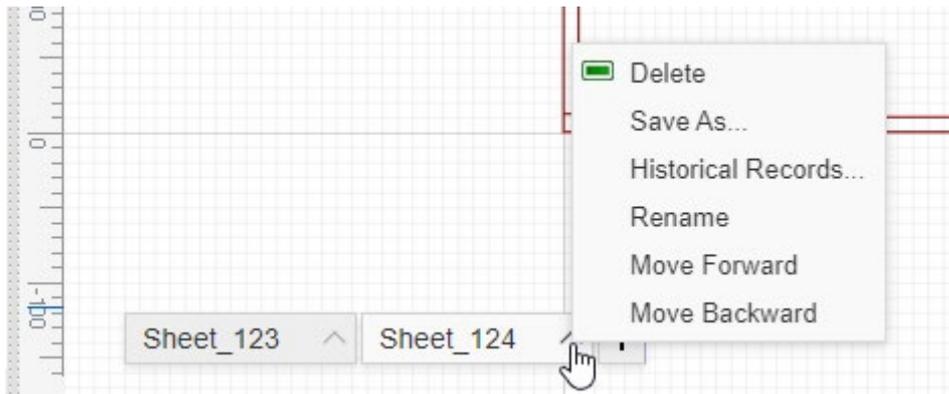
You can put several schematics in one project with connections between made by NetLabels/netPorts. All nets in EasyEDA are global so if you create a netlabel `DATA0` in sheet A and then create a netlabel `DATA0` in sheet B, when sheet A and sheet B are in the same project, they will be connected.



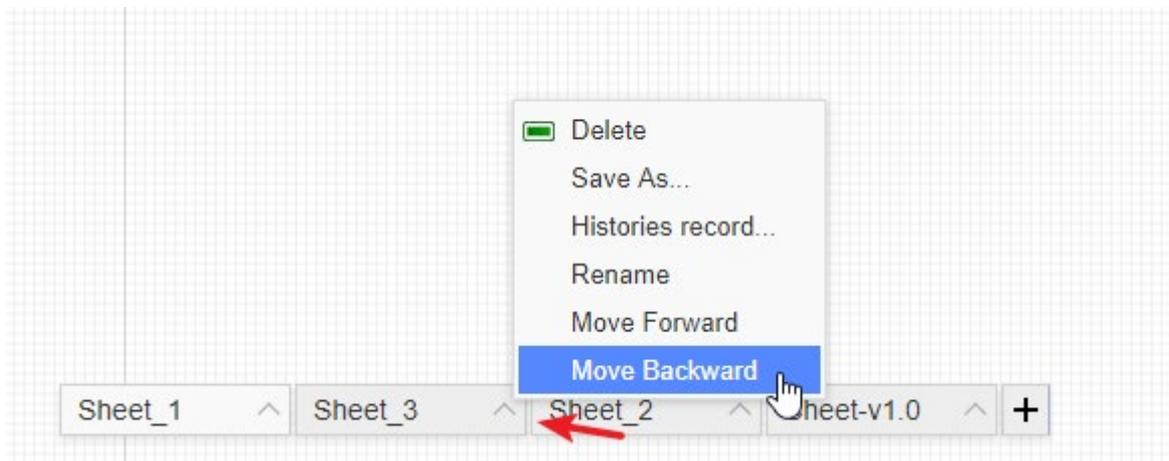
**Multi-sheet designs**(equivalent to a circuit spread over several pieces of paper), all schematics under the same project will be merged into one when be converted to PCB connecting in **Netlabel**, **Netflag**, **Netport**.



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

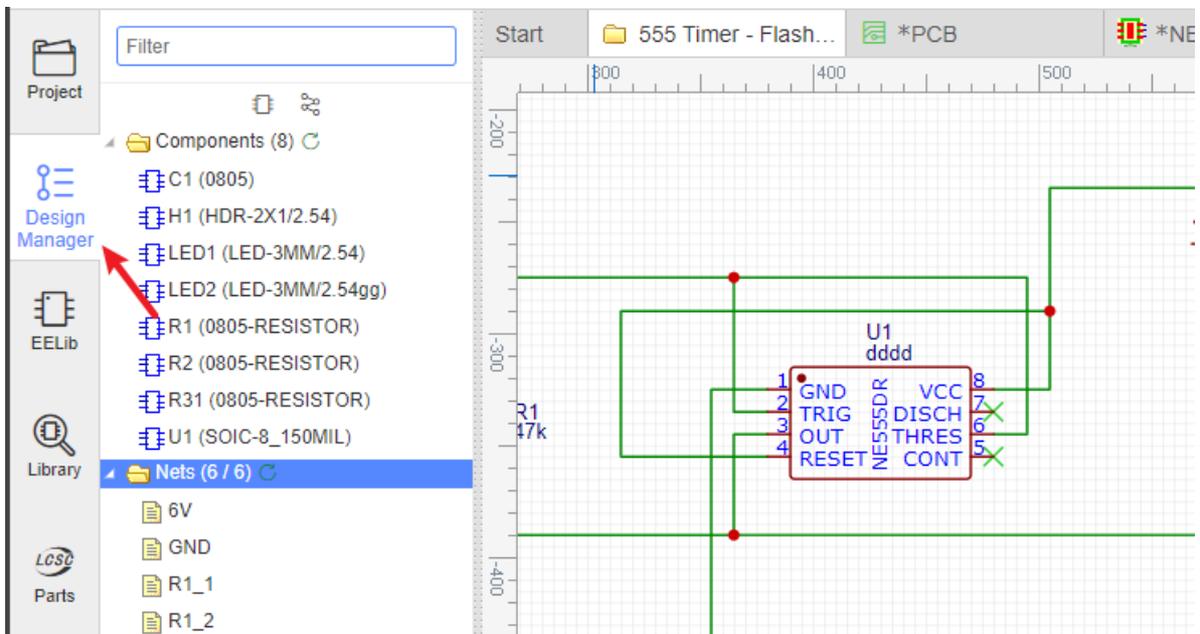
---

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.

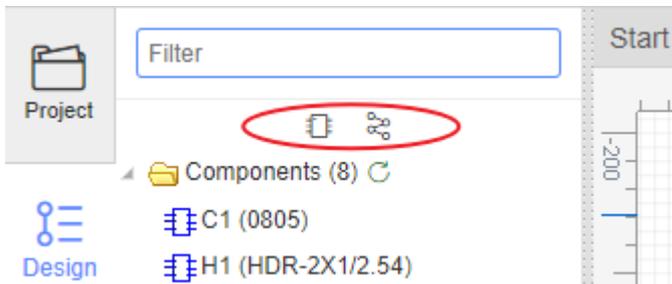
**Design Manager** is just the tool.

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

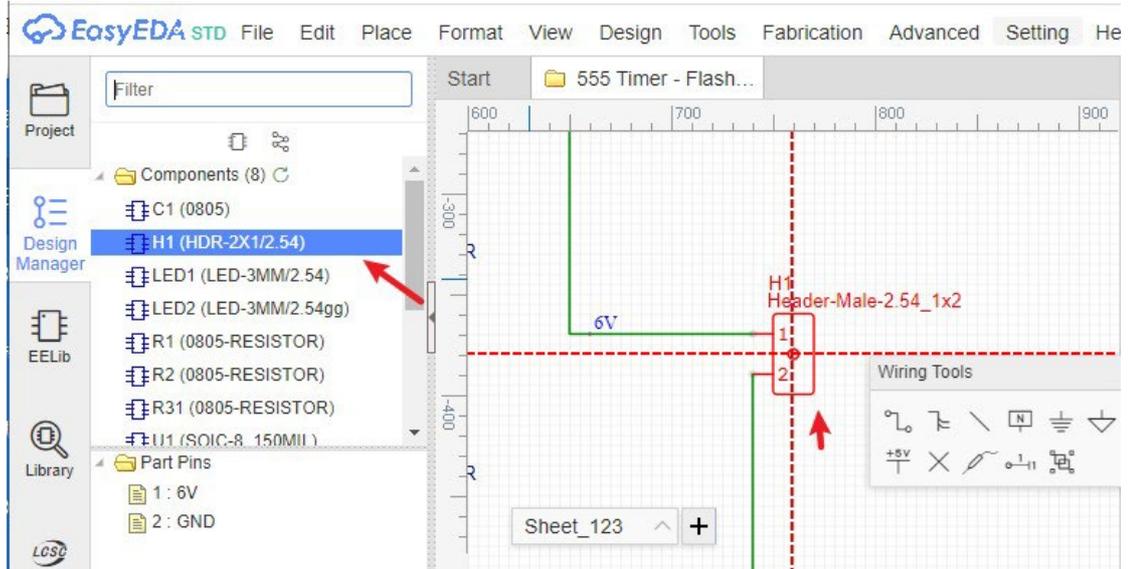
or click it via on the left navigation panel:



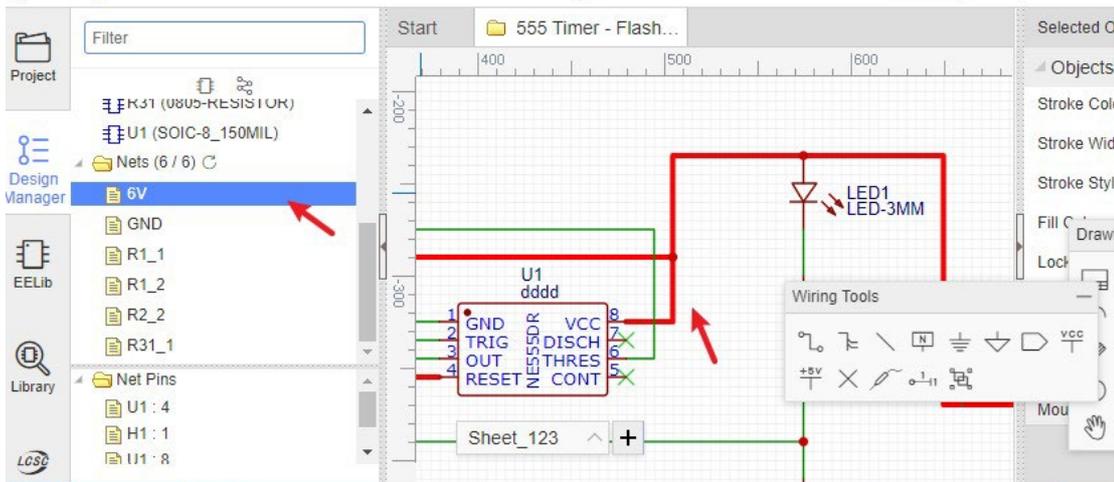
You can click the jump icon to goto the folder quickly.



- **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.

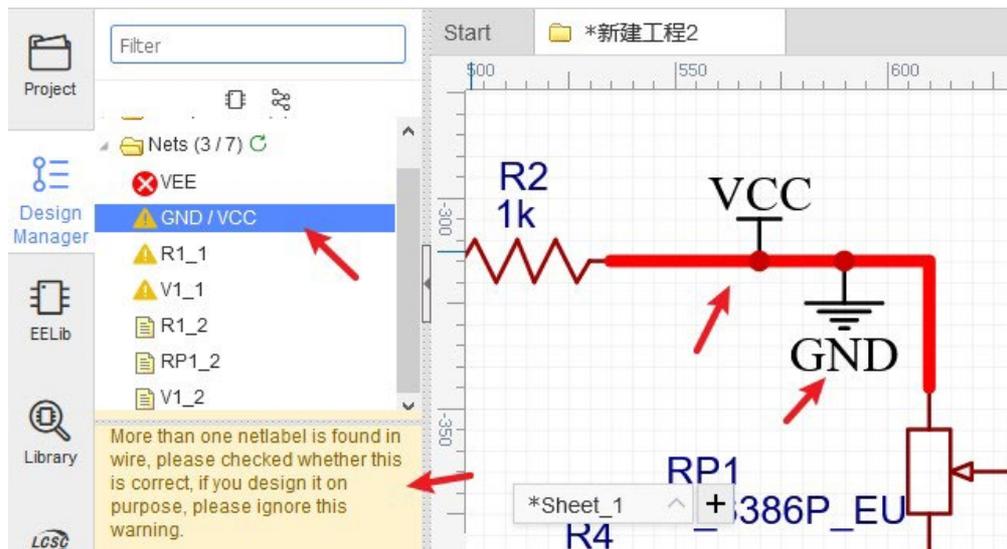


- **Nets:** Lists all the nets in this schematic. A net must connect at least two Pins, or the net name will be marked as a red error. When click the net name, the canvas wire will highlight and being large, when you click the empty space to unhighlight:



When you click the net name, you will see the tip at the bottom-left corner.

- Net warning: It will show a prompt exclamation point icon.
  - When multiple netlabels on one wire, please check whether if it is correct or just connected by mistake. You need to click this net and find it out. If your netflag or netlabel only connect one pin, it will show warning.

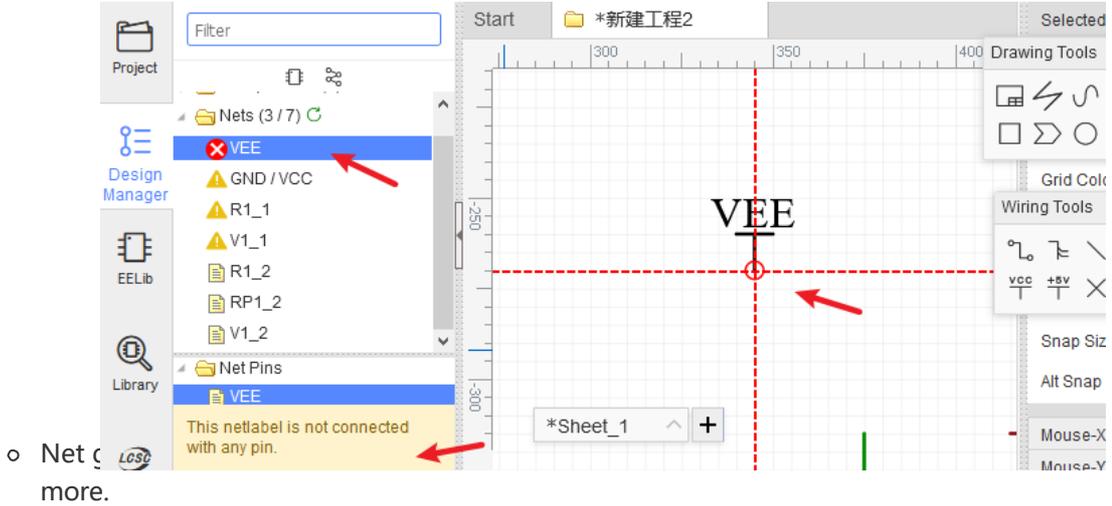


- The part's pin doesn't place the netlabel, or doesn't connect other pins, or doesn't place No Connect Flag. A completed net must connects two and more pins, so that, you need to modify your net connection. If you don't need to use this pin, please place a No Connect Flag on the pin.

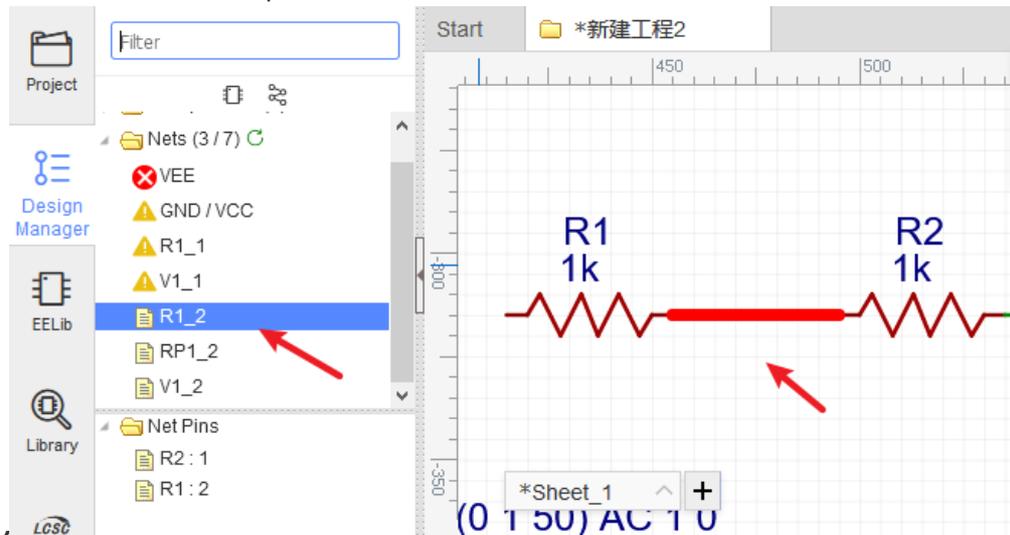


- Net error: Prompts a red error icon.

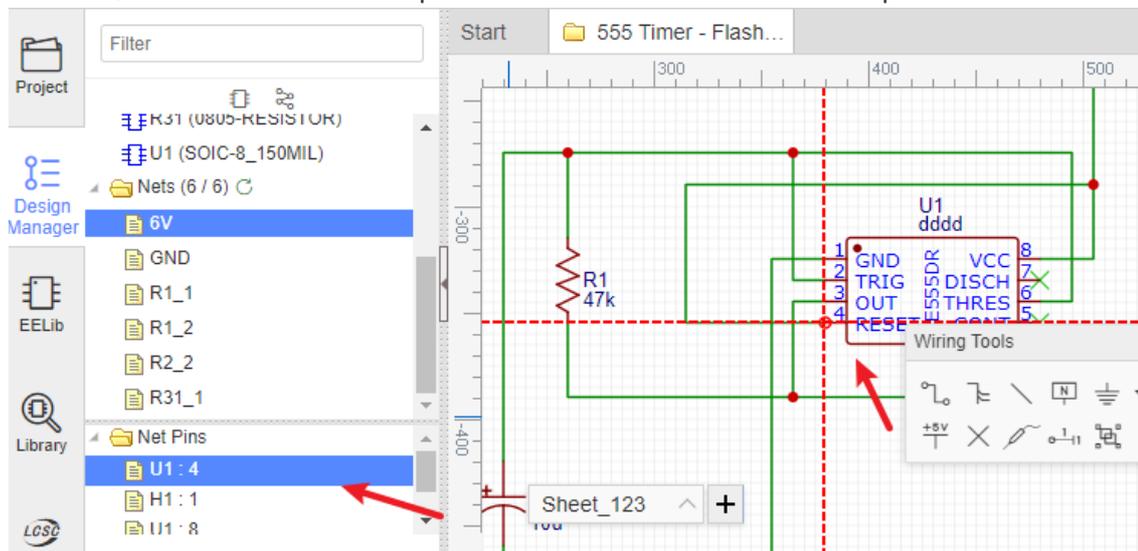
- When Netlabel haven't connected any pins.



- When the net is completed.



- Net Pins/Parts Pins. Lists all the pins of the selected net name or components.



# Experiment no: 11

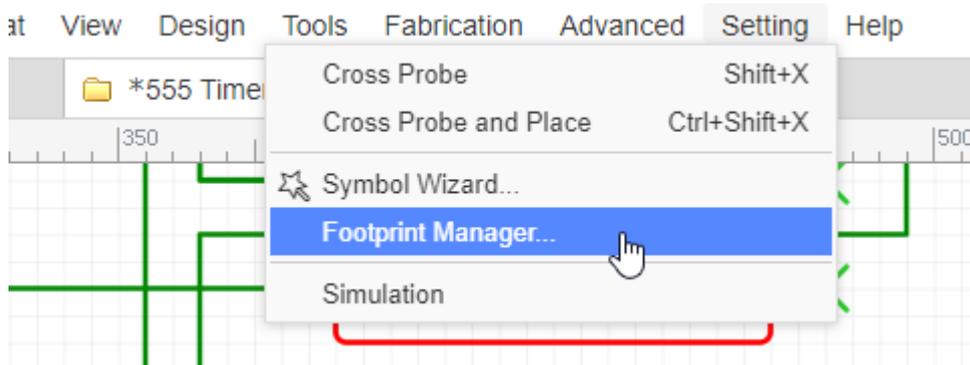
## Experiment Title: Footprint Management in EasyEDA

### Introduction

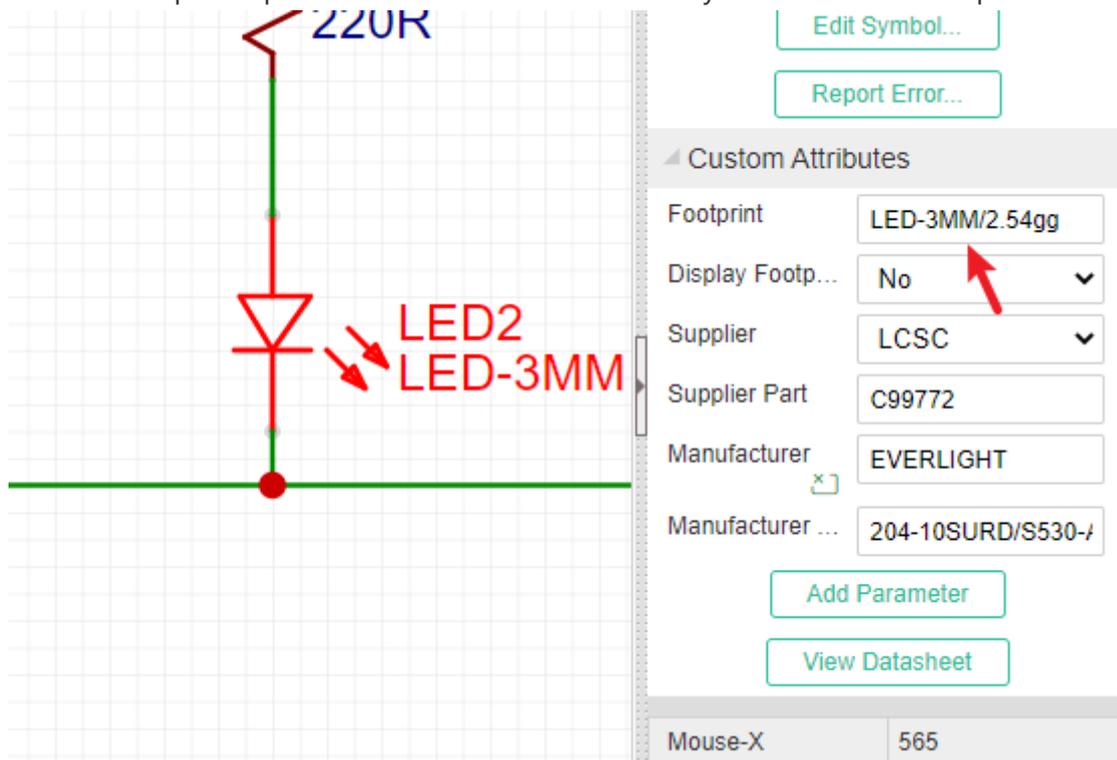
Want to batch modify components? Can't identify the corresponding relationship between component pins and footprint pins? Don't worry, EasyEDA can do this.

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.

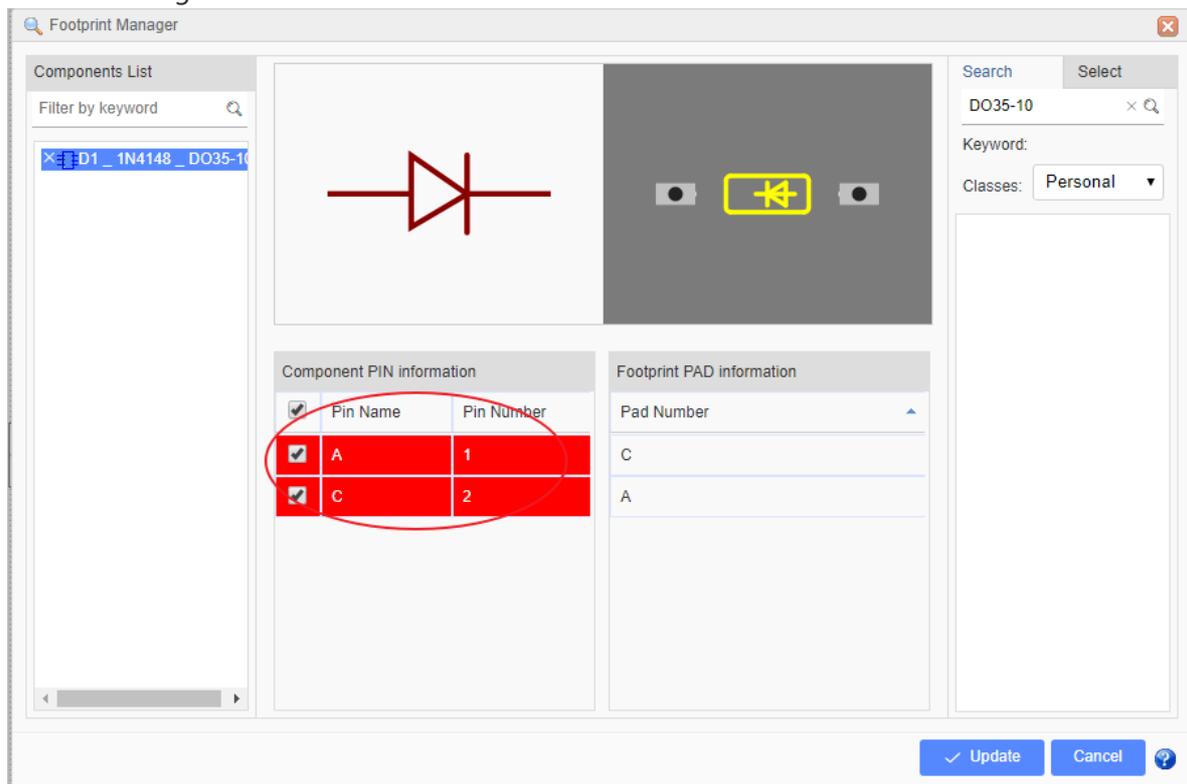
For example, If your part D1 has 2 pins,

- 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,

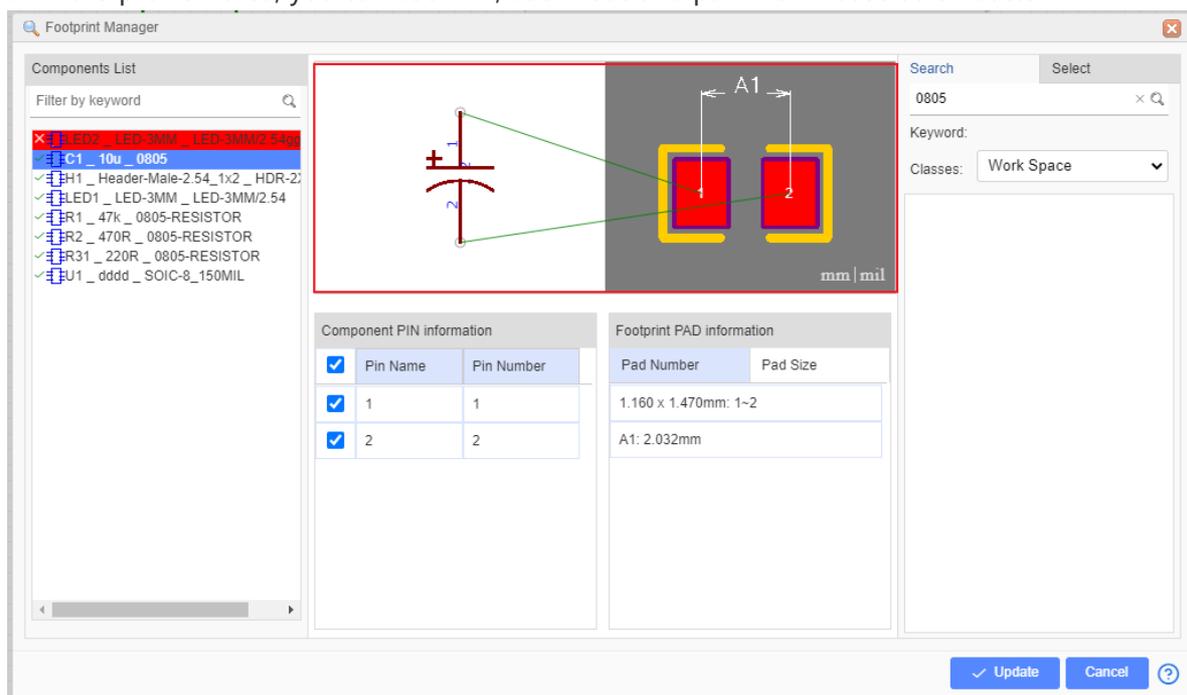
but the part's pin number doesn't match the pad number, so the the footprint manager will alert red background:



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

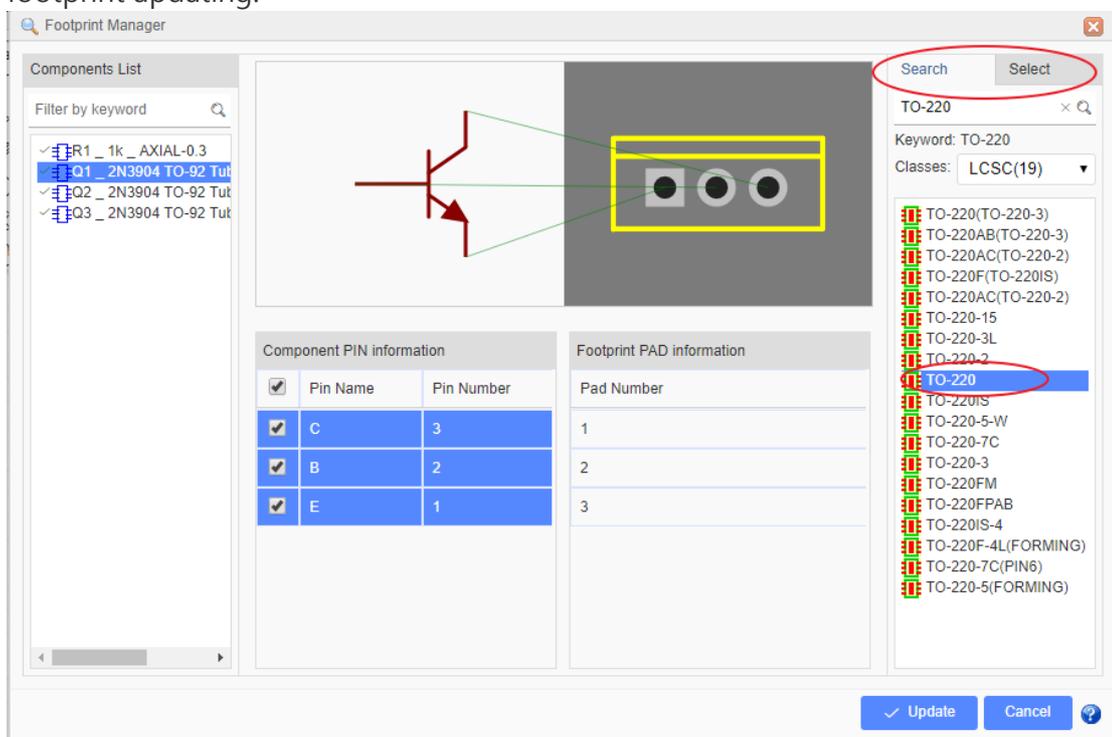
If you want to change the footprint, for example, select a component such as Q1, from **TO-92** TO **TO-220**, you just need to click in the footprint input box. EasyEDA will popup the footprint manager dialog. You can follow the instructions.

- ◆ 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.



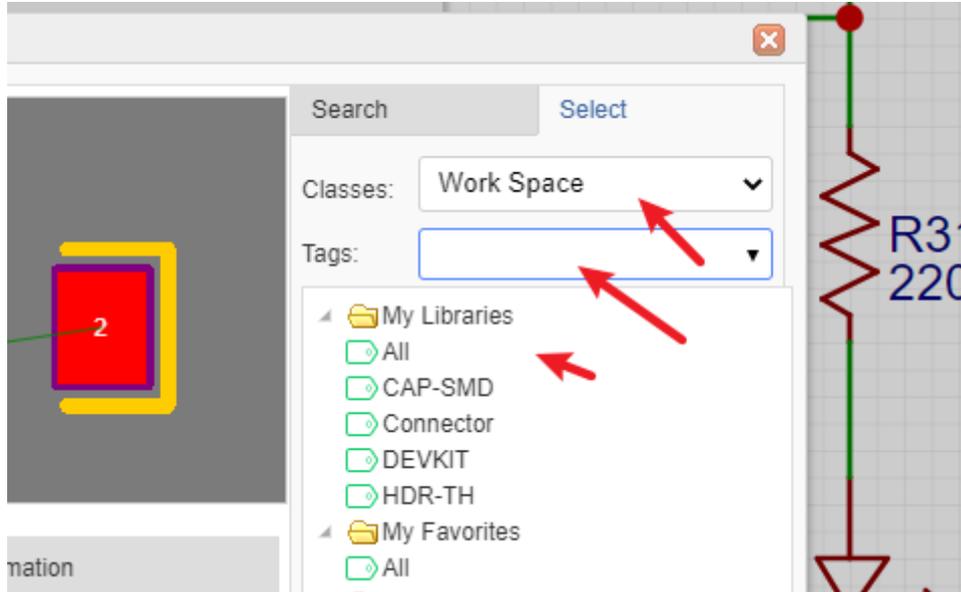
- ◆ When you select a subpart, the others subparts will be selected too, so they will update the footprint together.
- ◆ If the part's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

## 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.



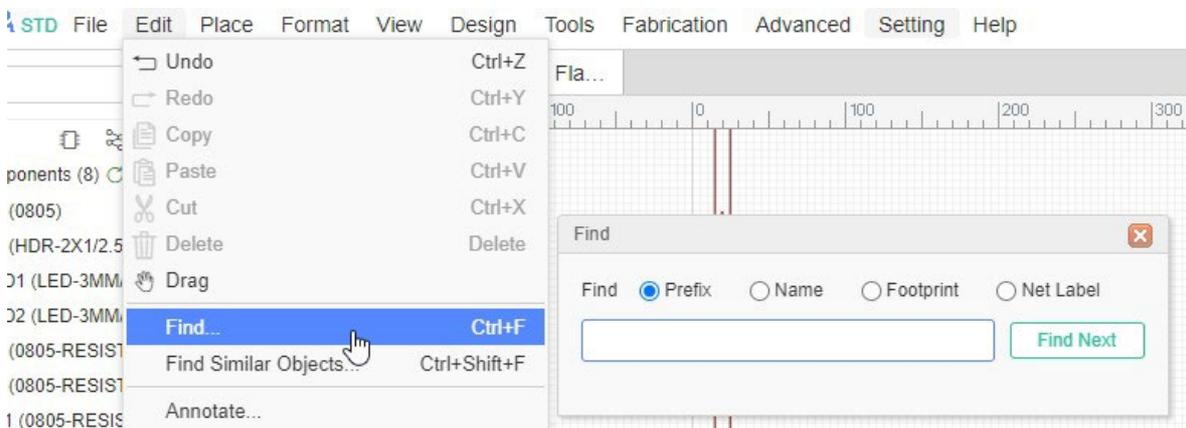
## Find Similar Objects

### 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.

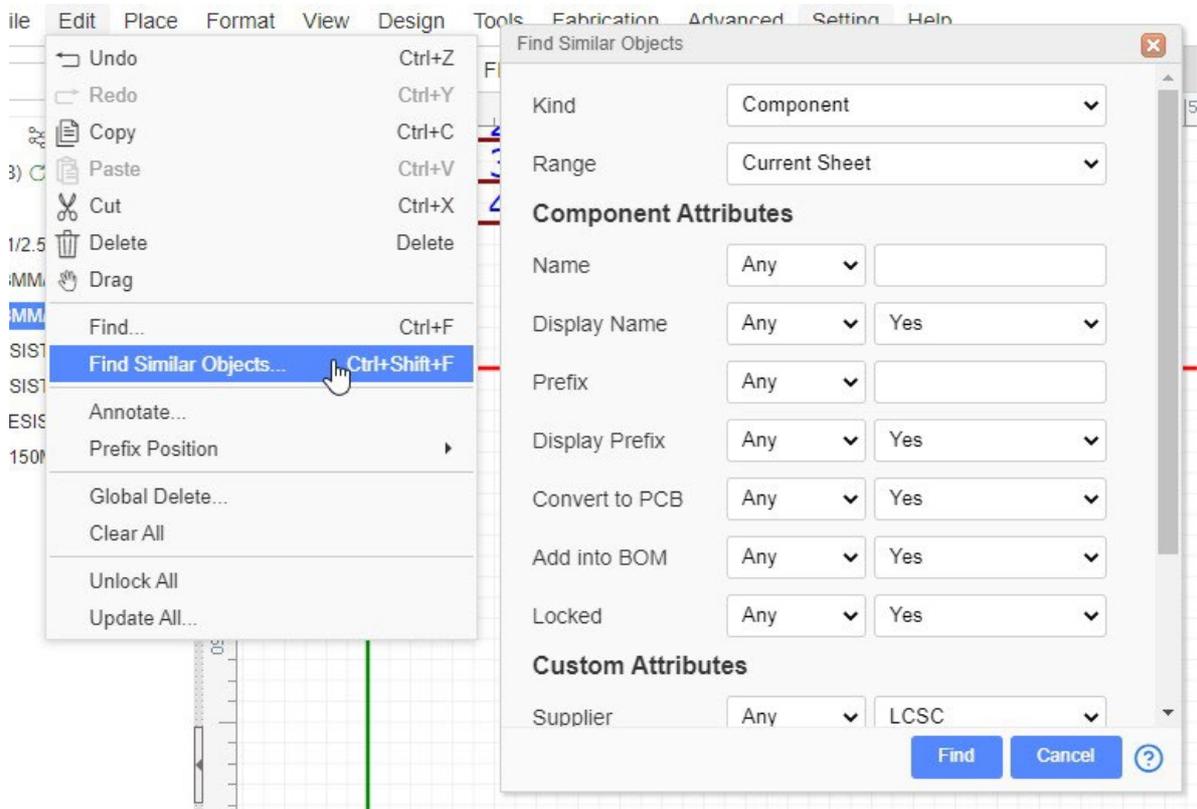
This feature will find, highlight and center in the window, parts by their Prefix (or reference designator). However, it cannot be used to find net names or other text in a schematic.

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

## Find Similar Objects

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

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



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

**Range:** This option 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 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":

The screenshot shows the 'Find Similar Objects' dialog box on the left and the 'Selected Objects' panel on the right. The dialog box has the following settings:

- Kind: Components
- Range: Current Sheet
- Part Attributes**
  - Name: Any
  - Name Display: Any, Yes
  - Prefix: Same, /R/
  - Prefix Display: Any, Yes
  - Locked: Any, Yes
- Custom Attributes**
  - Supplier: Any, LCSC
  - Supplier Part: Any
  - Mounted: Any, Yes

The 'Selected Objects' panel shows 7 objects. The 'Prefix' attribute in the 'Objects Attributes' section is highlighted with a red box. Below the dialog box, there are 'Find' and 'Cancel' buttons. The 'Selected Objects' panel also has 'Edit Symbol...' and 'Report Error...' buttons. At the bottom of the panel, there is an 'Add Parameter' button and a table with the following data:

Mouse-X	325
Mouse-Y	275

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 `<...>`, 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.

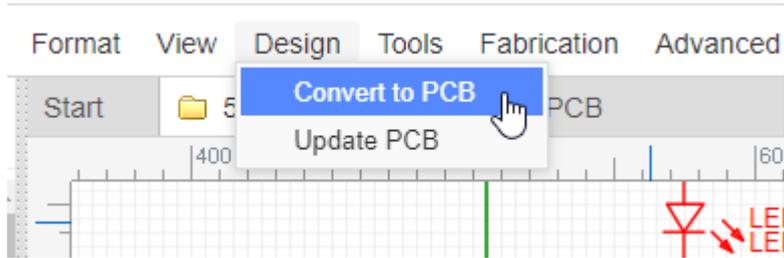
# Experiment no: 12

## Experiment Title: Convert Schematics to PCB in EasyEDA

---

### 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**.

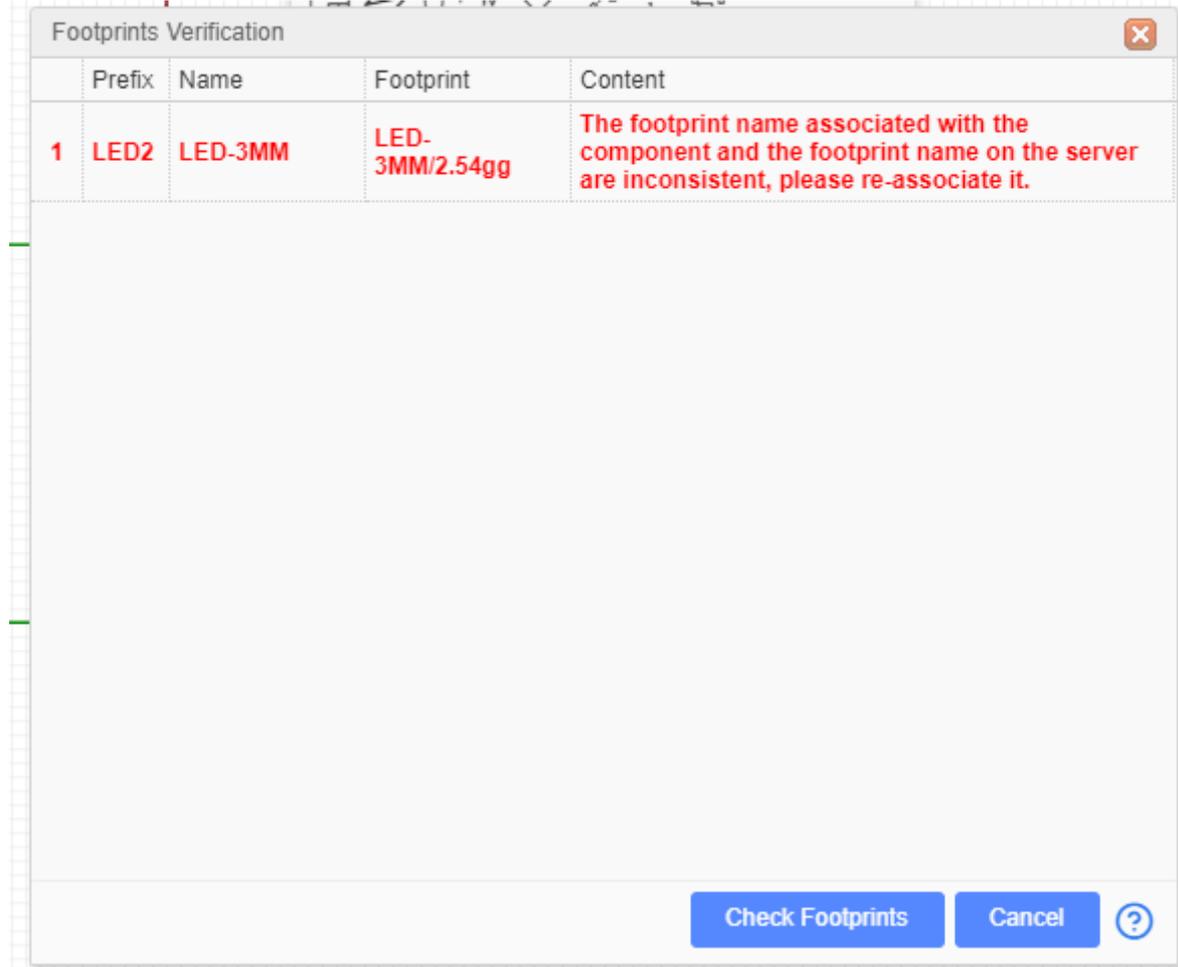


#### Note:

- 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:

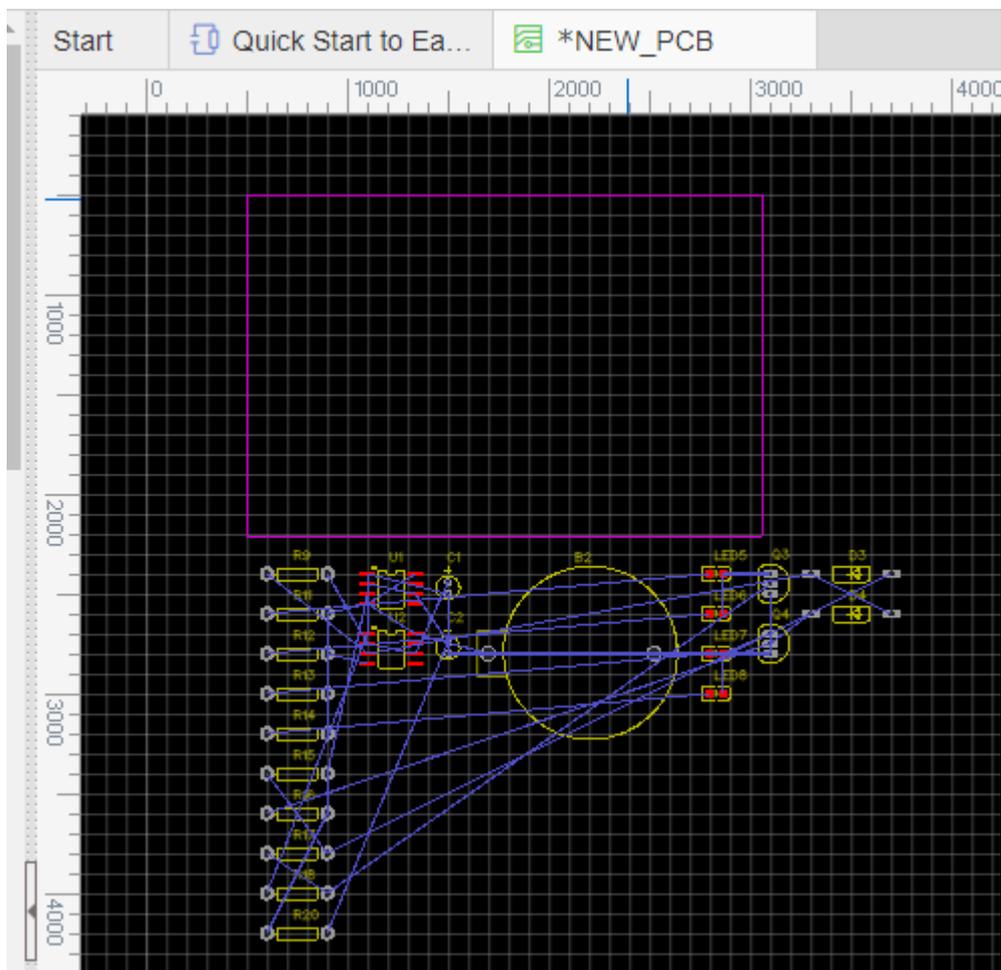


The row in red indicates that EasyEDA can't find a PCB footprint matching the footprint that the schematic symbol is calling for.

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.

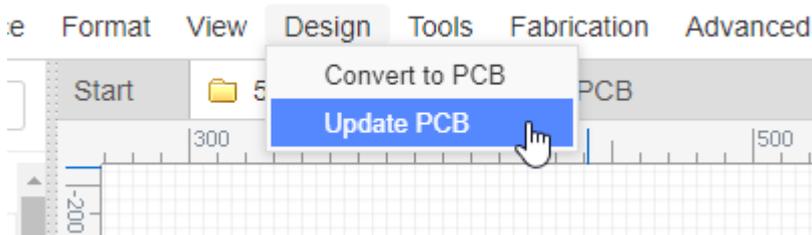
## Invalid footprint

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!

the 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.



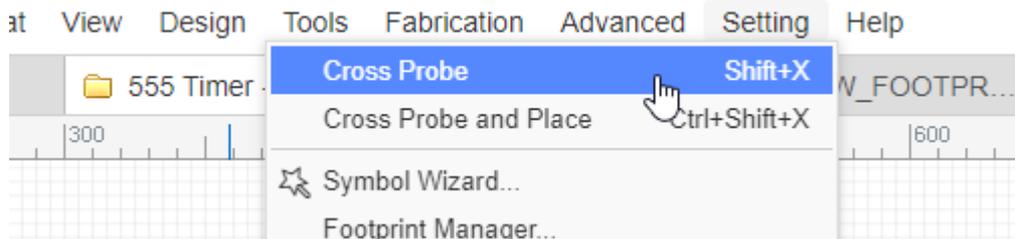
or you can use "Top Menu - Design - Import Changes" at PCB editor.

---

# Cross Probe

---

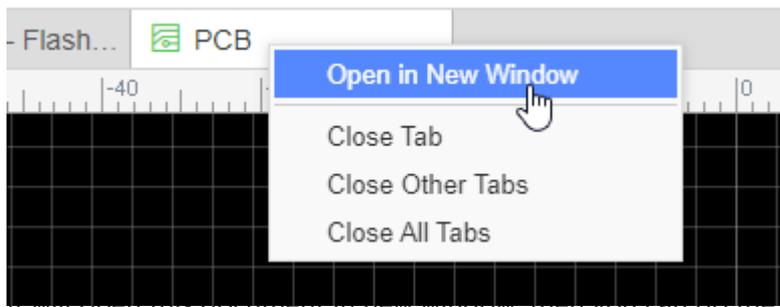
This tool is used to cross probe from chosen objects on the current schematic to its corresponding counterparts in the PCB, or from PCB Footprints to corresponding counterparts in the schematic.



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.

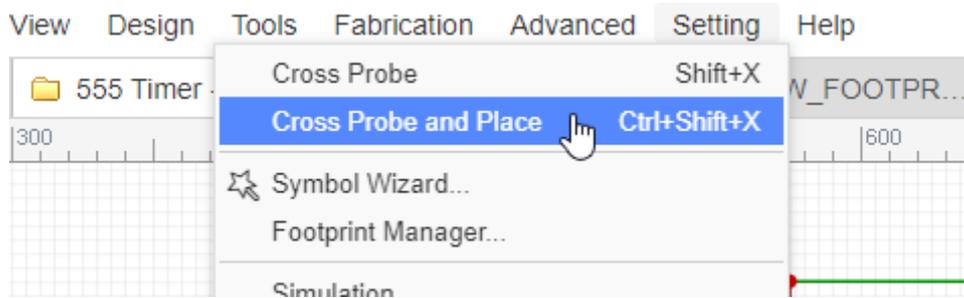
---

# 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".

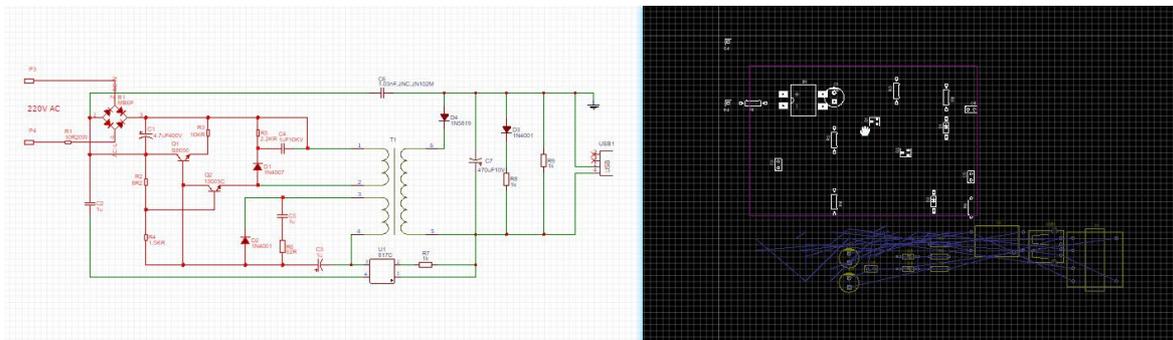
**Top Menu > Tools > Cross Probe And Place**



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

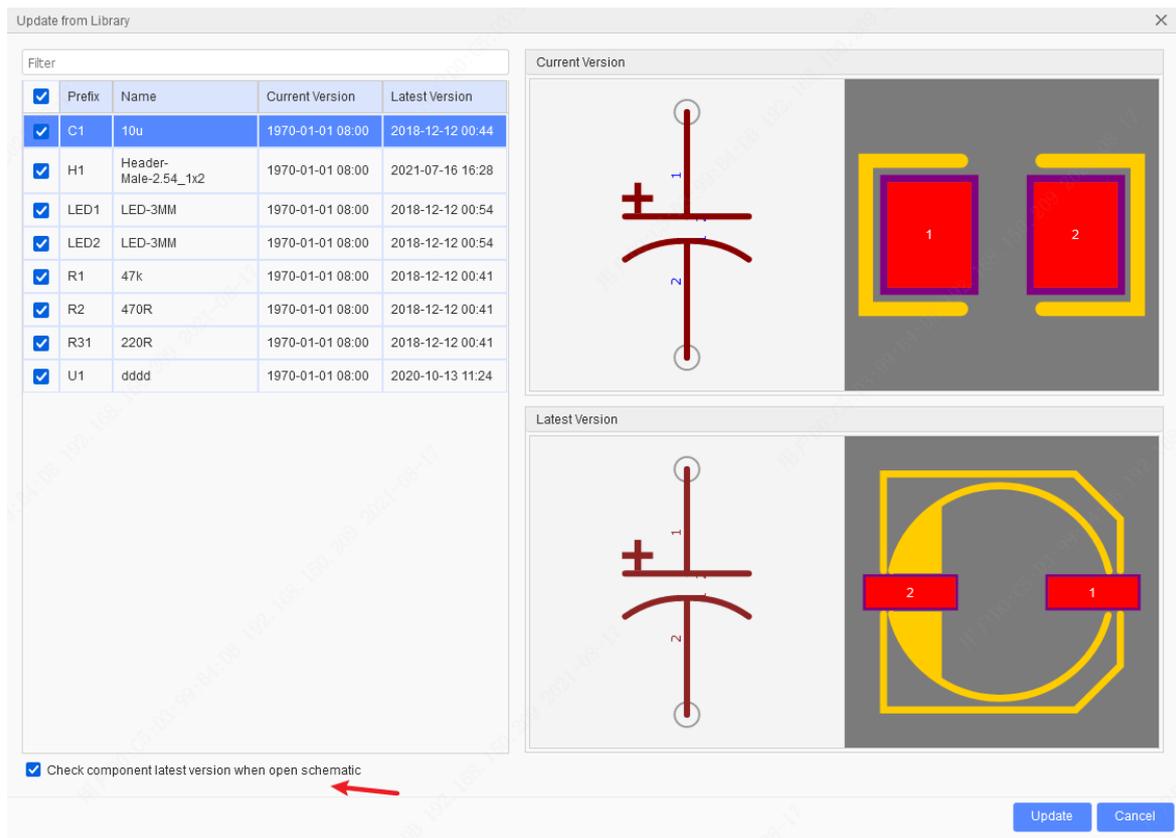
## Update Components from Library

If you want to update the component at schematic when you update the parts at Library, can use this feature.

Via: Top menu - Design - Update Components from Library



Click the menu you will see the update dialog, you can preview the current components and compare with latest version.



You can setting check the component latest version when open the schematic at bottom option.

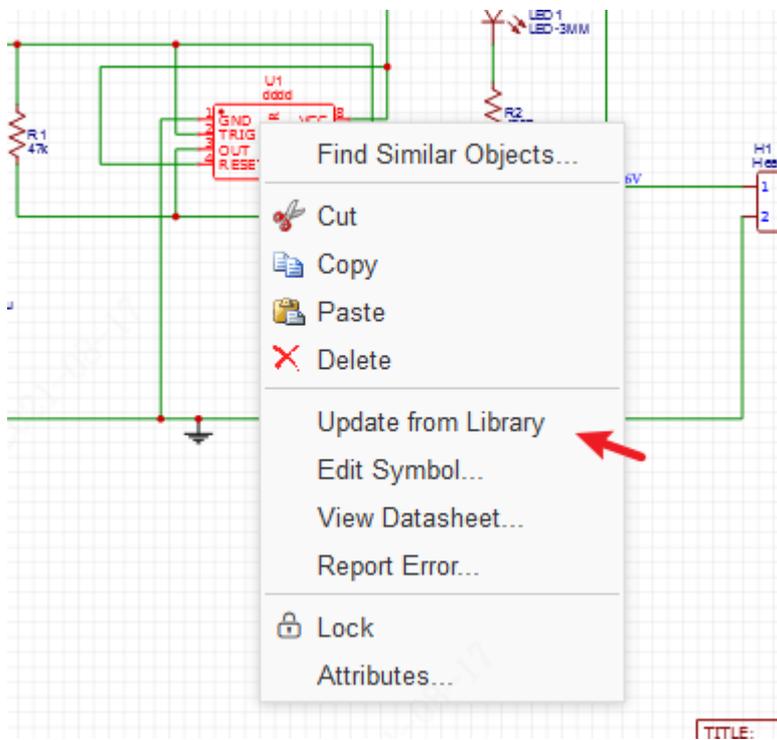
Before update components, please check the parts shape, pin number, footprint carefully.

PCB has this feature too at Design menu, you can update footprint from Library.

Notice:

Since v6.4.20.7, while placing the component at the schematic, that will keep the symbol and footprint corresponding at that time, no matter you update your footprint or not, it will not impact by latest footprint as previous editor version. when you import changes, the footprint will use at that time version, will not use the latest footprint version, if you want to use the latest footprint, you need to update component first.

If you want to update single one component, you can right-click it and update



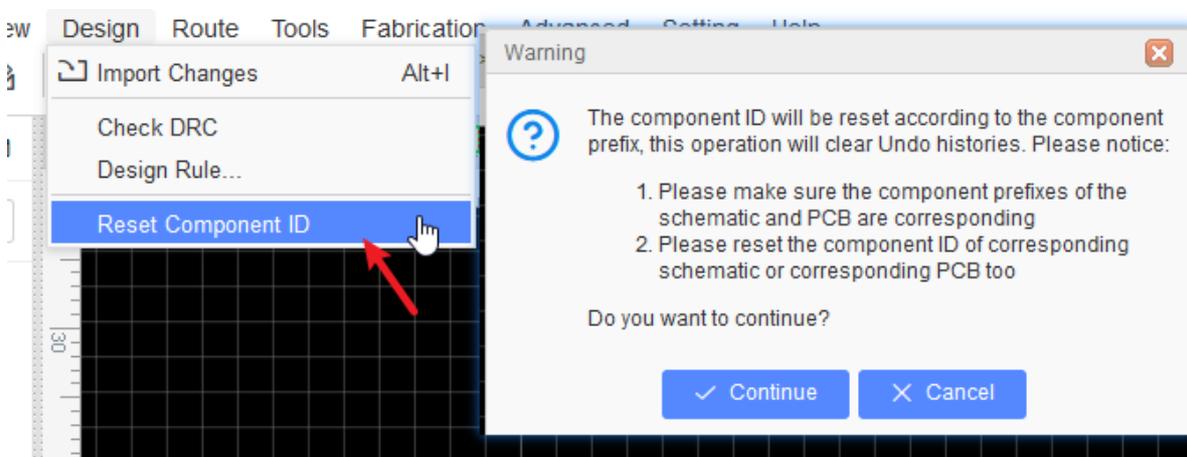
## Reset Component ID

This function resets the ID of the component.

Before V6.4.7, EasyEDA was created by using component prefix to correspond schematic components to PCB components. This method would lead to the situation that after the schematic component was modified with component prefix, the old method would be deleted and a new component would be added when imported and updated into PCB, which would affect the original layout of components.

Since V6.4.7, the component ID is used for matching associations, so old files or imported third-party EDA files can use this feature to reset the component ID so that the two IDs match.

Via: Top Menu - Design - Reset Component ID



Notice:

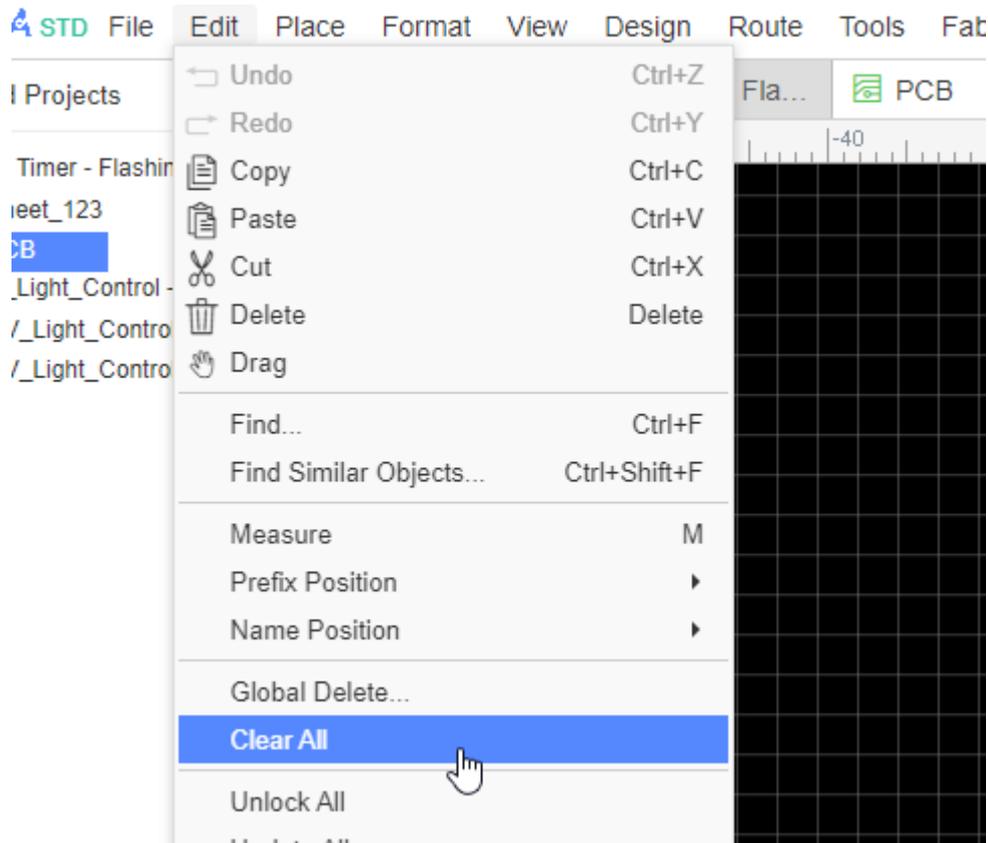
- 1. Make sure that the schematic diagram and the component prefix of the PCB match. The reset ID is reset based on the component prefix.

- 2. It is necessary to reset the component ID on both schematic diagram and PCB so that the component ID on both sides can match one by one.
- 3. For schematic diagram with subparts, it is necessary to change the component prefix of the subparts of PCB pair to U1.1 or other subparts prefixes before resets the ID, otherwise this component will still be deleted and replaced with a new one. Subsequent releases will solve this issue.

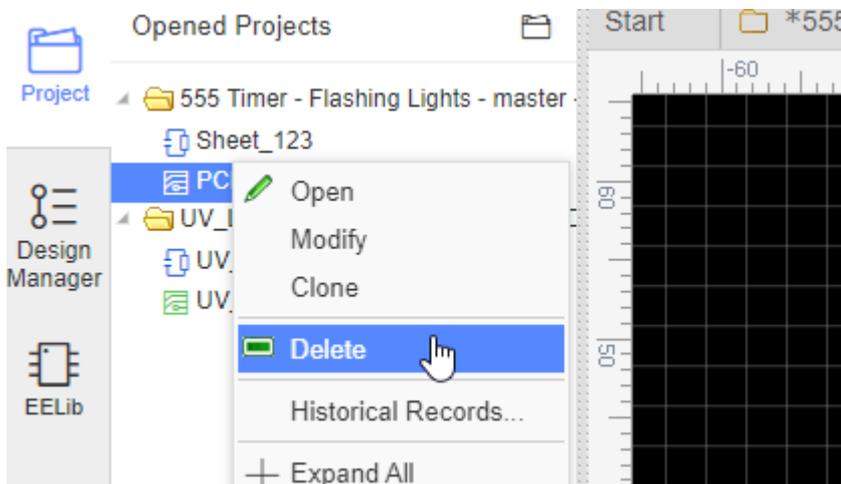
## Global Delete

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

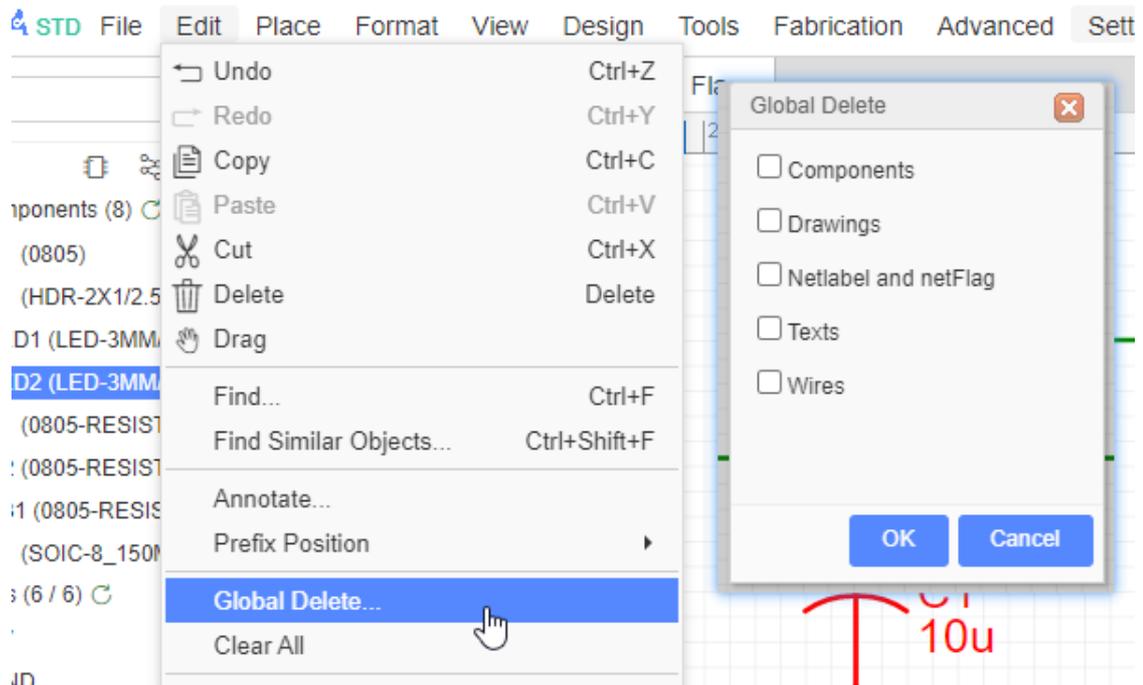
- **Top Menu > Edit > Clear All**, or CTRL + A select all and then press Delete key.



- Delete the document and create a new one.



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



## Schematic Modules

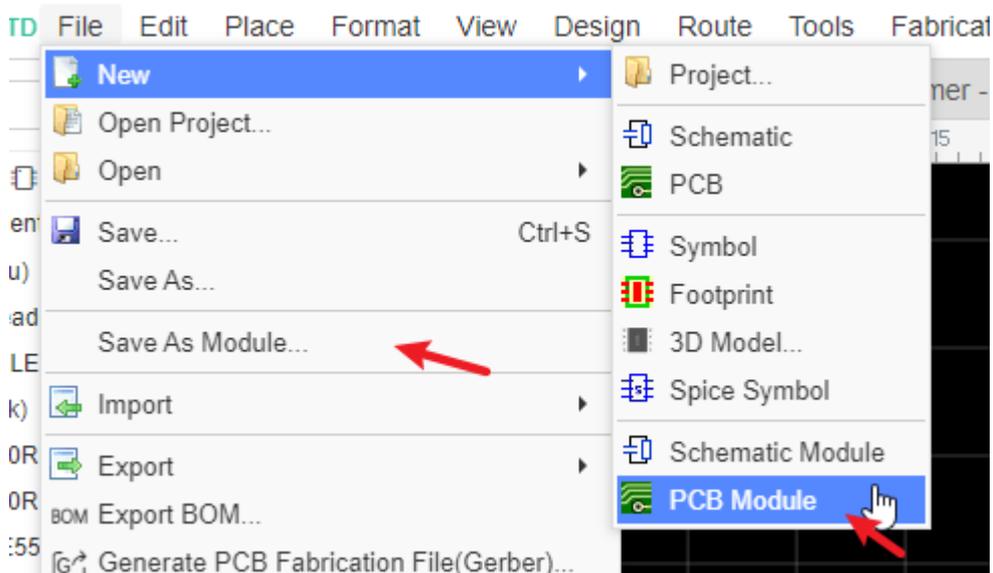
Copying codes is an easy job for coders, now copying and reusing a schematic or PCB is easy. Take a power supply unit for example, you can save this unit as a schematic module.

Via **File > Save as Module**:

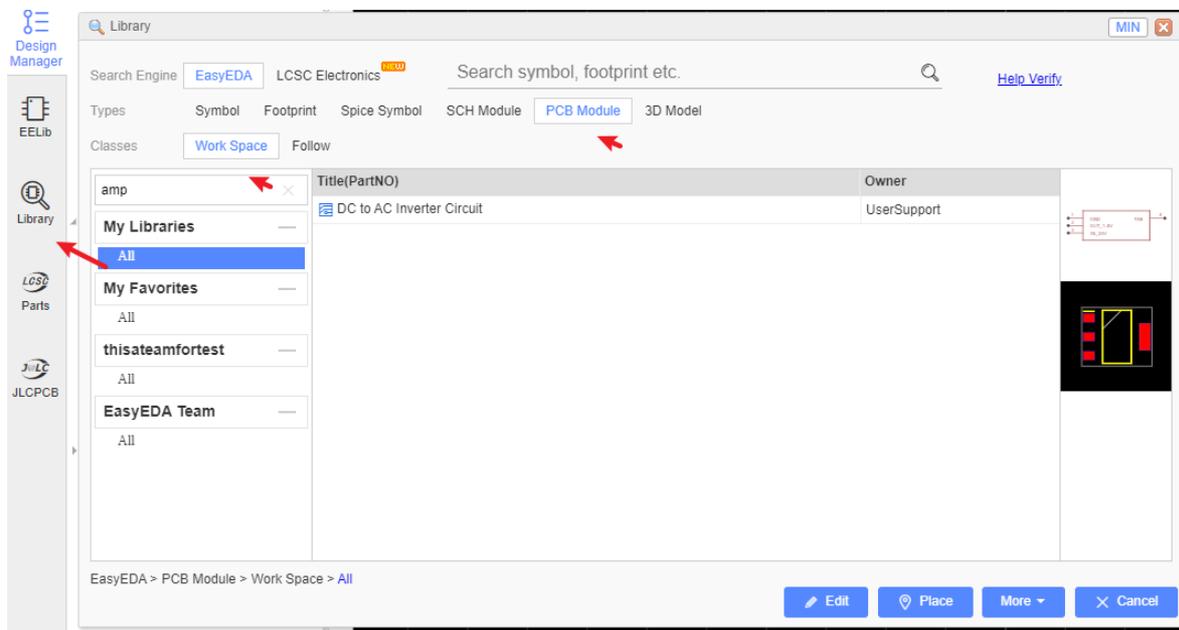
EasyEDA support create the PCB modules, it seems schematic module.

## 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**

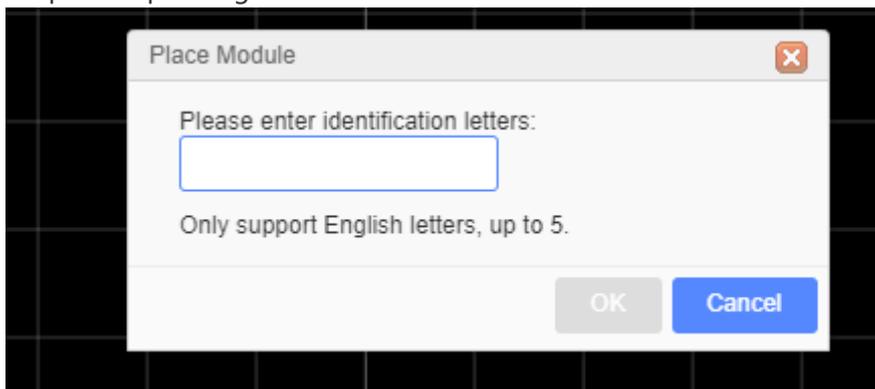


## How to use

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.

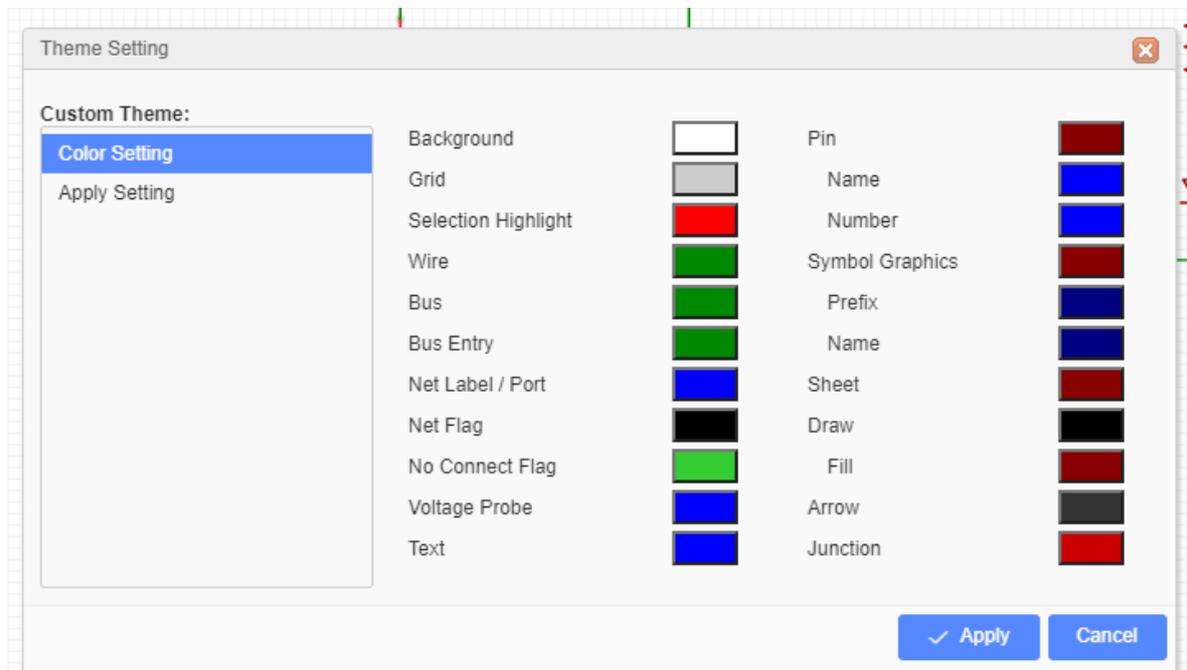
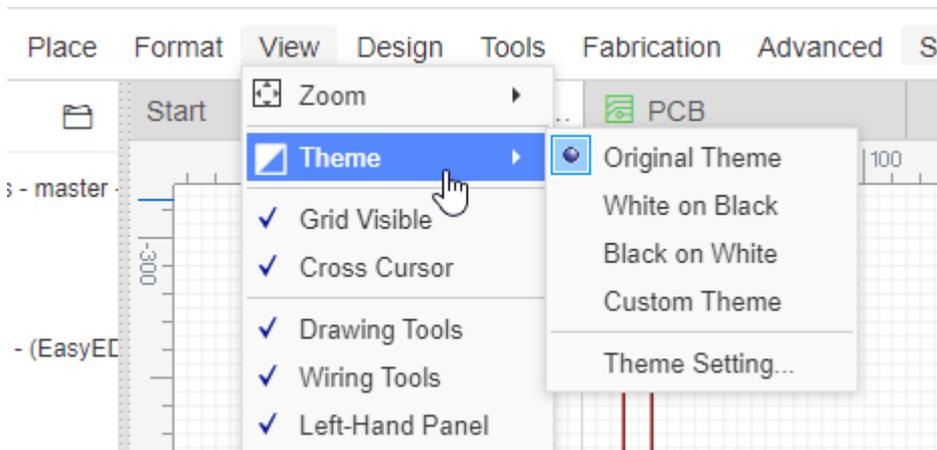
---

## 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.

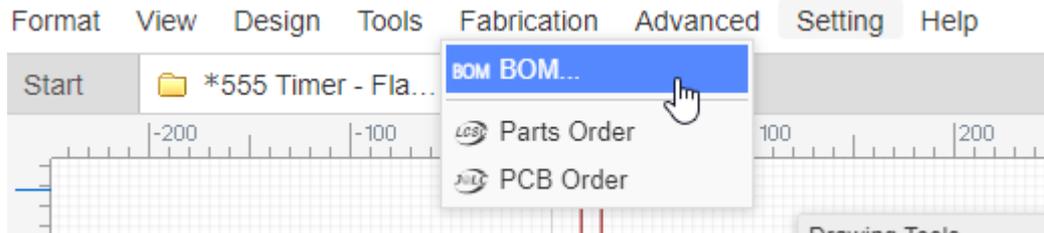
**My theme Settings:** You can apply "My theme" on: 1. Creating New Schematic, 2. Opening Existed Schematic.

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 synchronized to the server by default.

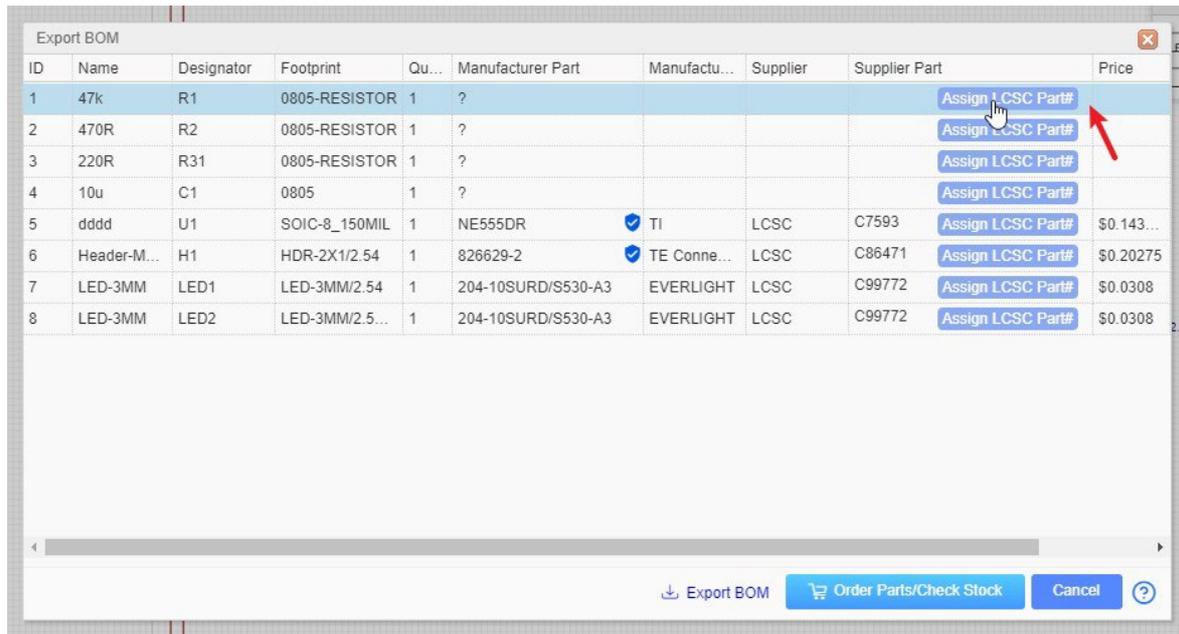
## Export BOM

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".

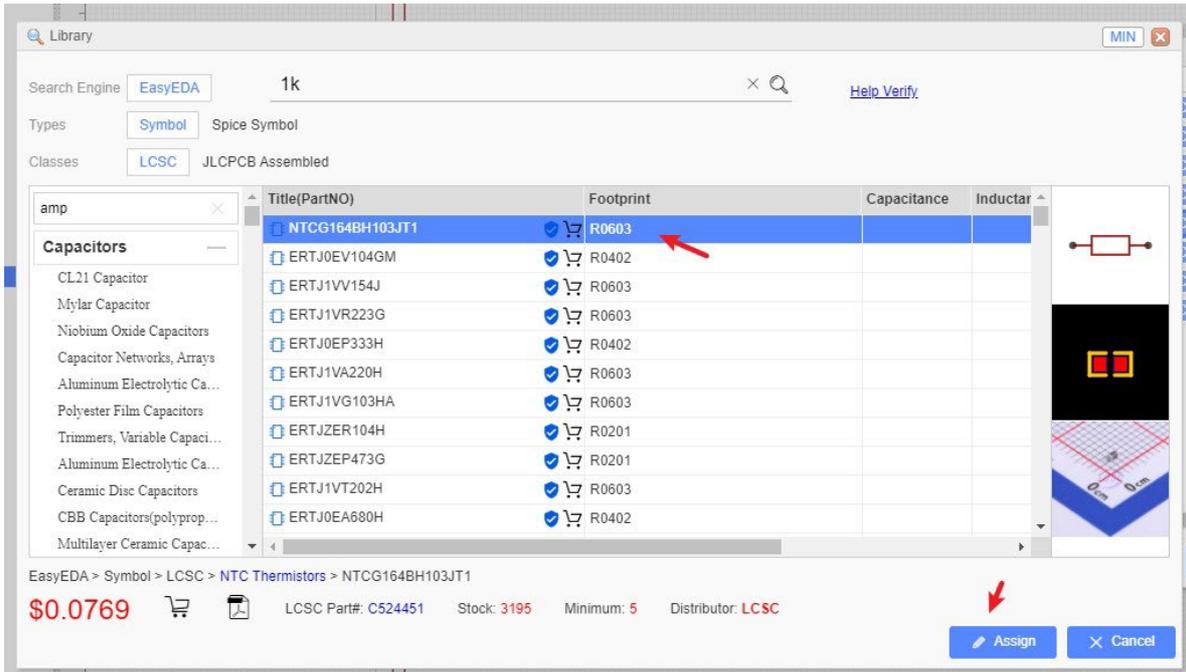


After clicking the BOM export option, the dialog below will open.

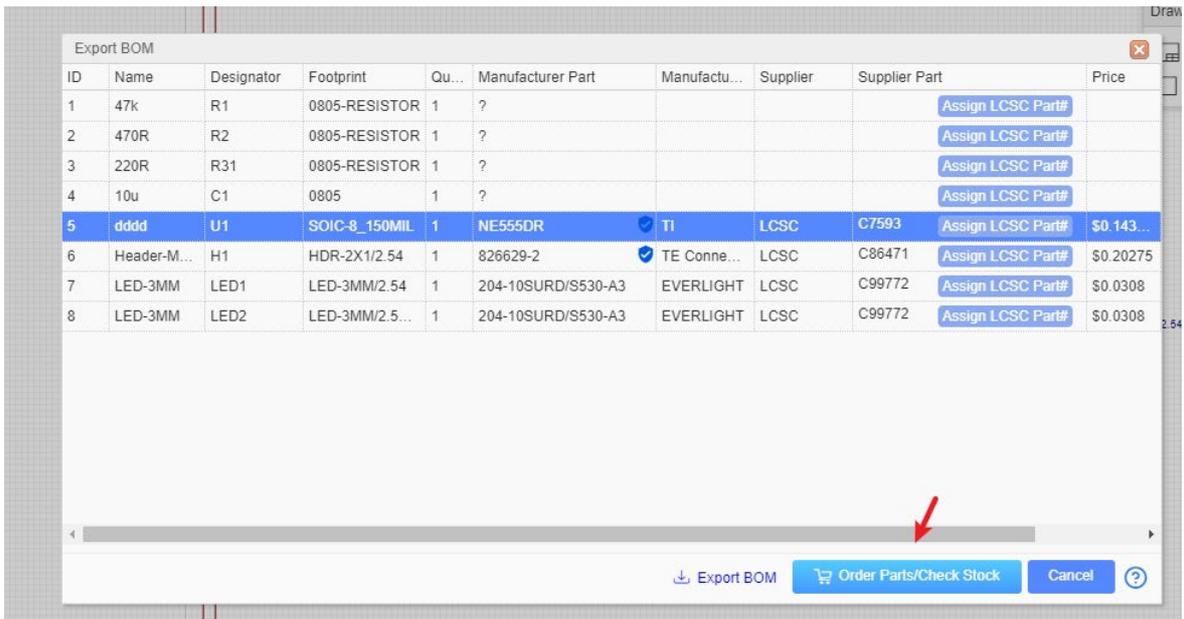
In this dialog, you can click the button to assign LCSC part's order code for your components.



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 form 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	K
	ID	Name	Designator	Footprint	Quantity	Manufacturer	Manufacturer	Supplier	Supplier Pa	LCSC Assembly	
	1	HDR-M-2.54	KJ1,AJ1,BJ1	HDR-M-2.54	8			LCSC	C66690		
	2	NE555P~NA	U1	DIP-8	1	NE555P	TI	LCSC	C46749		
	3	MC306(6pF	C1	CAP-D3.0X	1	HV010M050	CapXon	LCSC	C59954		
	4	0.1u	C63,C73	C1210K	2						
	5	MC306(6pF	C8	C1210	1						
	6	19-217/GHC	LED1,LED2	LED0603-R-	2	19-217/GHC	EVERLIGHT	LCSC	C72043	Yes	
	7	1N4148W	KD1,AD1,BI	SOD-123FL	8	1N4148W	Tak Cheong	LCSC	C129216		
	8	CAP-1uF	C2	C0805	1	RVT2A1R0M	HONOR	LCSC	C87863		
0	9	CAP-1uF	C4	RAD-0.1	1	?					
1	10	CAP-1uF	C5	R0805	1	?					
2	11	HDR-IDC-2.	P1	IDC-TH_6P-	1	2X3 2.54mm	BOOMELE	LCSC	C11214		
3	12	0.1u	KC1,AC1,BC	C1210	8						
4	13	1KOHM	R2	R0805	1	?					
5	14	1KΩ	R1	AXIAL-0.3	1	?					
5	15	2N3906(TO	KQ1,AQ1,BI	TO-92-3_L4	8	2N3906	CJ	LCSC	C9809		
7	16	1m	KL1,AL1,BL1	L0402	8						
3											

Export BOM supports to export LCSC part price, it is the same as LCSC website.

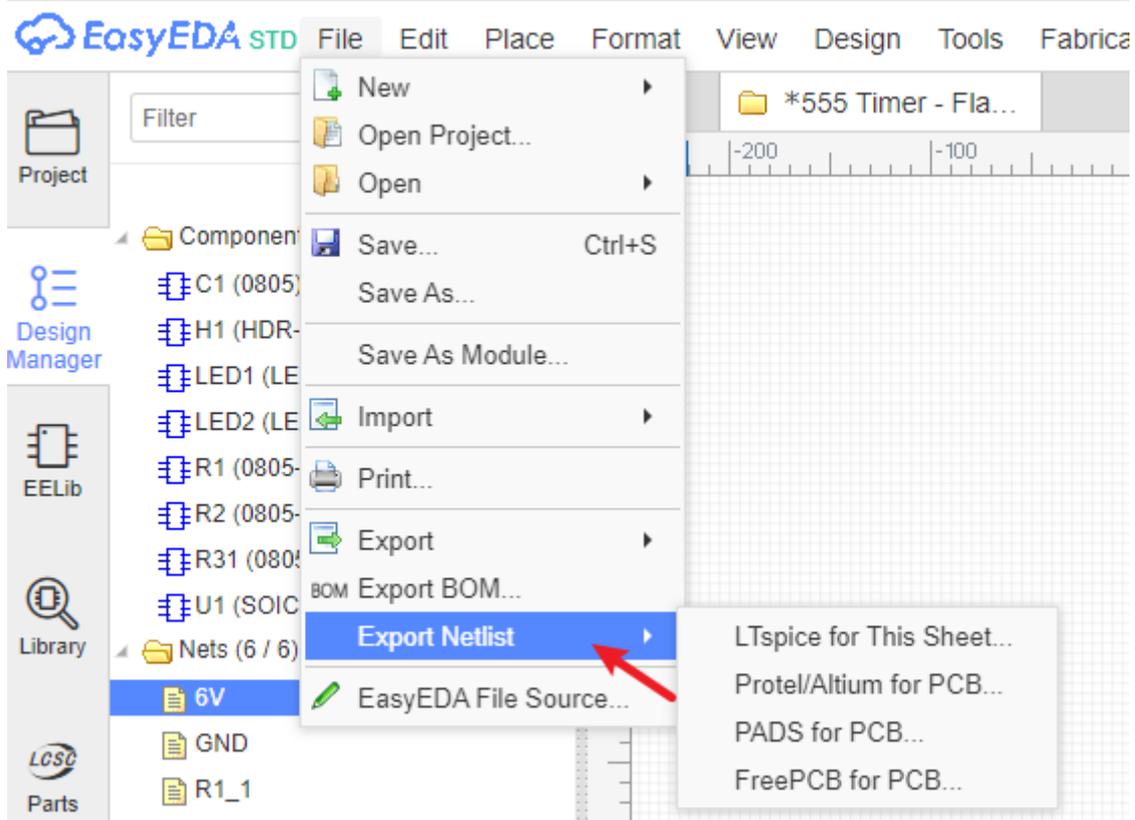
#### Notice:

- Before v6.4.17, 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.
- Since v6.4.17, the schematic BOM and PCB BOM are separated. If you assign the LCSC part at the PCB, it will not modify the schematic.
- 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.

---

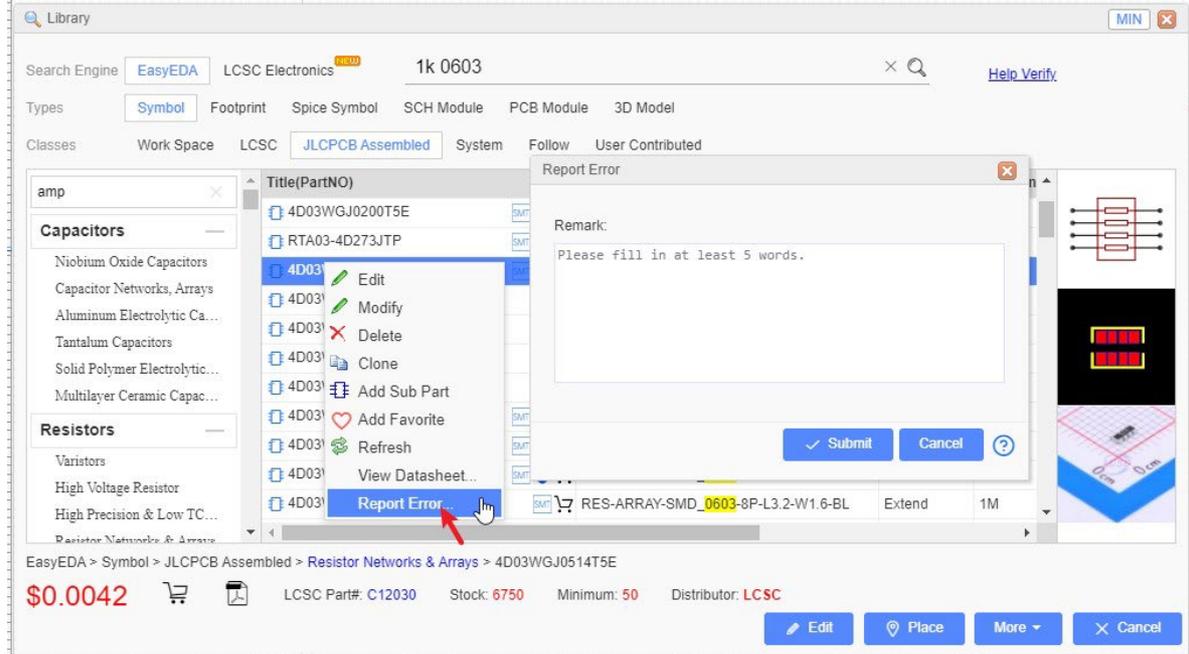
## Report Error

---

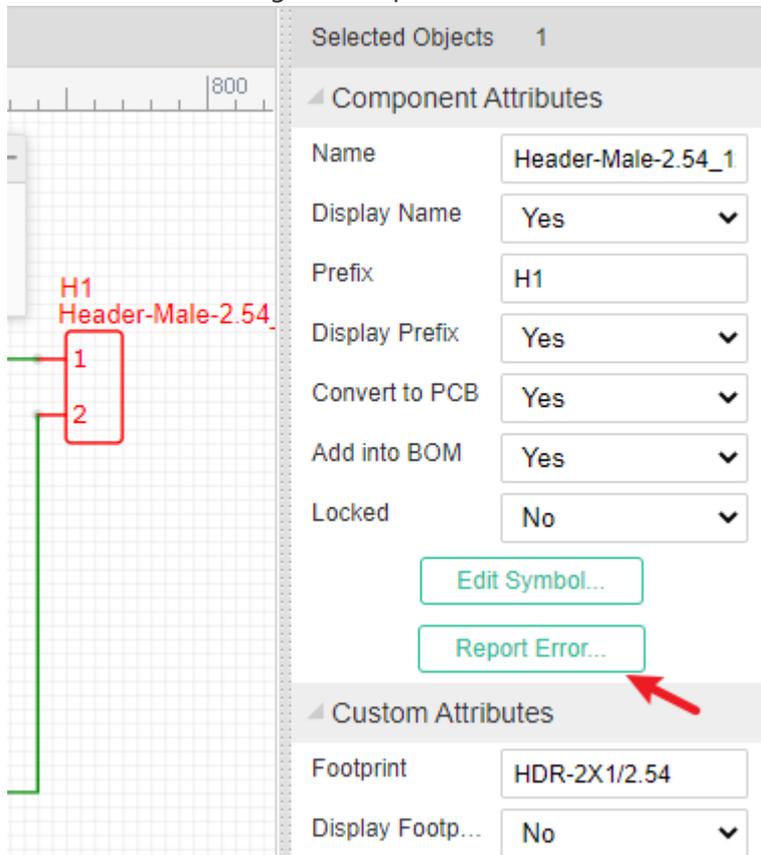
For EasyEDA official libraries, we have staffs to draw and maintain(LCSC & JLCPCB Assembled part) and we will try to keep them correctly as we can, but EasyEDA(System part) included a lot of open source of the libraries and the official drawing of the libraries, that can not avoid the wrong situation 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:

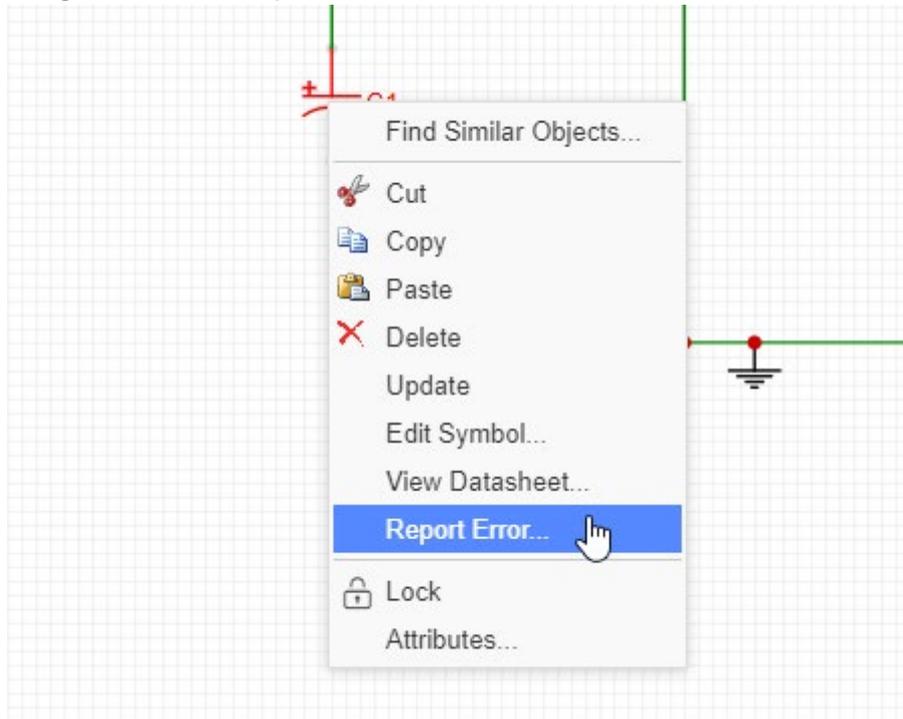
1. Right-click the official library and use the "Report Error" function on the "Libraries".



Report Error" button at the right-hand panel.



or right-click the component:



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

---

# Experiment no: 13

## Experiment Title: Creating Schematic Symbol in EasyEDA

### Create the Schematic Symbol

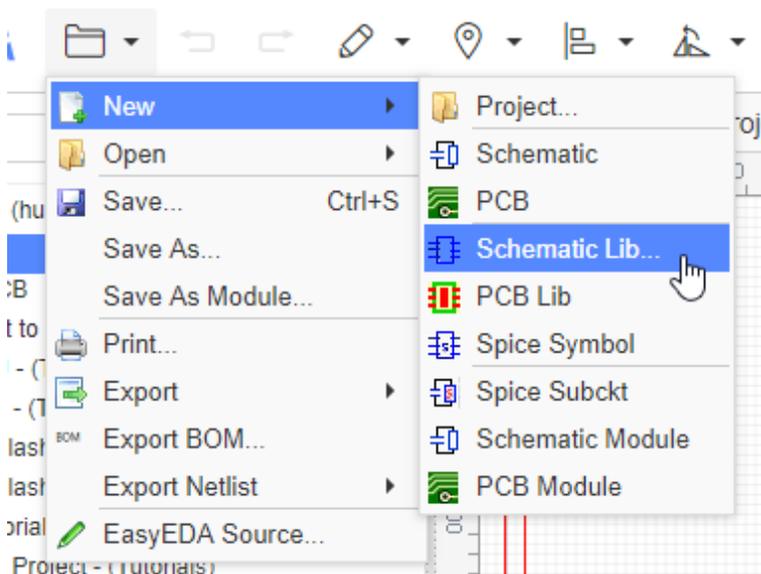
Using **Schematic Symbol Wizard** and **Group/Ungroup...** is a quick way to create schematic symbols but they are placed directly into the schematic that they are built in.

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

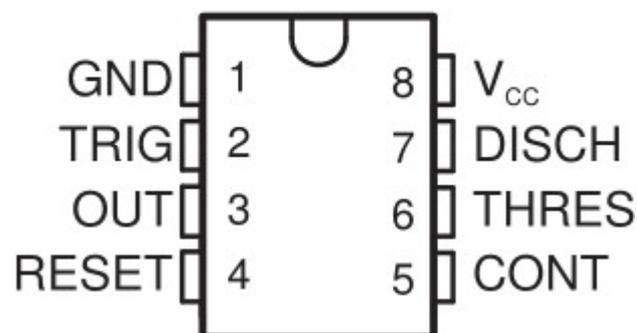
- ◆ **Get the Datasheet**

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

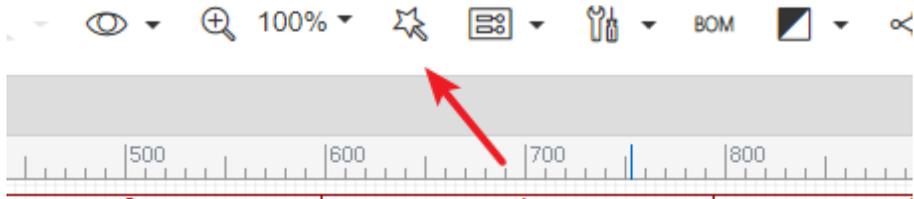
And then create the symbol and place the pins for the library base on the datasheet.

This component have 8 pins and names.

(TOP VIEW)



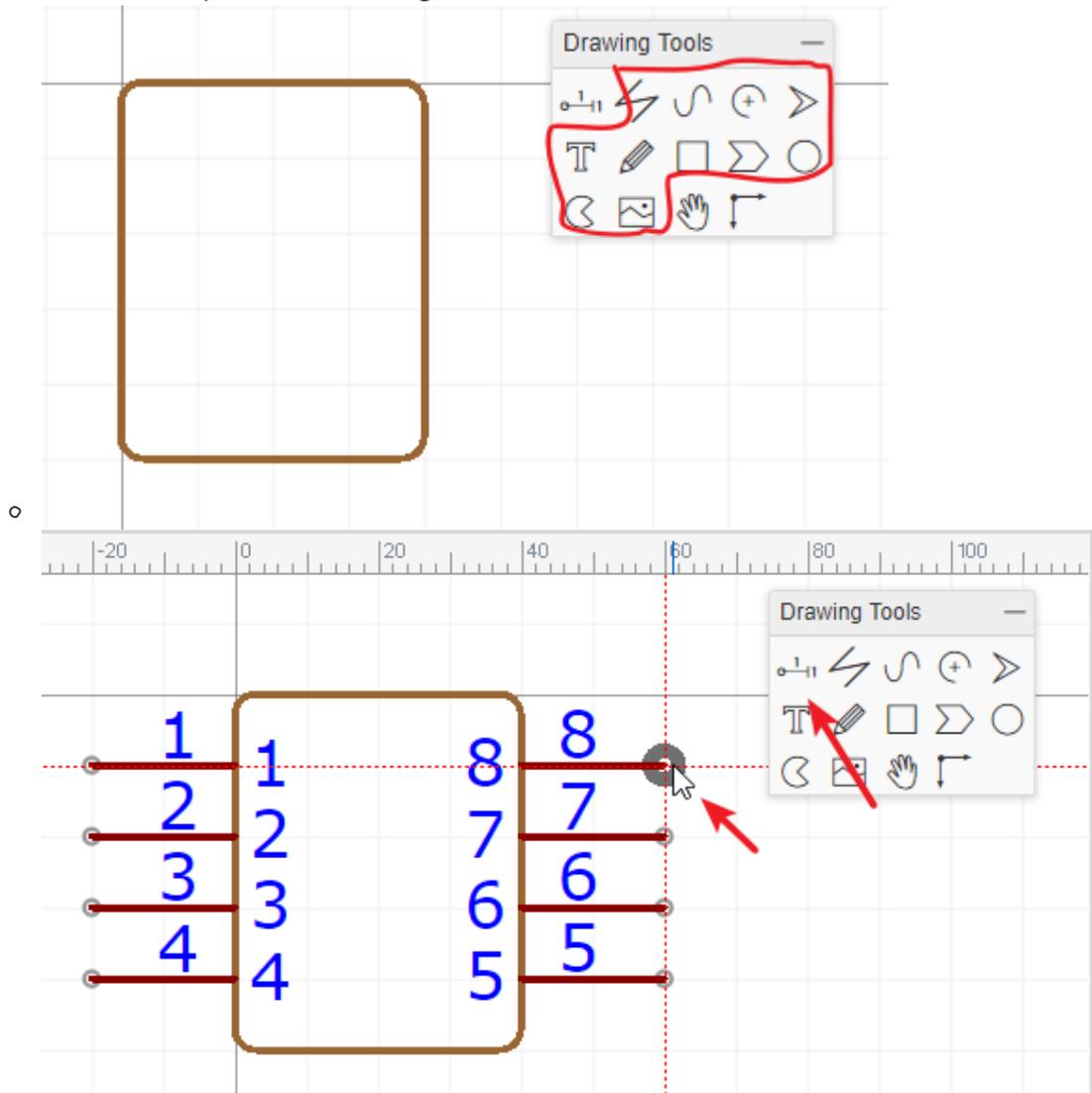
- ◆ Create via Schematic Symbol Wizard



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

- ◆ Create by Manually

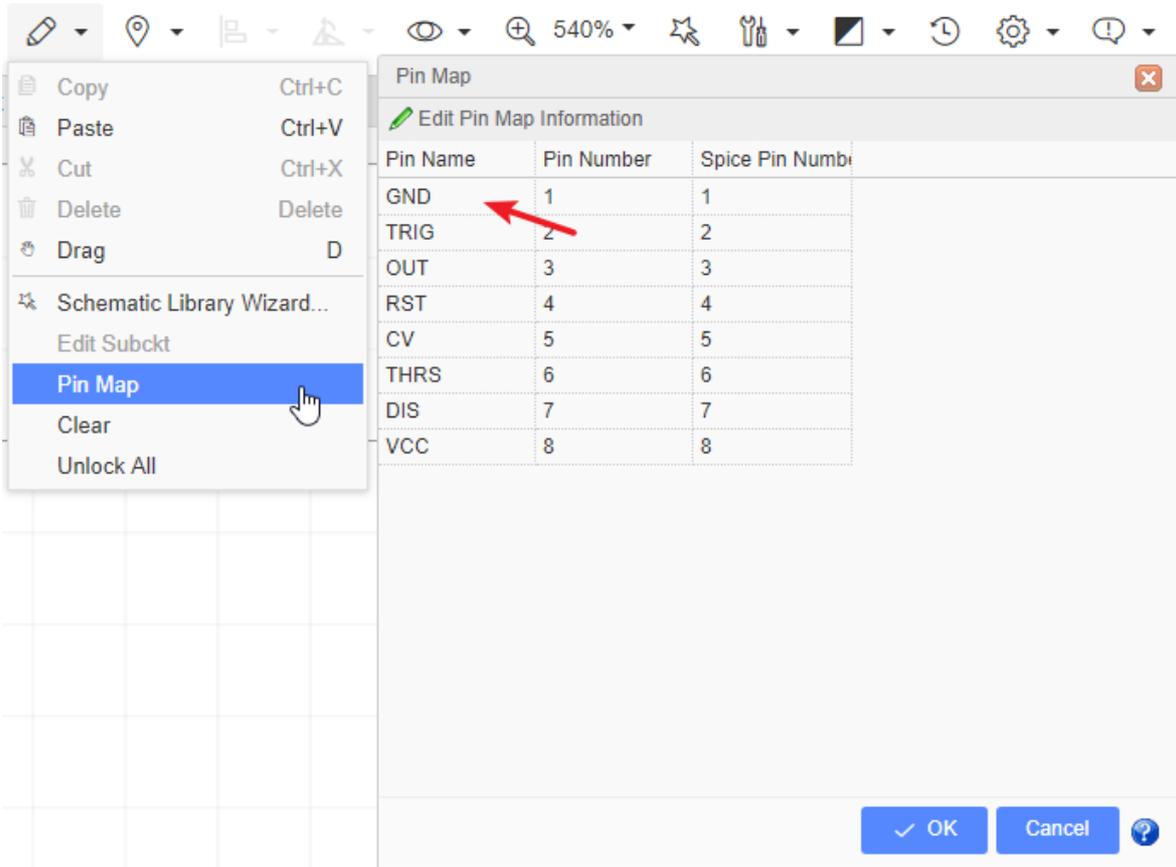
- Draw the shape via the Drawing Tools



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

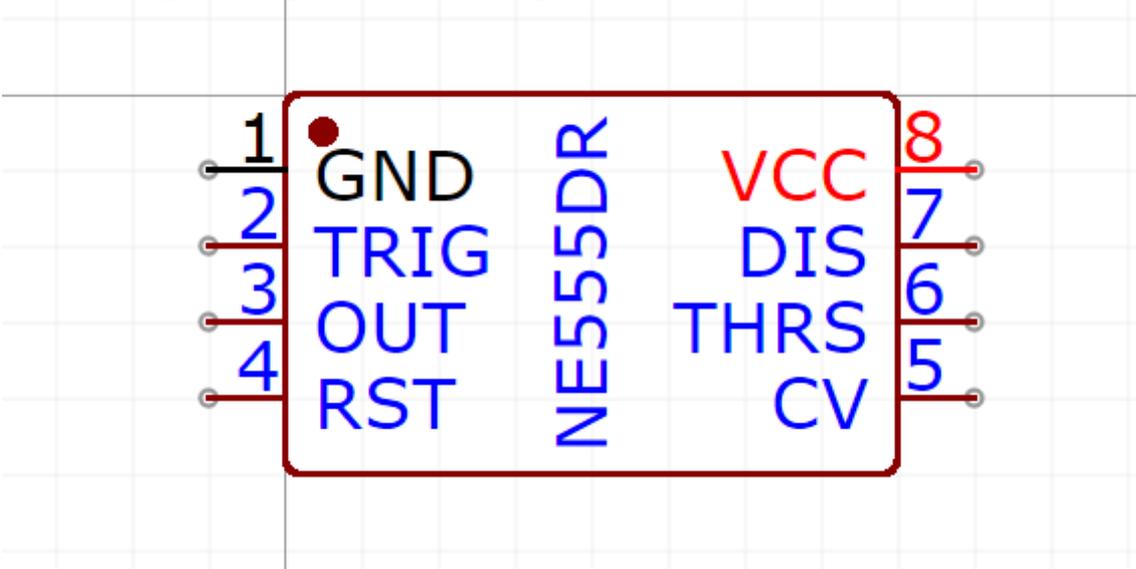
Via **Edit > Pin Map...**, change Pin names and Pin numbers. For some complicated IC, will use the alphabet for the pin number.



Note: if the pin is hidden, a network will be generated according to the pin name for connection. If it is not necessary, it is recommended not to hide.

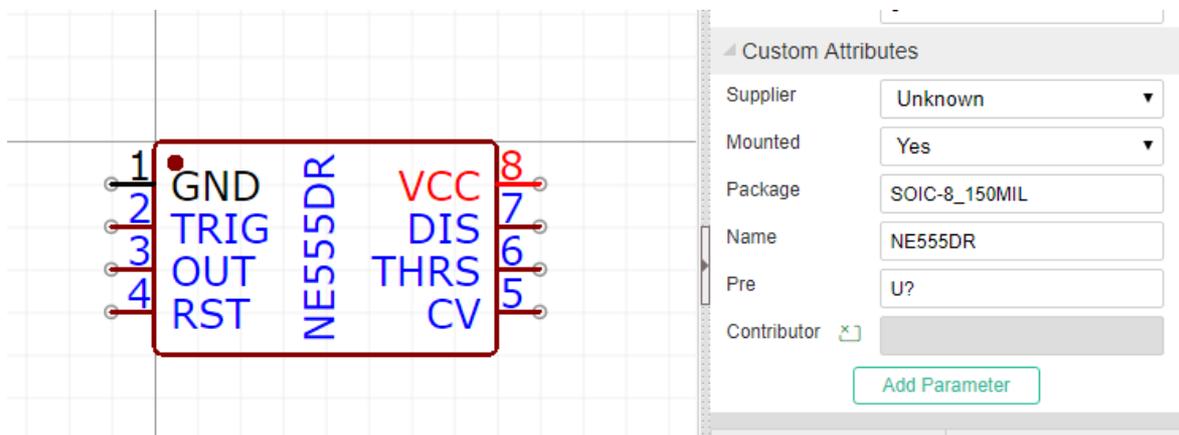
### 4. Modify the Detail

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 package, 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

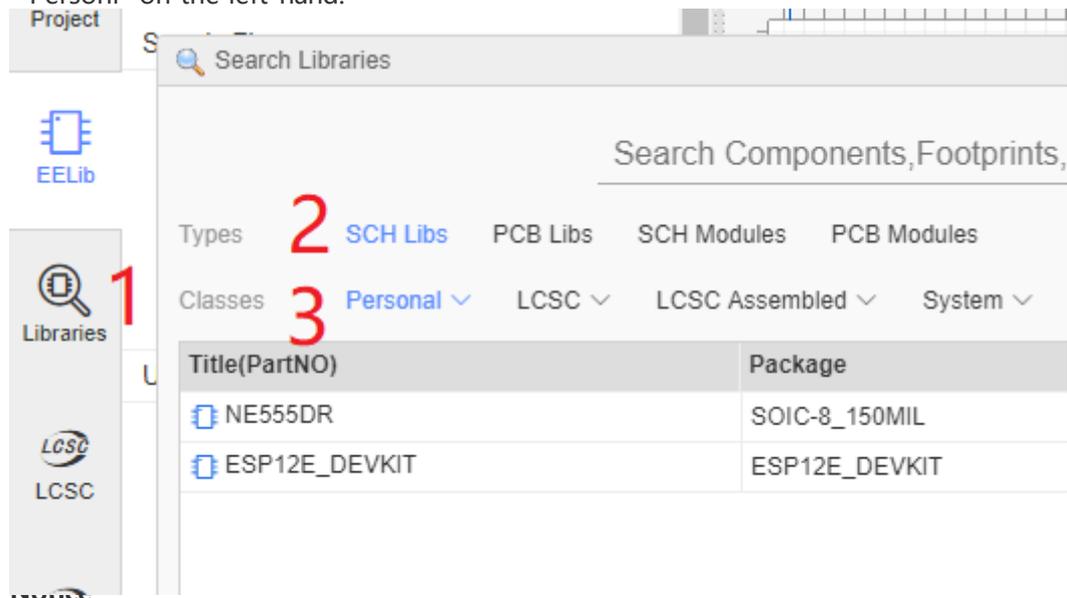
You can via: "Top Menu - Place - Set Canvas Origin - By Center Grid of Symbols" to set the origin.

## 7. Save your SchameticLib

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

The image shows a 'Save as a schematic Lib' dialog box with the following fields and values: Owner: Tutorials; Title: NE555DR; Manufacturer Part: NE555DR; Supplier: Unknown; Supplier Part Number: 296-6501-2-ND; Link: http://www.ti.com/lit/ds/symlink/ne555.pdf; Tags: 555 Timer; Description: 555 Timer. There are 'Save' and 'Cancel' buttons at the bottom right.

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

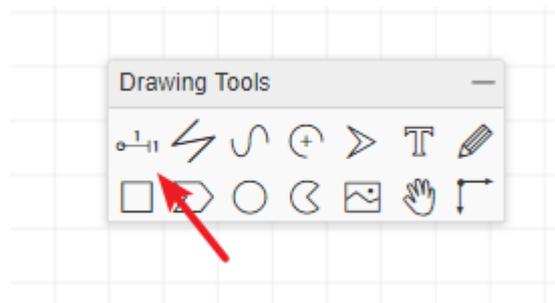


- **Note the Origin Point.** To simplify rotating your symbols when they are placed into the canvas, make sure all of your symbols are created as near as possible centered around that point. Suggesting the first Pin/Pad or its center to be the origin point.
- Please make sure all pins dot are placed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

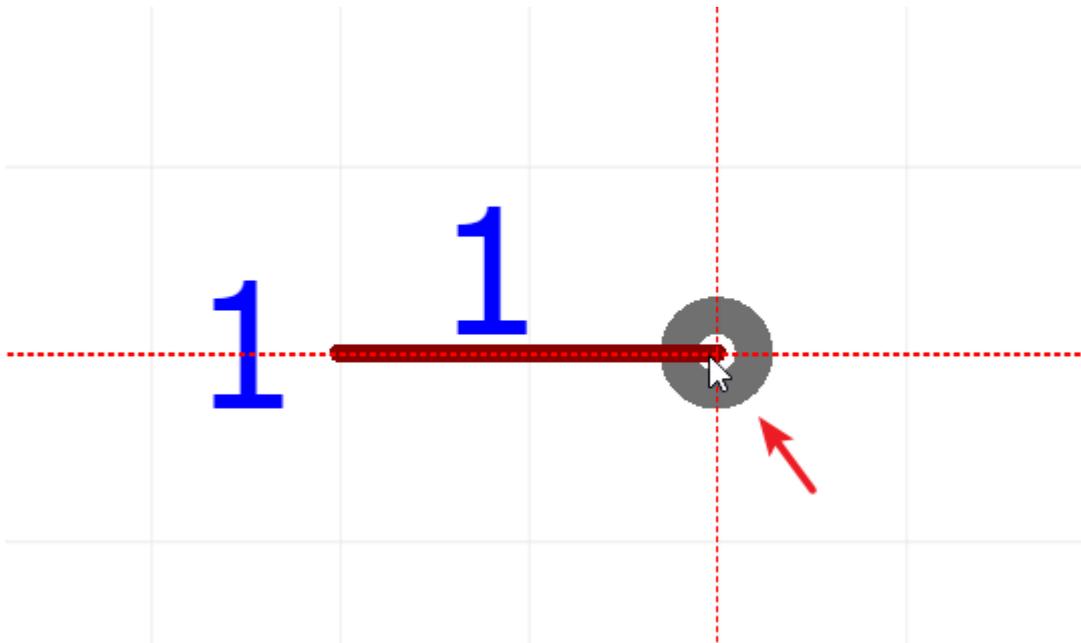
## Pin Attributes

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.

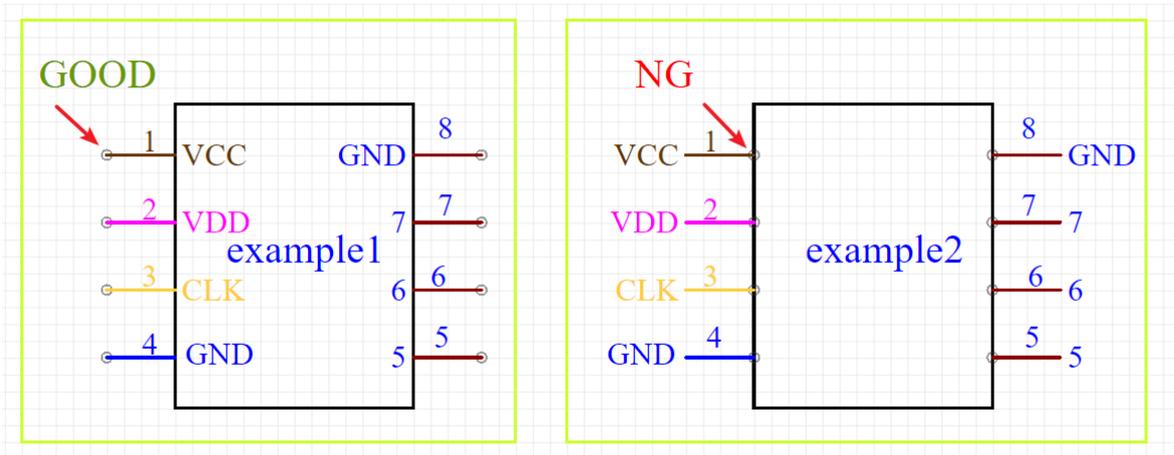
You can use the **P** hotkey to add a Pin or from the Draw Tools palette:



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:

The image shows a software interface for creating a pin. On the left, a grid displays a red pin labeled 'VCC 1' with a dot at its end. The grid has a horizontal axis with markers at 250 and 300. On the right, a 'Pin Attributes' panel is open, showing various settings for the selected pin.

Pin Attributes	
Orientation	0°
Start X	300
Start Y	90
Length	20
Name	VCC
Number	1
Spice Pin Order	1
Name Display	Yes
Number Display	Yes
Color	#880000
Dot	No
Clock	No
Show	Yes
Electric	Undefined
Font Family	Verdana
Font Size	7pt
Locked	No

**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 pin dot 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.

**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.

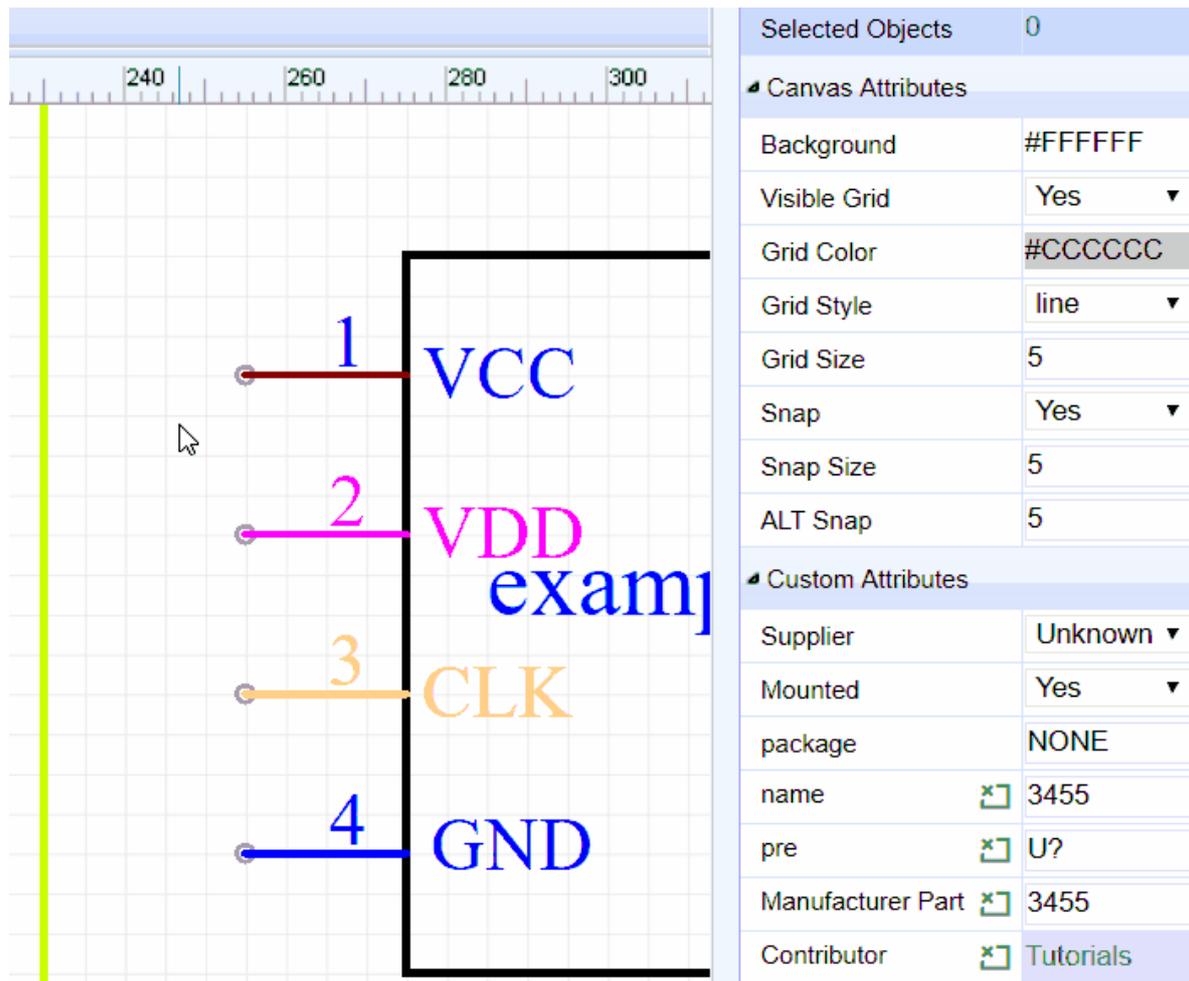
You can adjust the Name or Number position using your mouse but note that rotate and flip applies to the whole pin including the name and pin number; these items cannot be rotated and flipped independently of the pin itself.

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.



The screenshot shows the Schematic Lib symbol editor in EasyEDA. On the left, a grid-based canvas displays a pin symbol with four pins. Pin 1 is red and labeled '1 VCC'. Pin 2 is pink and labeled '2 VDD'. Pin 3 is orange and labeled '3 CLK'. Pin 4 is blue and labeled '4 GND'. A yellow vertical line is on the left. The right panel shows the properties for the selected object (0). The 'Canvas Attributes' section includes: Background (#FFFFFF), Visible Grid (Yes), Grid Color (#CCCCCC), Grid Style (line), Grid Size (5), Snap (Yes), Snap Size (5), and ALT Snap (5). The 'Custom Attributes' section includes: Supplier (Unknown), Mounted (Yes), package (NONE), name (3455), pre (U?), Manufacturer Part (3455), and Contributor (Tutorials).

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

Save as a schematic Lib

Owner: **Tutorials** [Create Team](#)

Title: **NE555DR**

Manufacturer Part: **NE555DR**

Supplier: **Unknown** Or

Supplier Part Number: **296-6501-2-ND**

Link: **http://www.ti.com/lit/ds/symlink/ne555.pdf**

Tags: **555 Timer**

Description: **555 Timer**

Save

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

Search Libraries

Search Components, Footprints, Modules

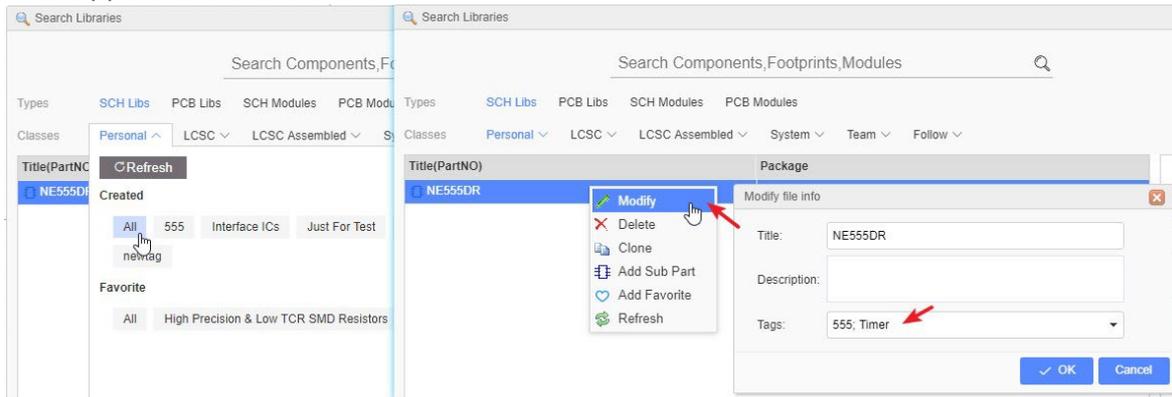
Types: **SCH Libs** | PCB Libs | SCH Modules | PCB Modules

Classes: **Personal** | LCSC | LCSC Assembled | System | Team | Follow

Title(PartNO)	Package
<b>NE555DR</b>	SOIC-8_150MIL

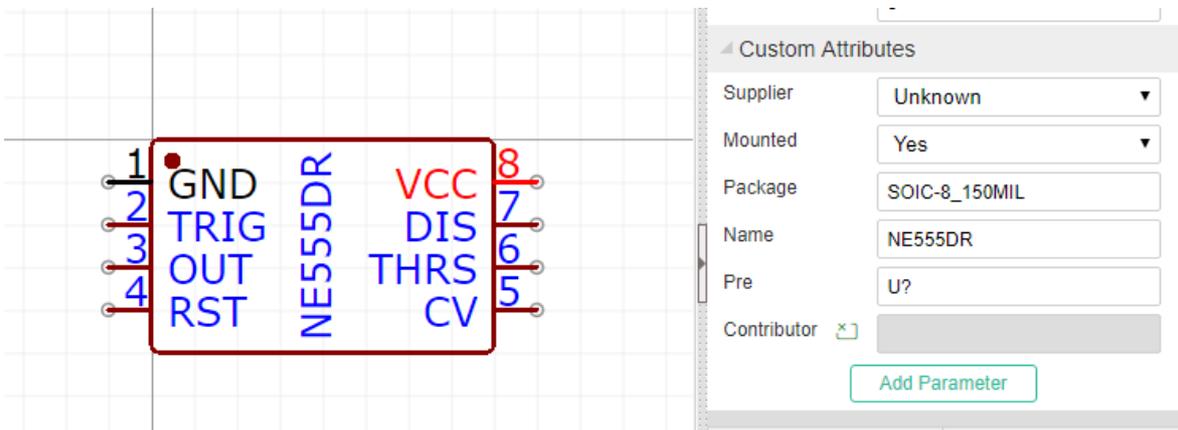
SCH Libs > Personal > Created > 555 > NE555DR

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 appear 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 build 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

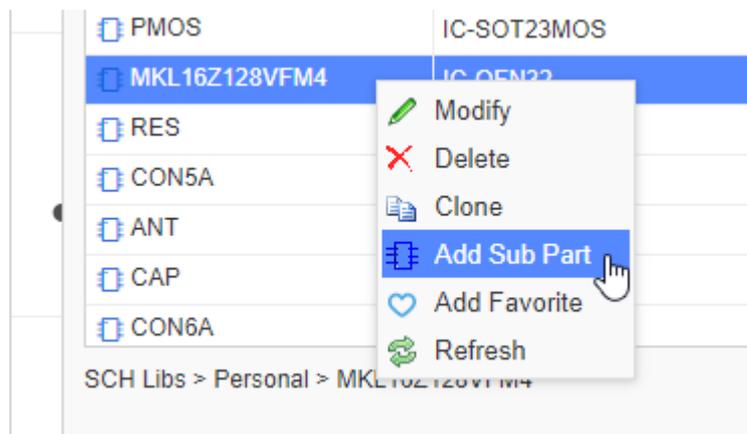
---

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?

---

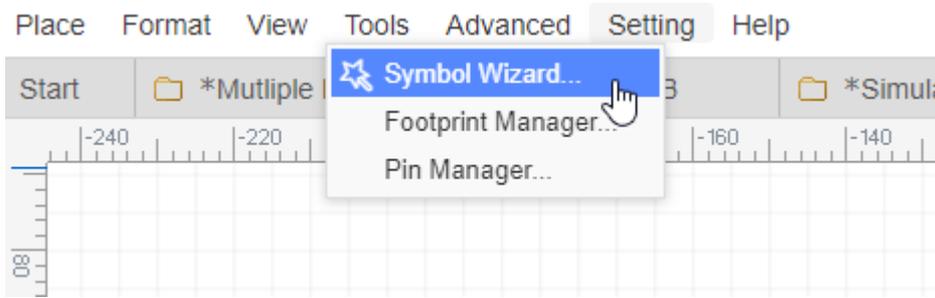
## Schematic Symbol Wizard

---

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.

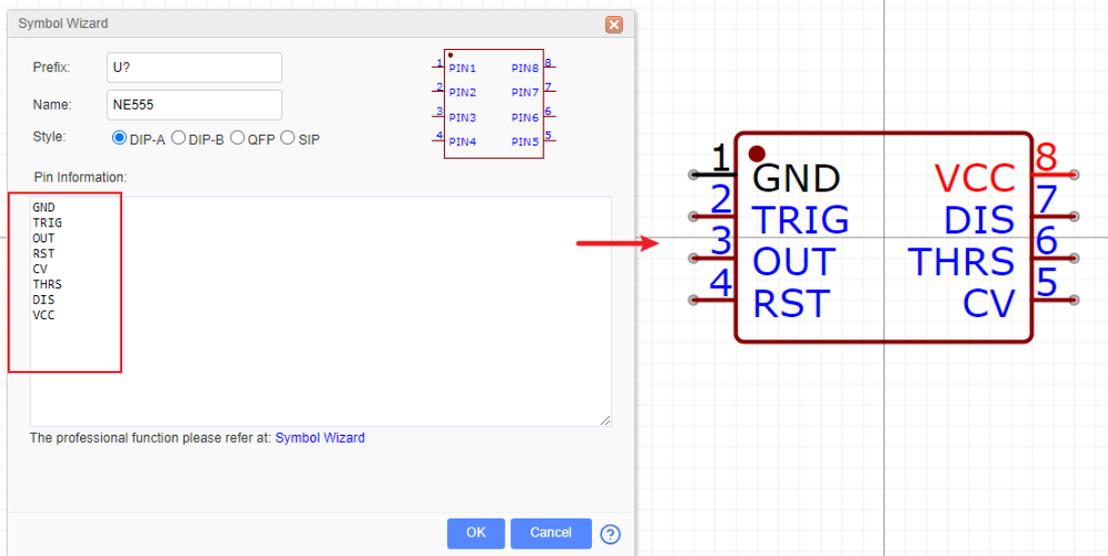
Via: Top Menu - Tools - Symbol Wizard



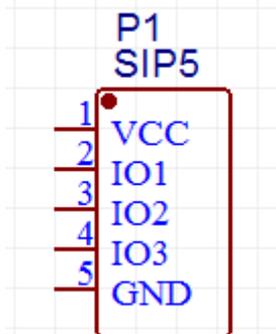
## Basic Function

### 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.

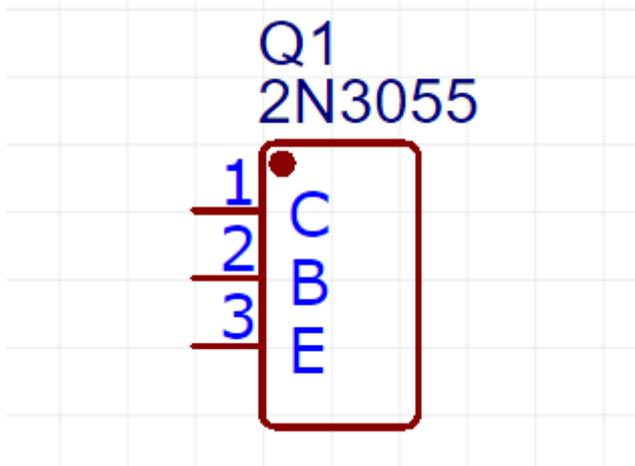


2. The EasyEDA Schematic Symbol Wizard allows you to create DIP, QFP or SIP styles symbols. If you are designing Arduino Shields then you will need lots of SIP symbol, so you can create a SIP symbol like the one shown below in a few seconds.



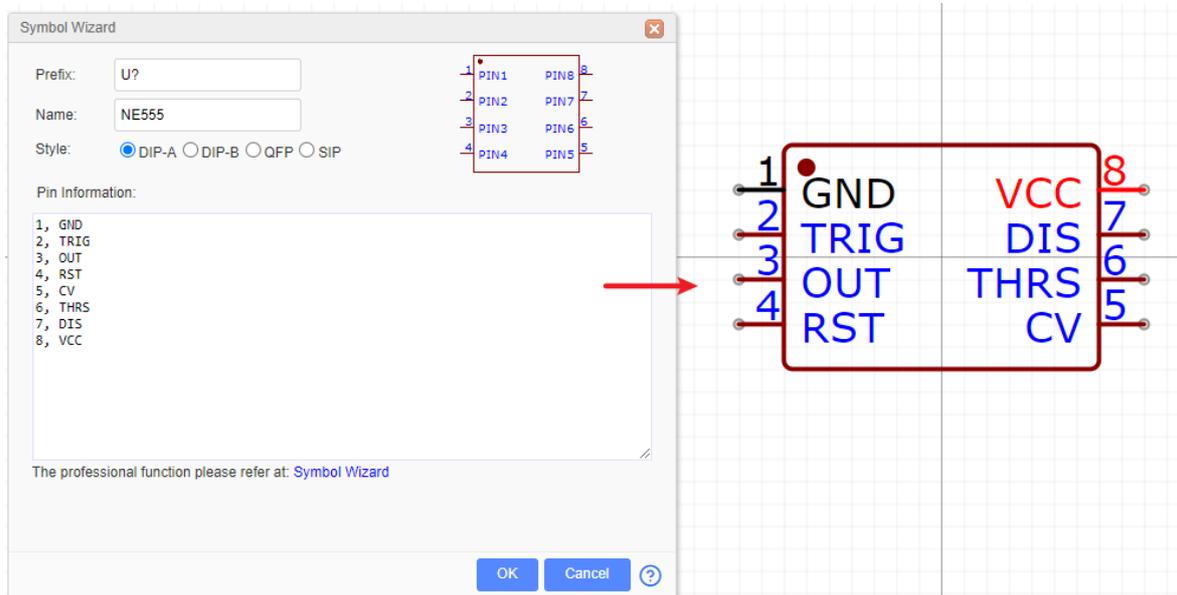
3. If you are not too worried that the symbols may not look quite the way people might expect and that they may not look anything like the Type you select, then of course you

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.

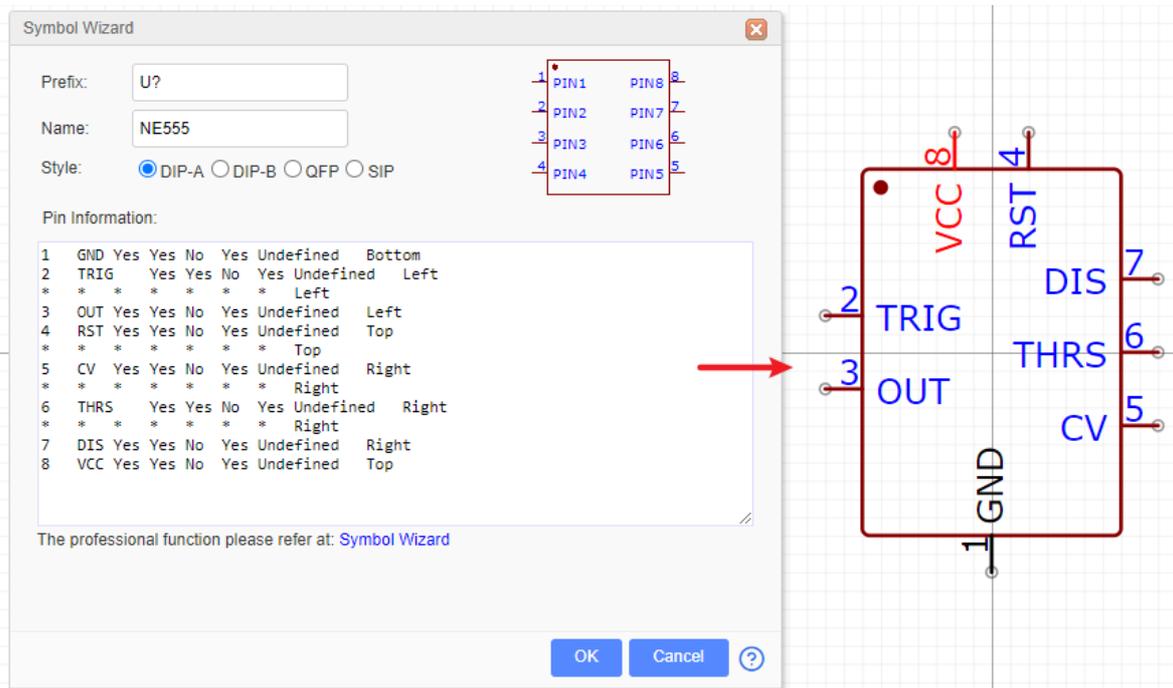
1. Download [Schematic Symbol Wizard Template.xlsx](#)

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
1	Please copy the content without title, and paste on the schematic library wizard.								
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.

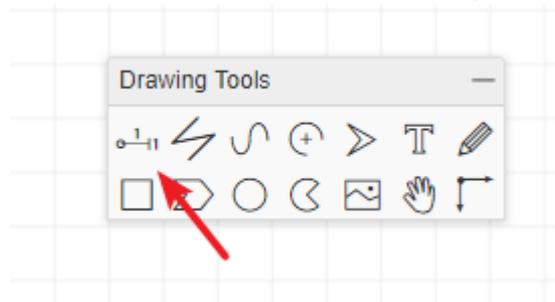
# Experiment no: 14

## Experiment Title: Schematic Symbol Attributes in EasyEDA

### Pin Attributes

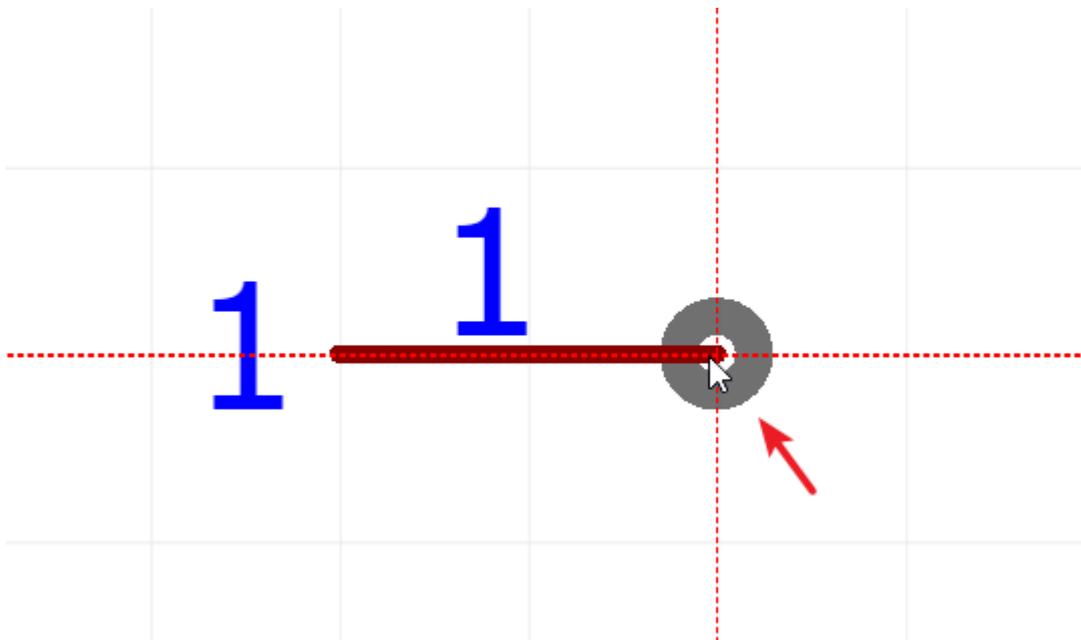
Symbols pins are the most important part of any Schematic Lib symbol. They are the things that allow wires to be attached to symbols to connect up your circuit.

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

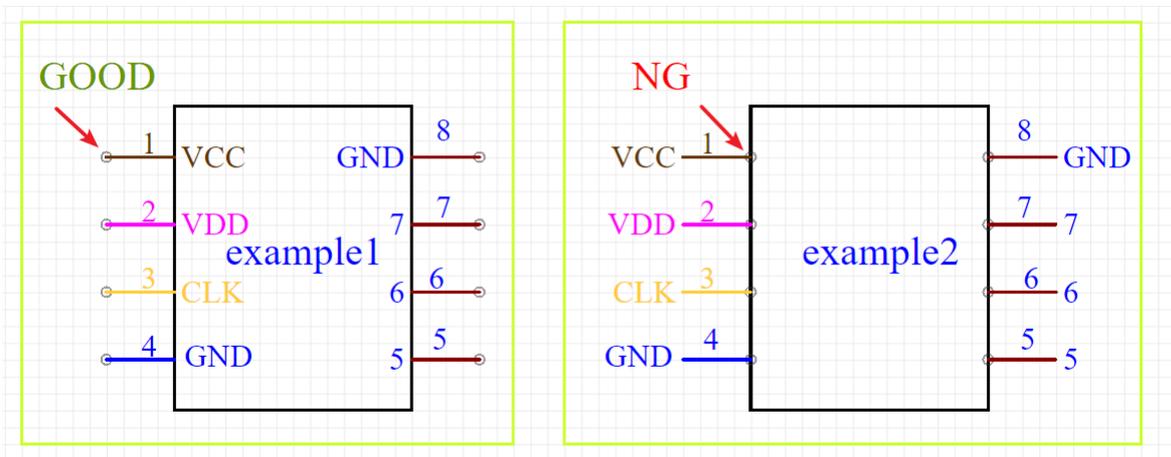


### Pin Orientation

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:

Selected Objects 1	
Pin Attributes	
Orientation	0°
Start X	300
Start Y	90
Length	20
Name	VCC
Number	1
Spice Pin Order	1
Name Display	Yes
Number Display	Yes
Color	#880000
Dot	No
Clock	No
Show	Yes
Electric	Undefined
Font Family	Verdana
Font Size	7pt
Locked	No

**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 pin dot 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.

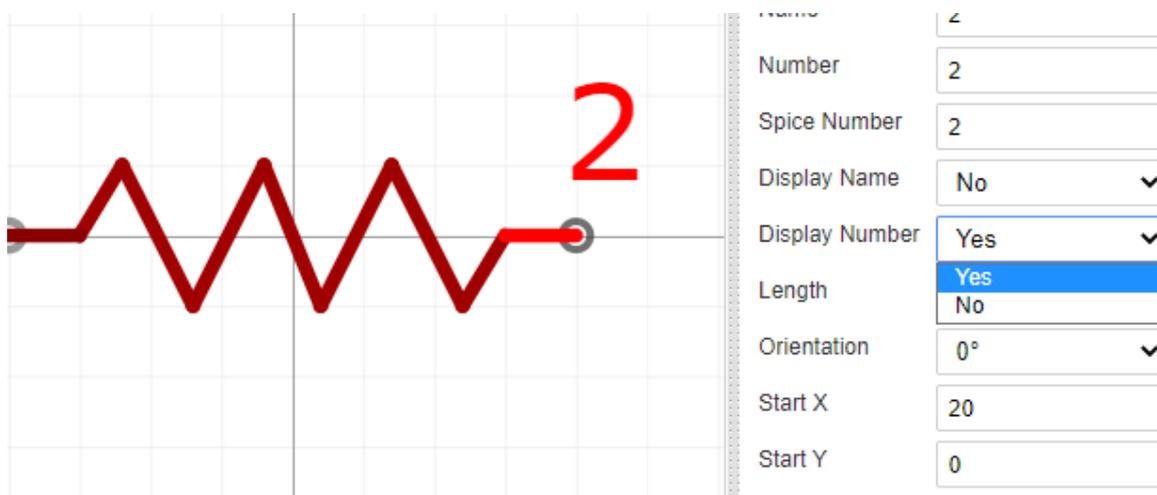
You can adjust the Name or Number position using your mouse but note that rotate and flip applies to the whole pin including the name and pin number; these items cannot be rotated and flipped independently of the pin itself.

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 is 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** [Create Team](#)

Title: **NE555DR**

Manufacturer Part: **NE555DR**

Supplier: **Unknown** Or **Others**

Supplier Part Number: **296-6501-2-ND**

Link: **http://www.ti.com/lit/ds/symlink/ne555.pdf**

Tags: **555 Timer**

Description: **555 Timer**

**Save** **Cancel**

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

Search Libraries

Search Components, Footprints, Modules

Types: **SCH Libs** PCB Libs SCH Modules PCB Modules

Classes: **Personal** LCSC LCSC Assembled System Team Follow

Title(PartNO)	Package
<b>NE555DR</b>	SOIC-8_150MIL

SCH Libs > Personal > Created > 555 > NE555DR

**Edit** **Place** **More** **Cancel**

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**.

Search Libraries

Search Components, Footprints, Modules

Types: **SCH Libs** PCB Libs SCH Modules PCB Modules

Classes: **Personal** LCSC LCSC Assembled System Team Follow

**NE555DR**

**Modify**

**Modify file info**

Title: **NE555DR**

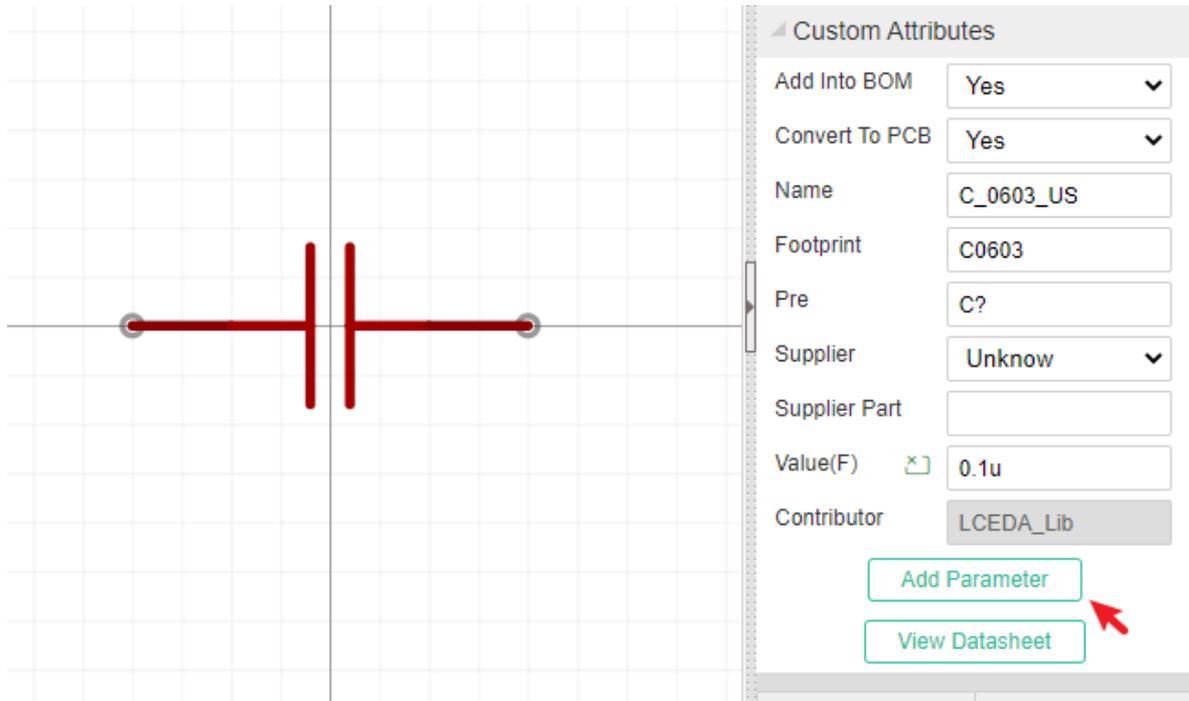
Description:

Tags: **555; Timer**

**OK** **Cancel**

# 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!

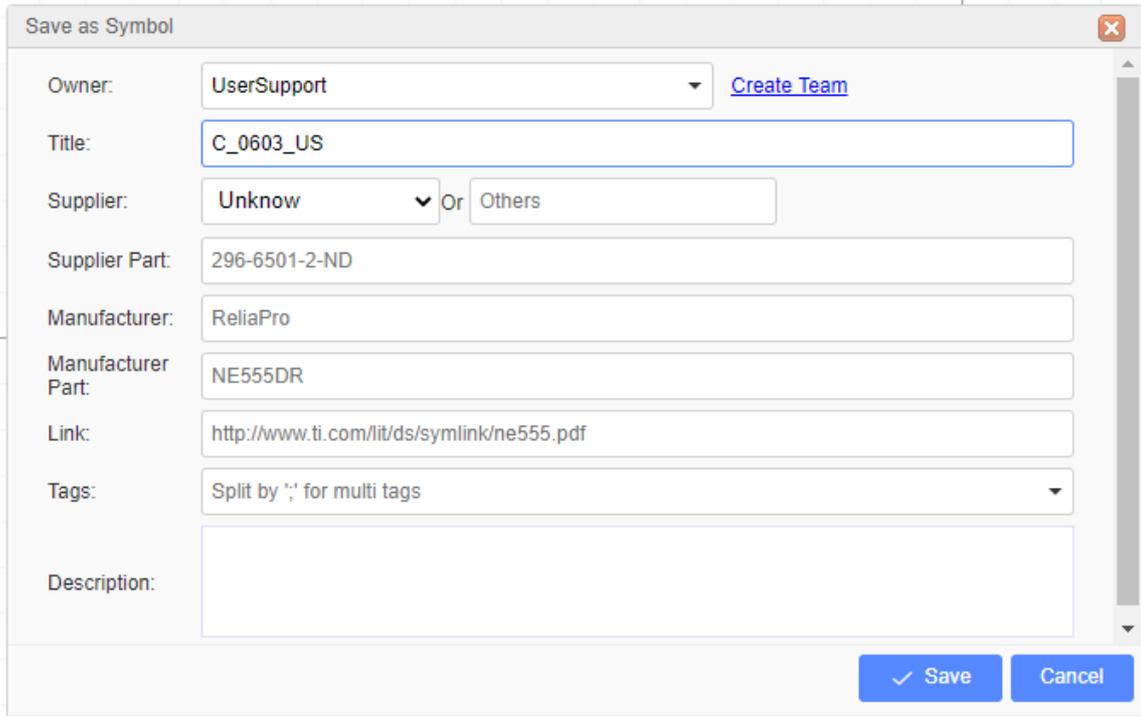
## Show symbol value as component name when place component at schematic

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.



## Personal Libraries

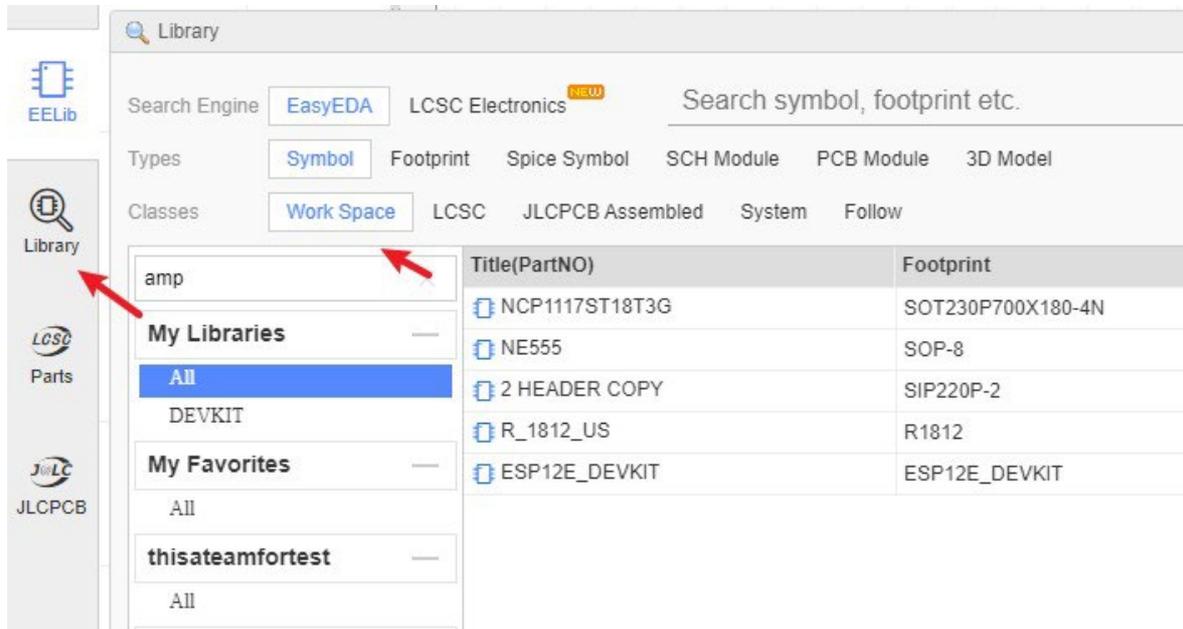
When you **CTRL+S** to save the Schematic Symbol, will pop up a dialog, you can choose this library's owner:



The 'Save as Symbol' dialog box contains the following fields and options:

- Owner: UserSupport (dropdown menu) with a [Create Team](#) link.
- Title: C\_0603\_US (text input field).
- Supplier: Unknow (dropdown menu) with an 'Or' option and an 'Others' text input field.
- Supplier Part: 296-6501-2-ND (text input field).
- Manufacturer: ReliaPro (text input field).
- Manufacturer Part: NE555DR (text input field).
- Link: http://www.ti.com/lit/ds/symlink/ne555.pdf (text input field).
- Tags: Split by ";" for multi tags (dropdown menu).
- Description: (empty text area).
- Buttons: Save (with checkmark icon) and Cancel.

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



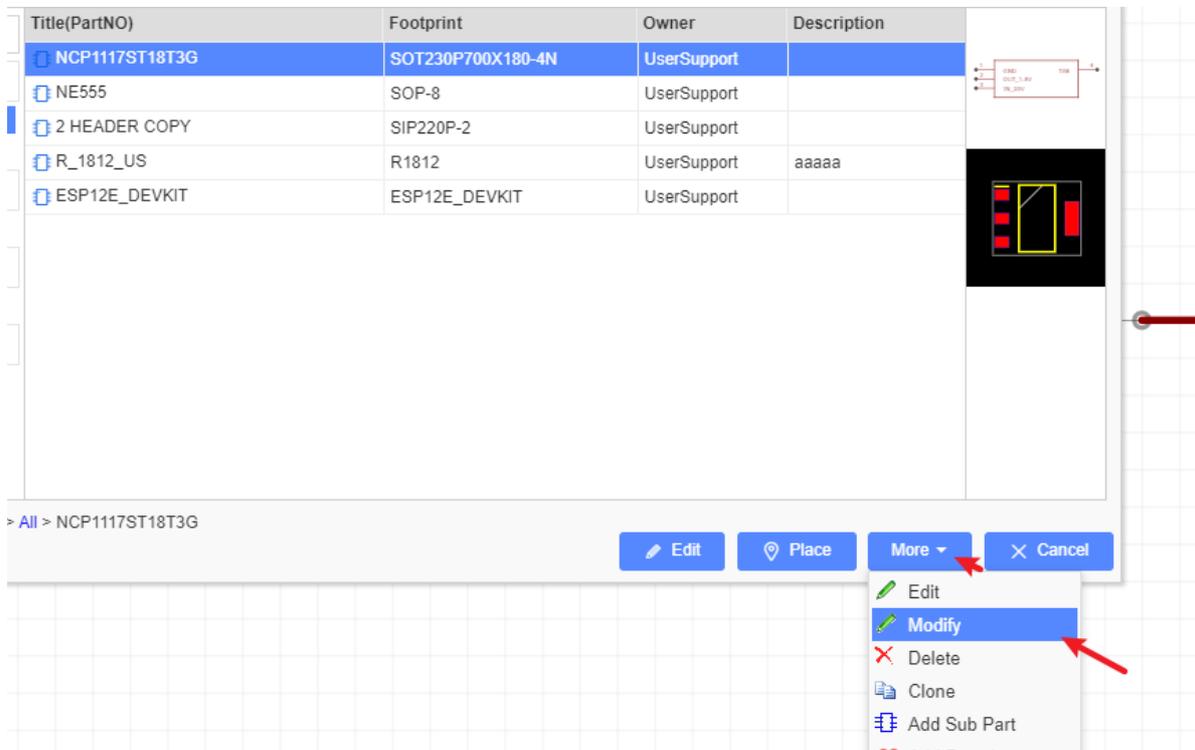
The Library panel shows the following structure:

- Search Engine: EasyEDA, LCSC Electronics (NEW), Search symbol, footprint etc.
- Types: Symbol (selected), Footprint, Spice Symbol, SCH Module, PCB Module, 3D Model
- Classes: Work Space (selected), LCSC, JLCPCB Assembled, System, Follow
- Library list (left): amp, My Libraries (All, DEVKIT), My Favorites (All), thisateamfortest (All). Red arrows point to the 'amp' entry and the 'All' sub-entry under 'My Libraries'.
- Table (right):

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

## Tag

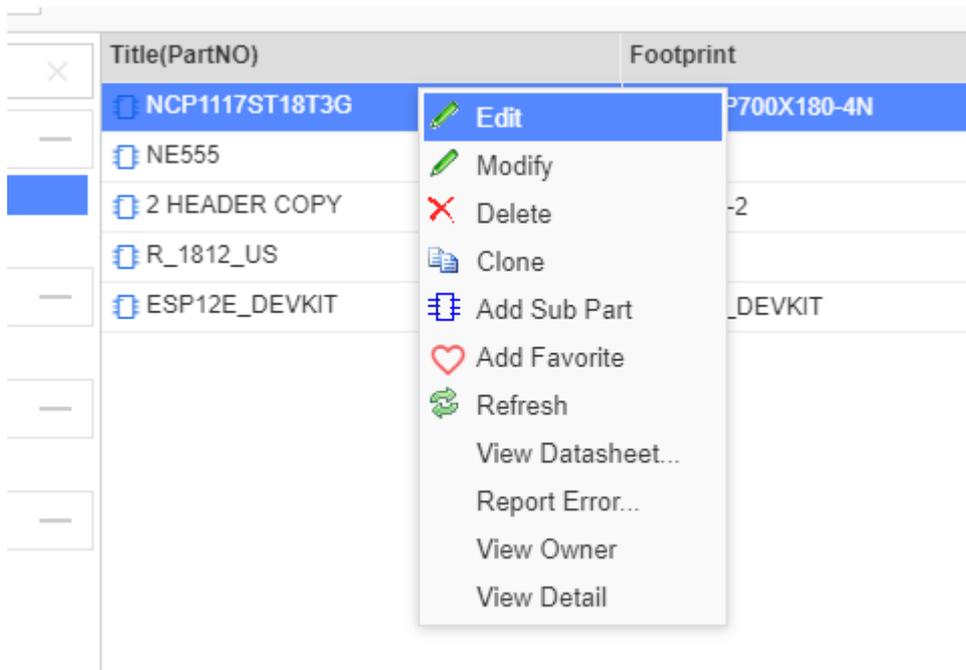
When you select it, right-click it and select the menu "modify", you can add a tag for it.



### Favorite

When you favorite a library, you can find it at **Library > Symbols > Work Spacel > 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.

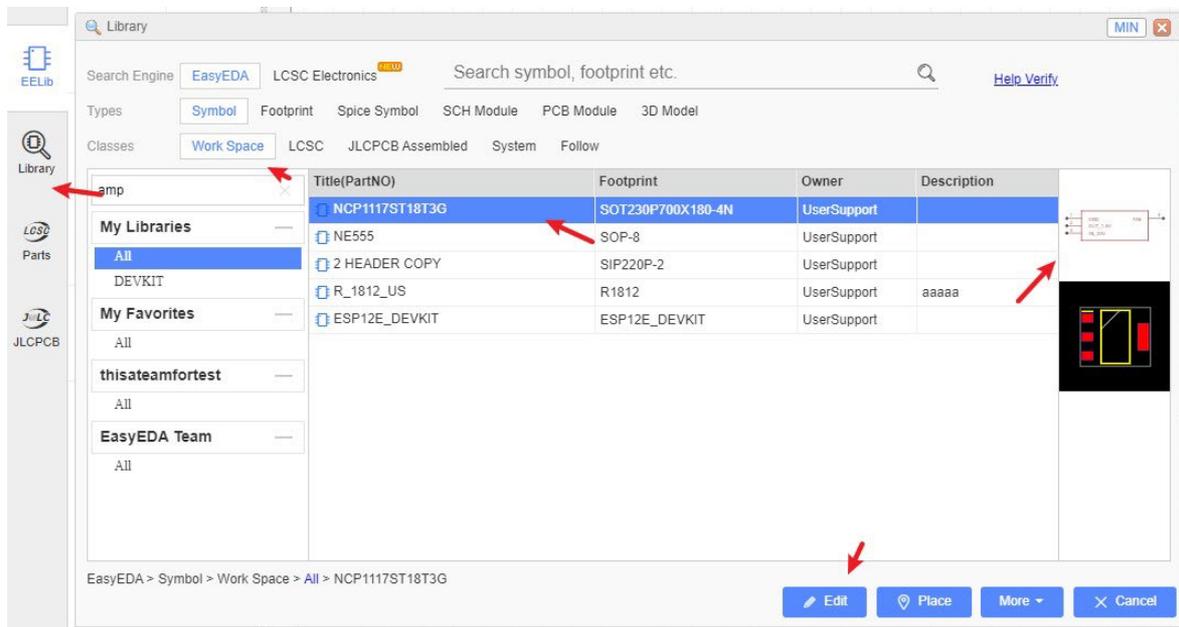


### Edit Symbol in the Library

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

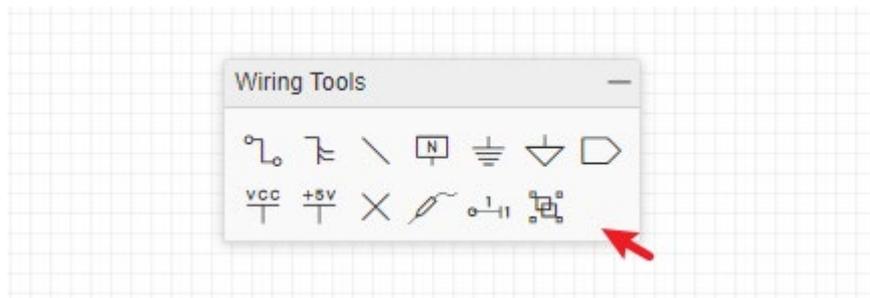


when you finish and save, it will be saved to your personal libraries **Work Space** and become your personal libraries.

## 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:

The image shows a dialog box with the title "Group these items as a SCHLIB/Symbol" and a close button (X) in the top right corner. The dialog contains three input fields: "Prefix:", "Name:", and "Package:". The "Package:" field has a text instruction: "Please assign the package at right panel after created the symbol." Below the input fields, there is a note: "Keep these fields empty if you just want to build a symbol". At the bottom of the dialog, there are three buttons: "OK", "Cancel", and a help icon (question mark).

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.

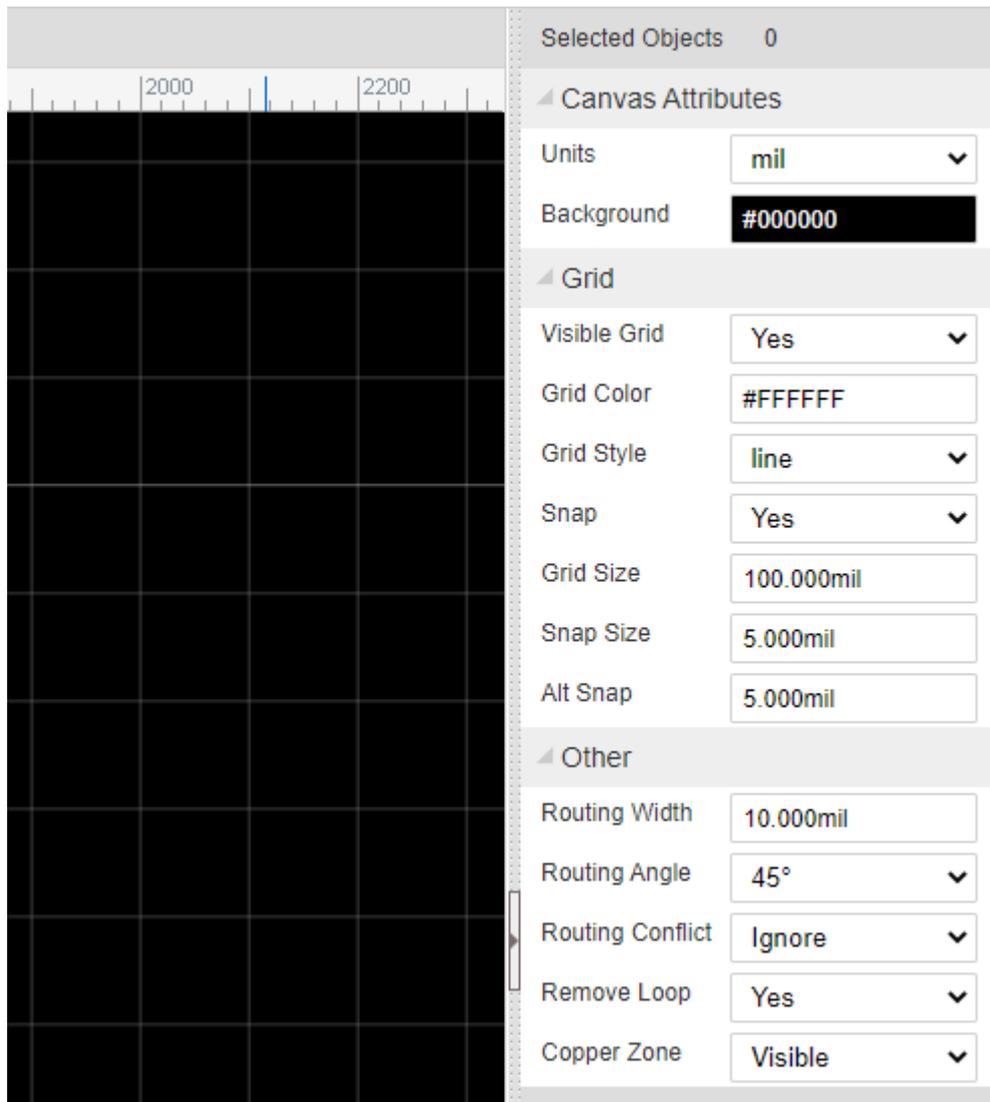
# Experiment no: 15

## Experiment Title: PCB Design in EasyEDA

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 around 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.

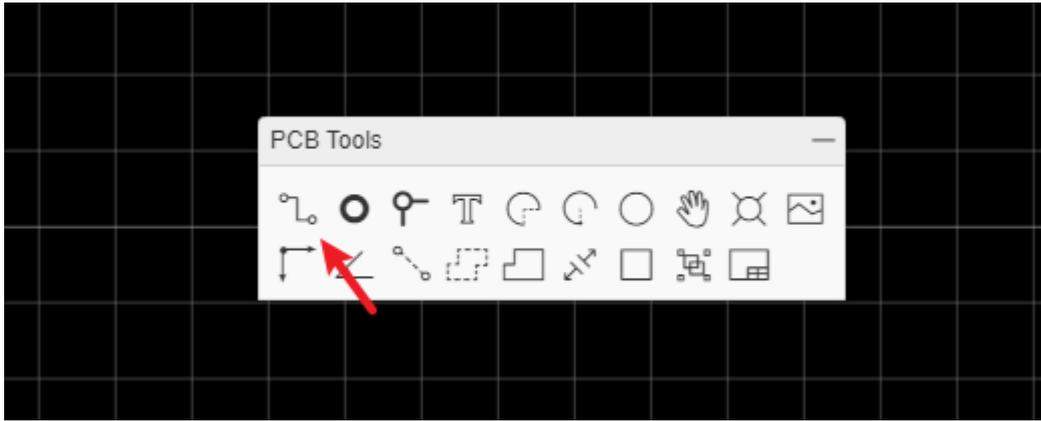
Such as: Track, Pad, Via, Text, Arc, Circle, Move, Hole, Image, Canvas Origin, Connect Pad to Pad, Copper Area, Solid Region, Measure/Dimension, Rect, Group/Ungroup. etc.



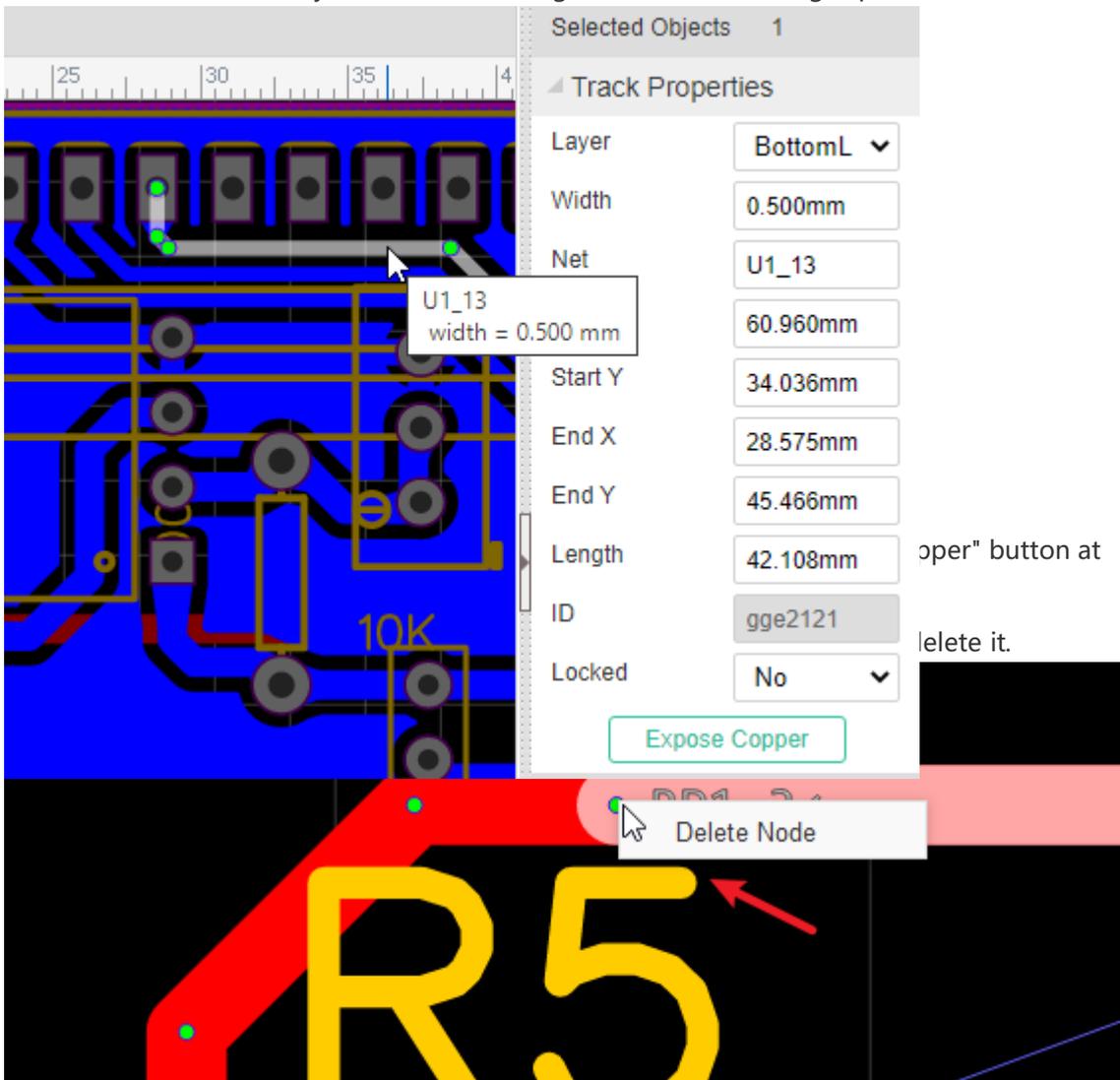
### Track

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.

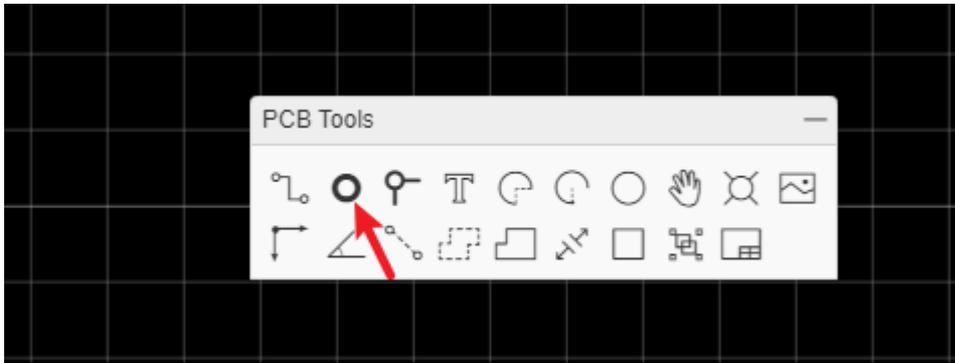


when select the point to point separated tracks, you can convert them as Solid Region or continus track at right-click menu.

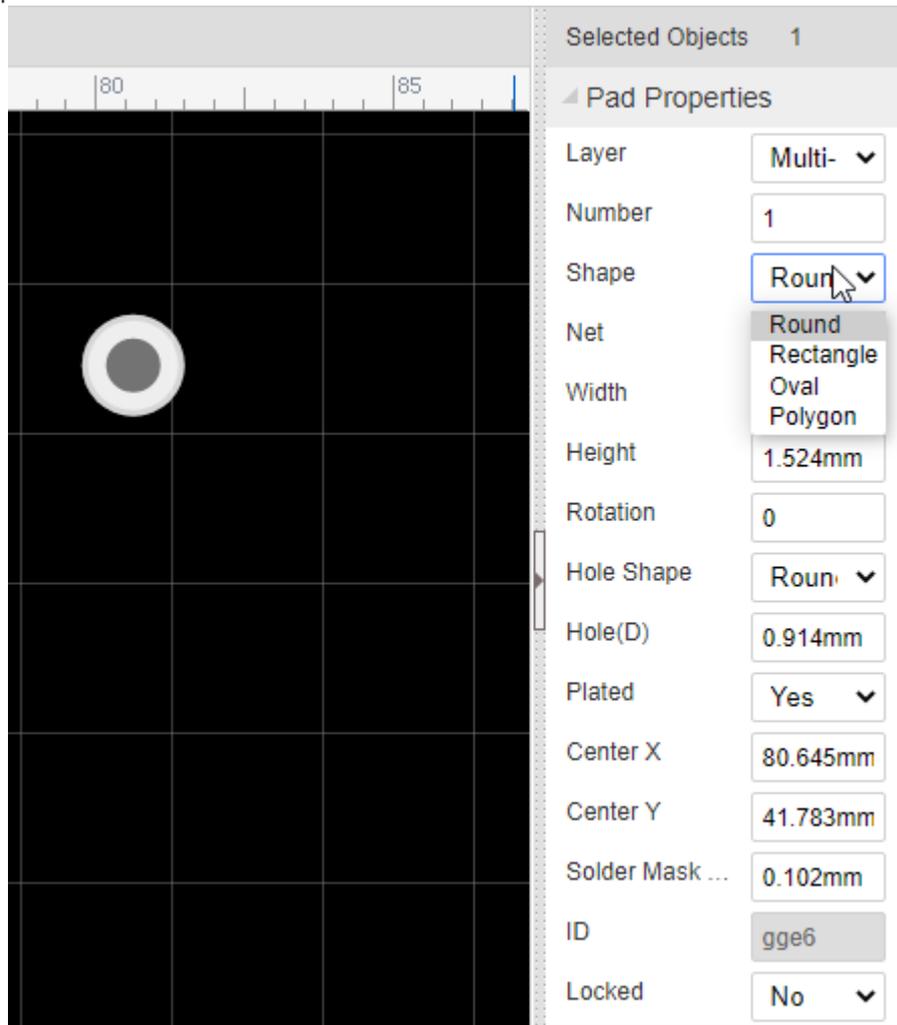
The more information of routing, please refer at [PCB: Route Tracks](#)

## 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 your 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

The screenshot shows the 'Edit Points' dialog box in a PCB design software. The dialog is divided into three main sections:

- Diagram:** A 2D coordinate system with a red dashed crosshair. A polygon pad is shown with a central circular hole. A red arrow points to the rightmost vertex of the polygon.
- Table:** A table with columns 'No.', 'X', 'Y', and 'Action'. It lists the coordinates for the five vertices of the polygon.
- Pad Properties:** A sidebar on the right containing various settings for the selected polygon pad.

No.	X	Y	Action
Hole	0.000	0.000	Hole is draggable
0	-0.762	0.762	+↑ +↓ -
1	0.762	0.762	+↑ +↓ -
2	1	0	+↑ +↓ -
3	0.762	-0.762	+↑ +↓ -
4	-0.762	-0.762	+↑ +↓ -

**Pad Properties:**

- Layer: Multi
- Number: 1
- Shape: Polyg
- Net: (empty)
- Rotation: 0
- Hole Shape: Roun
- Hole(D): 0.814mm
- Plated: Yes
- Center X: 83.820mm
- Center Y: 42.672mm
- Solder Mask ...: 0.102mm
- ID: 00e6
- Locked: No

**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 multi-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.

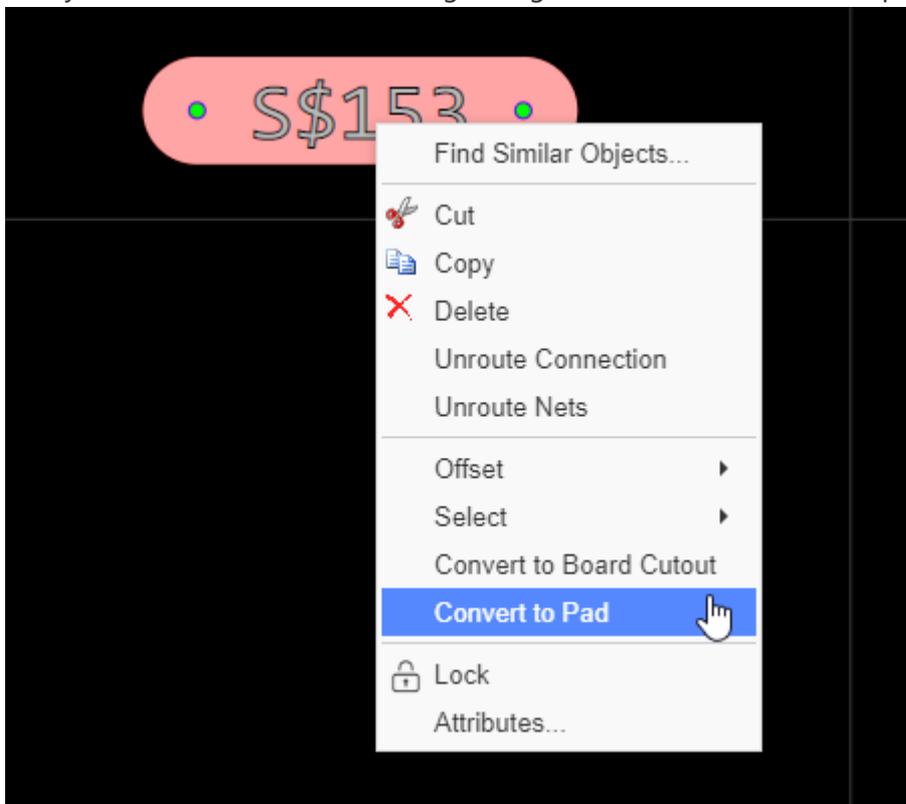
**Center-X and Center-Y:** using these two attributes, you can set the pad's position with more precision, compared to using the mouse.

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

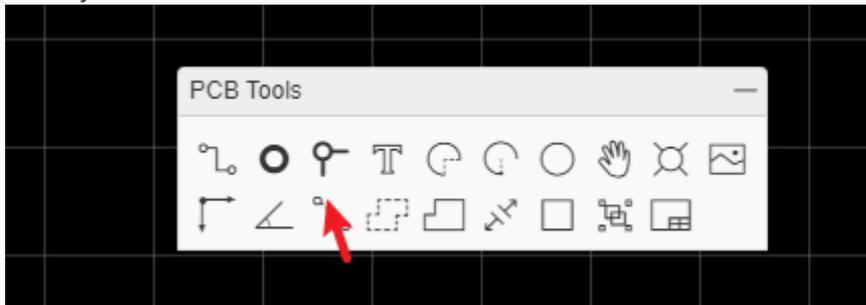
**Paste Mask Expansion:** For single layer pad. This property affects the size of the tin area on the plate of the steel mesh. If you want to set a pad that is not open in the steel mesh, you can set the value to be negative, which is usually larger than the diagonal of the pad.

**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.

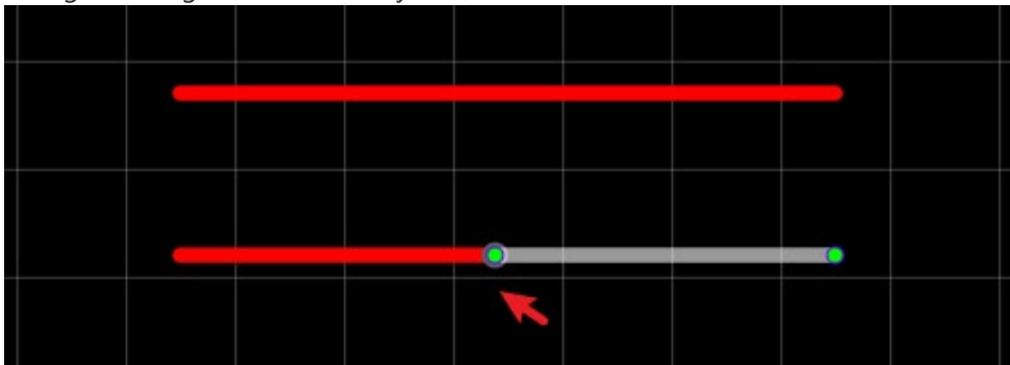


When you want to lay a multilayer PCB, you need to add Vias for nets getting through layer and layer.



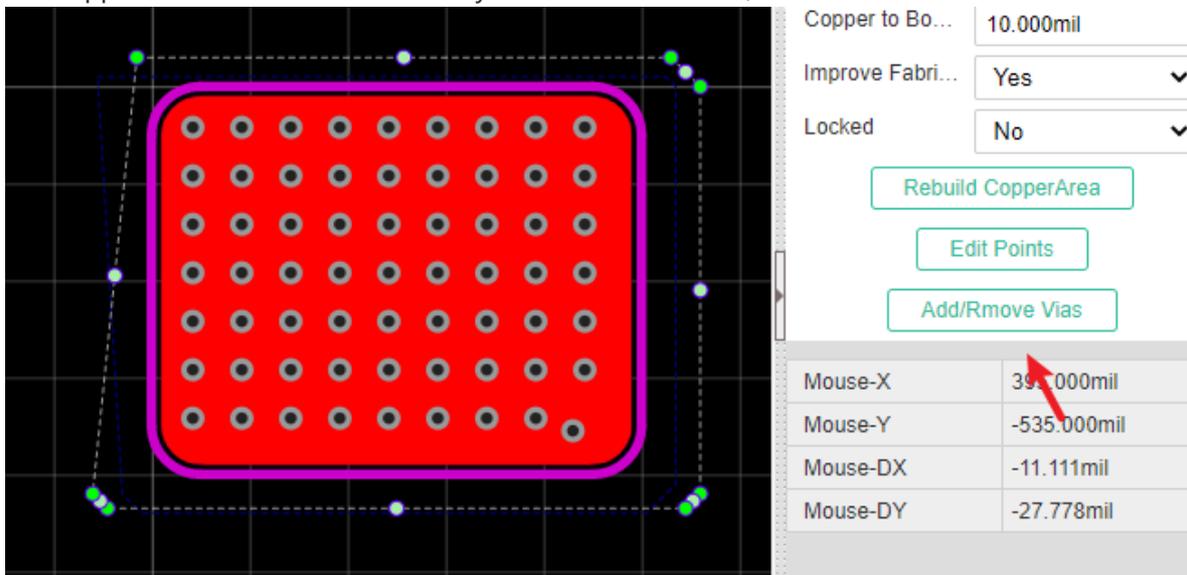
### Place a Via on a Track

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.

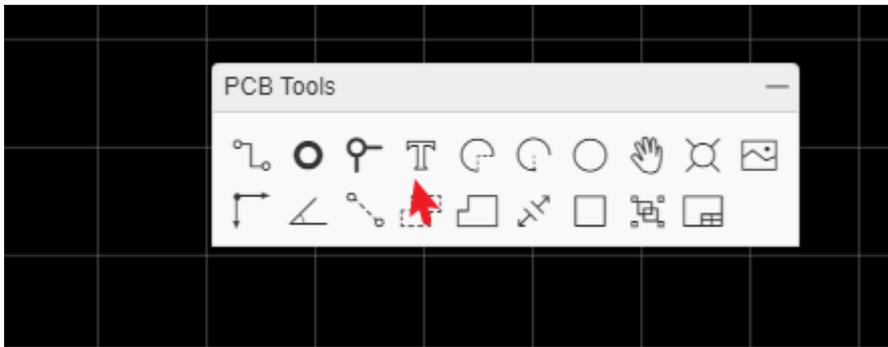


### Notice:

- 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 [free fonts:](http://www.fontspace.com/)



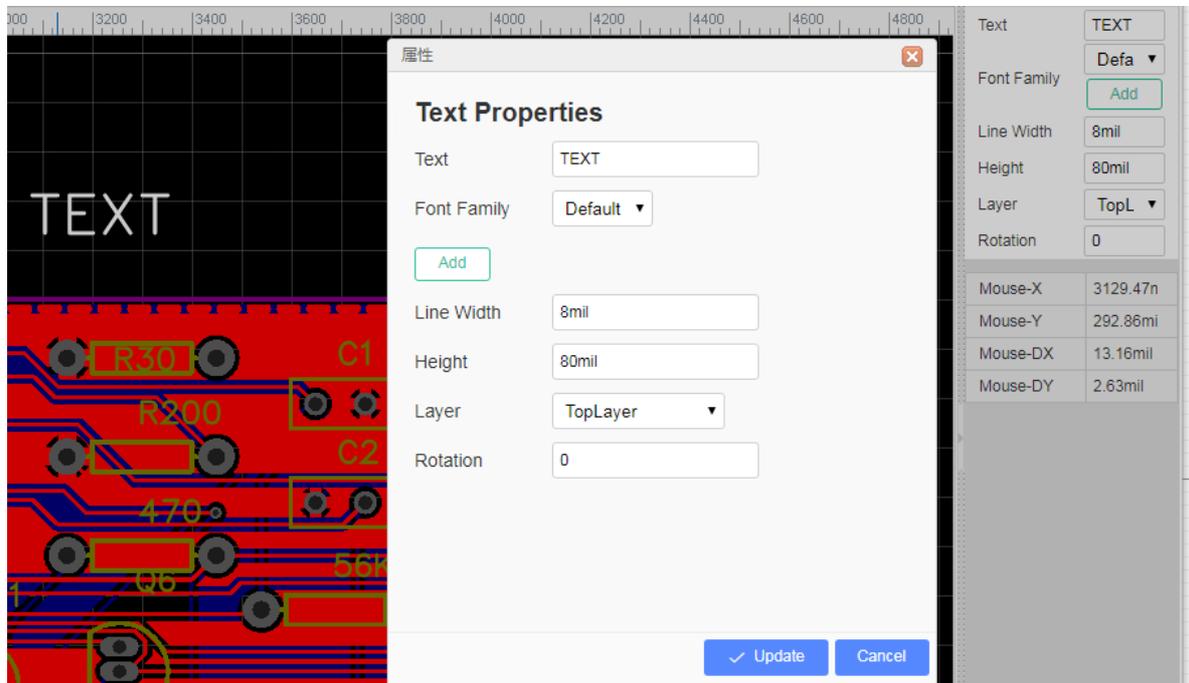
if you need Japanese or Korean you can use [Google Noto fonts](#)

The editor including fonts are:

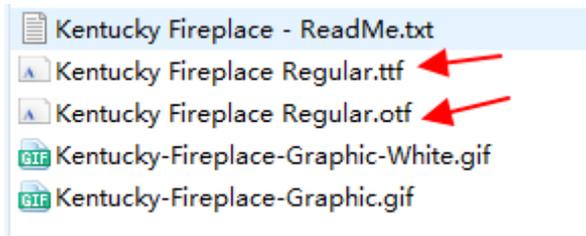
(NotoSansCJKsc-DemiLight)[<https://github.com/googlefonts/noto-cjk/blob/main/Sans/OTF/SimplifiedChinese/NotoSansCJKsc-DemiLight.otf>]

(NotoSerifCJKsc-Medium)[<https://github.com/googlefonts/noto-cjk/blob/main/Serif/OTF/SimplifiedChinese/NotoSerifCJKsc-Medium.otf>]

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`.

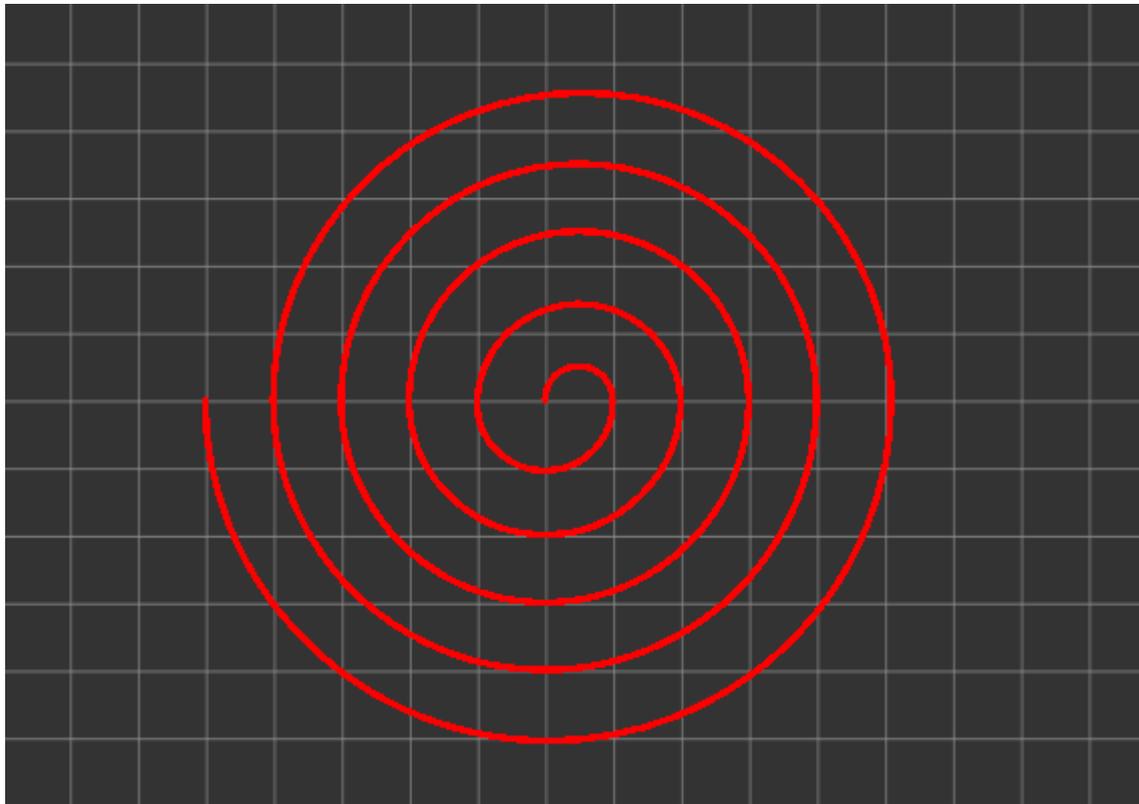
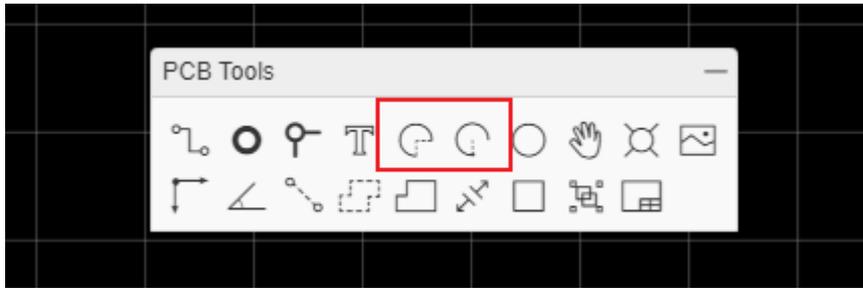


So you can add any fonts by yourself. EasyEDA doesn't cache the font on our server, so if you close the editor, you need to add the font again by yourself.

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

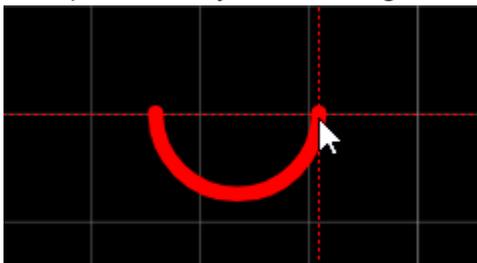
## Arc

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:

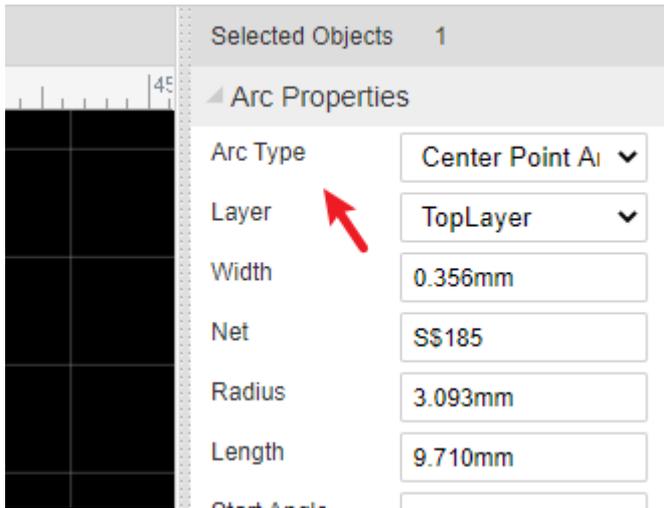
- Start point fixed, you can change the end point position and radius.



- Center point fixed, you can change the radius.

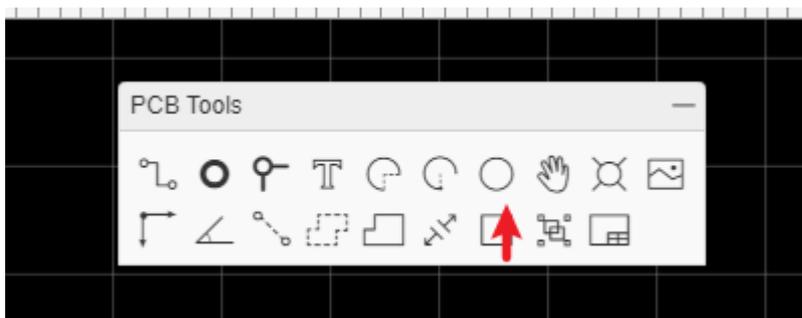


Select the arc, you can change the arc type at property panel, different arc type has different drag behavior.



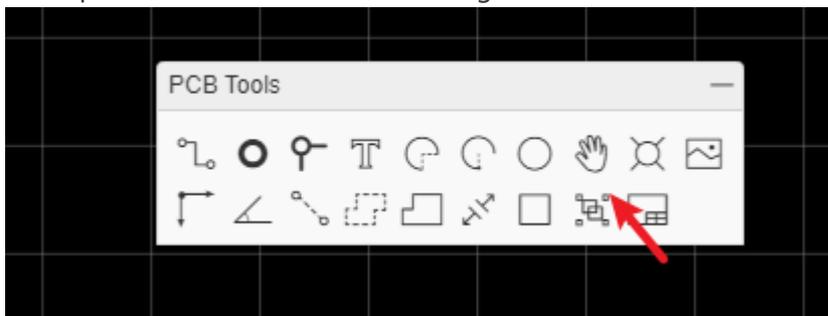
## Circle

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



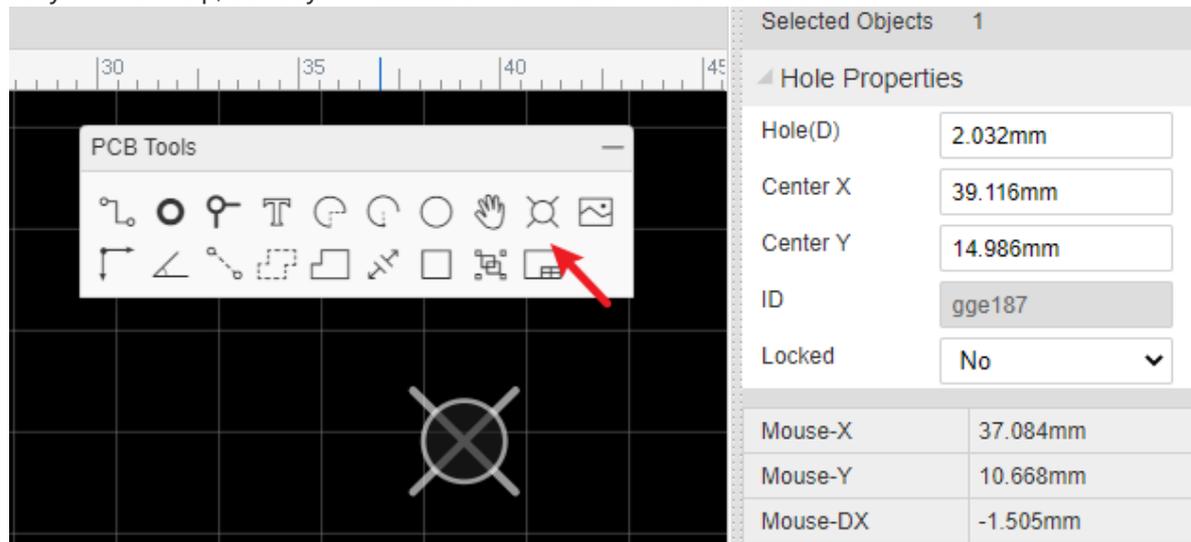
## Move

This option is same as schematic's drag.



## Hole

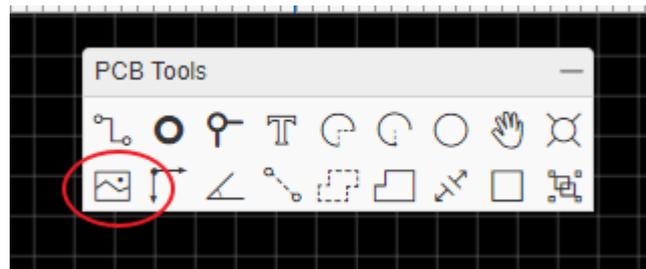
There were lots of users that didn't know how to use PAD or VIA as a HOLE, they asked EasyEDA for help, so EasyEDA added a HOLE TOOL in the PCB toolbar.



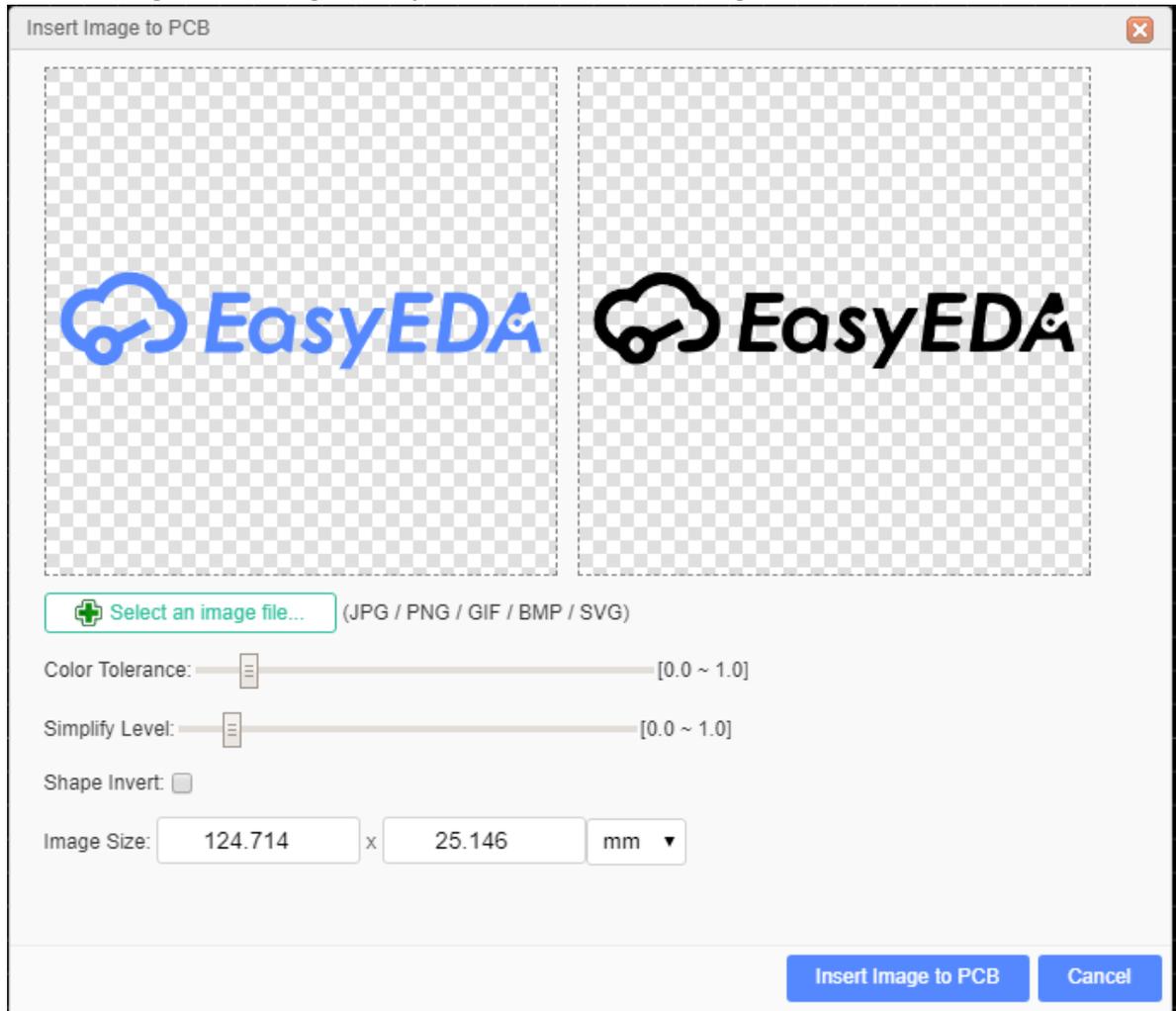
And if you want to create the slot hole, you can use solid region(Type: NPTH), or route a track, and then right-click the "Convert to NPTH" menu.

## 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.

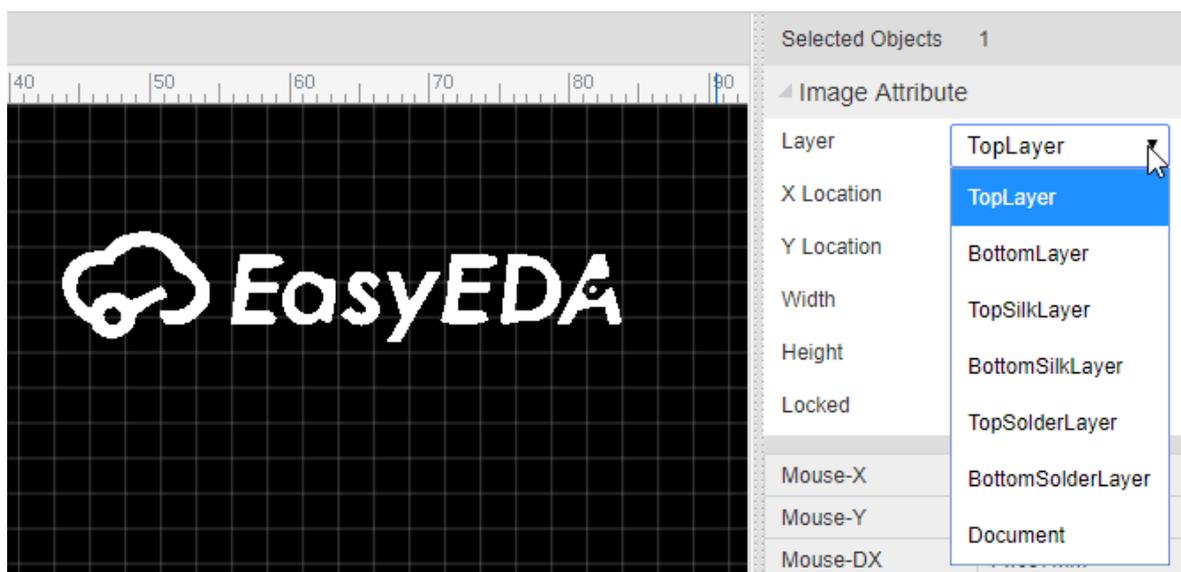


SVG. Unlike some other EDA tools which only support a Monochrome Bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

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.



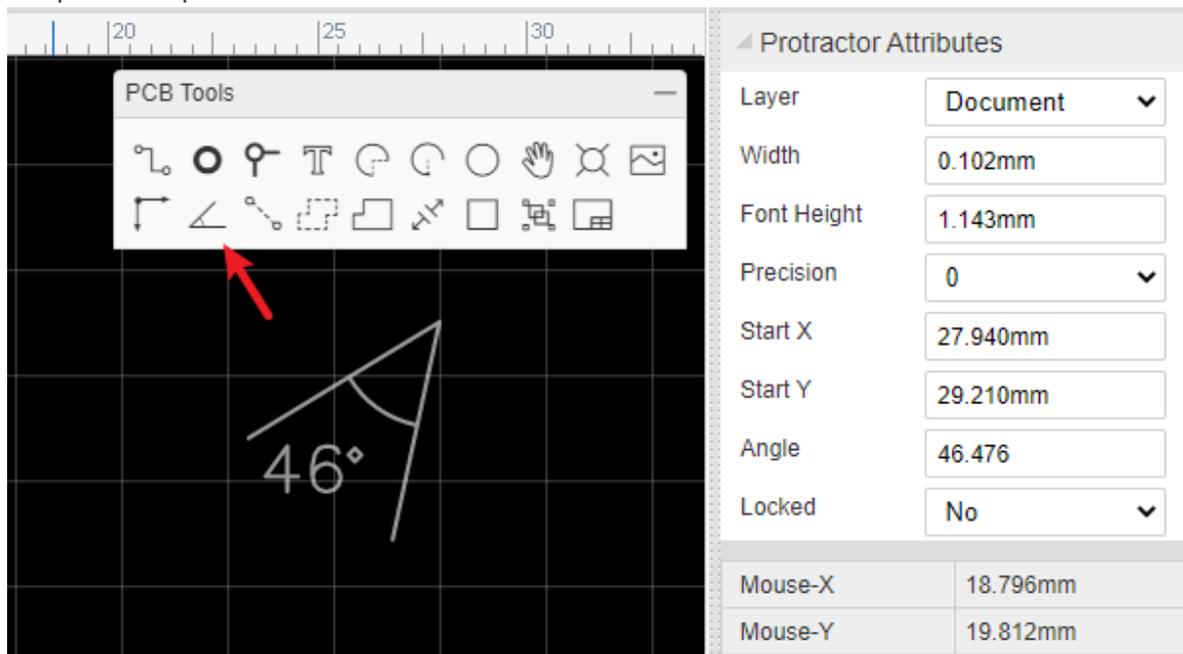
## Canvas Origin

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



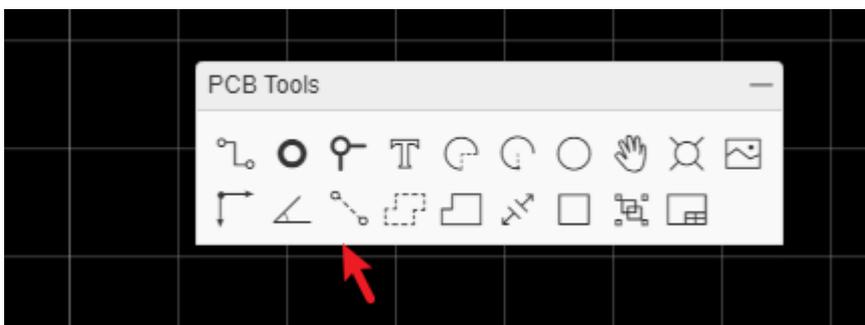
## Protractor

We provide a protractor for PCB tools.



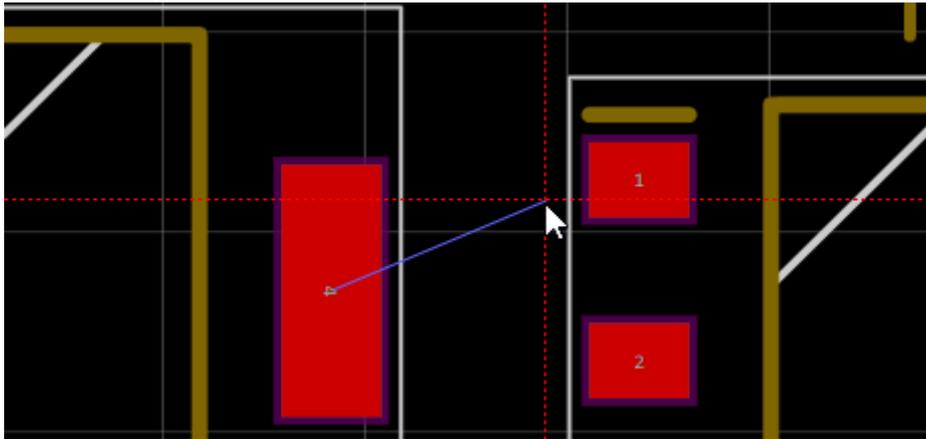
## Connect Pad to Pad

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.

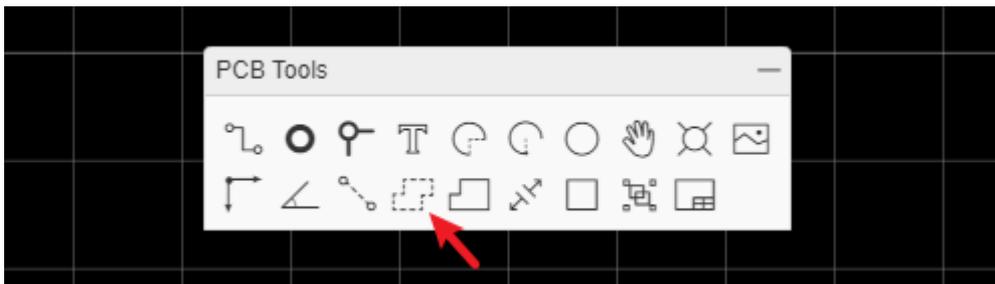
You could also do this by setting net names for all the pads: if the two pads are given the same net name then EasyEDA will understand that they are connected together and will automatically create a ratline between them.



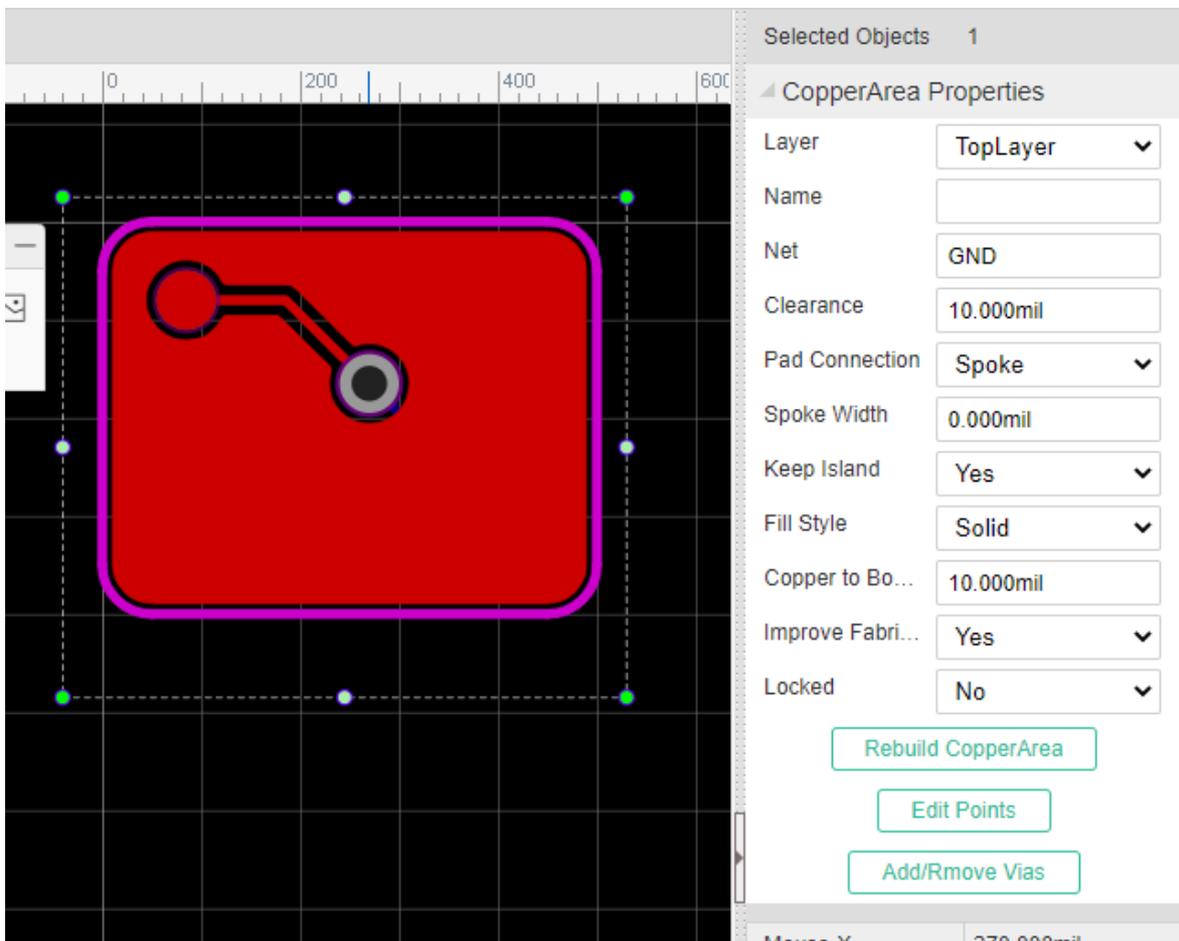
Or you can set these two pads with the same net name at the right panel Pad Properties after you click the pad.

## 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.



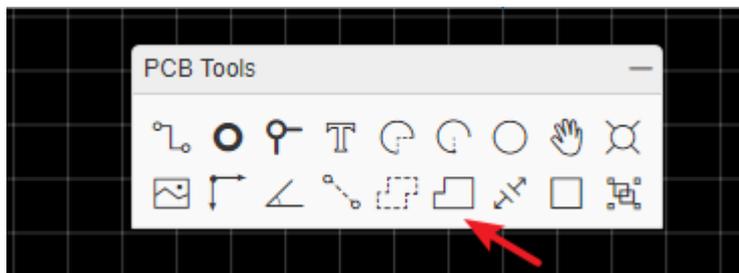
When selecting a copper area, you can find its attributes from the right hand **Properties** panels.



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.



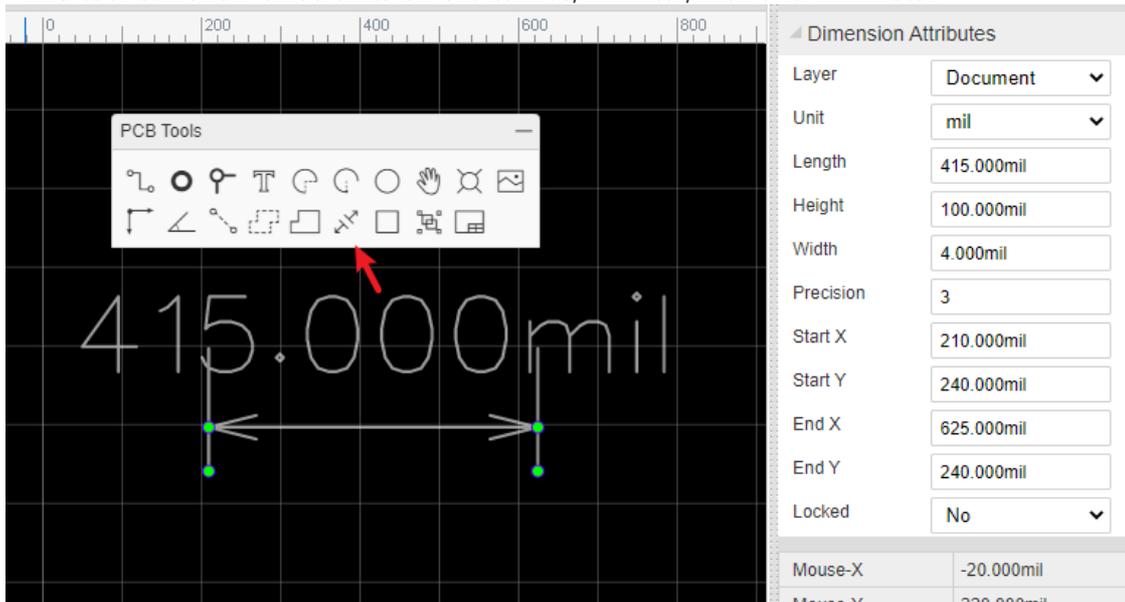
The more information please refer at [PCB: Solid Region](#)

## Measure/Dimension

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

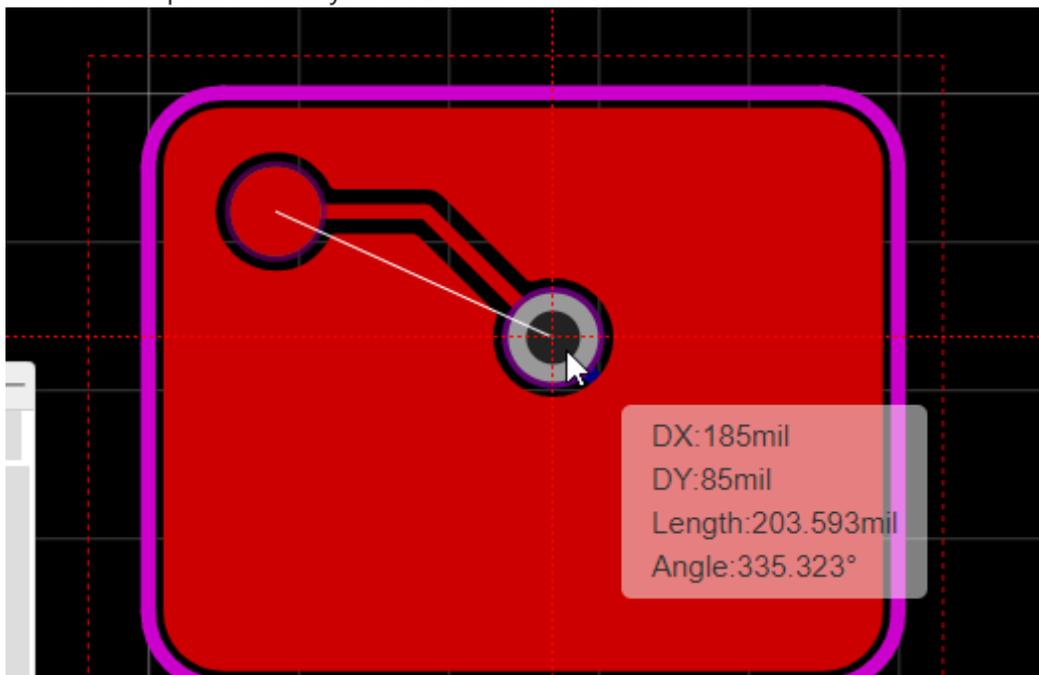
1. Dimension tool in the PCB Tools palette:

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



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.

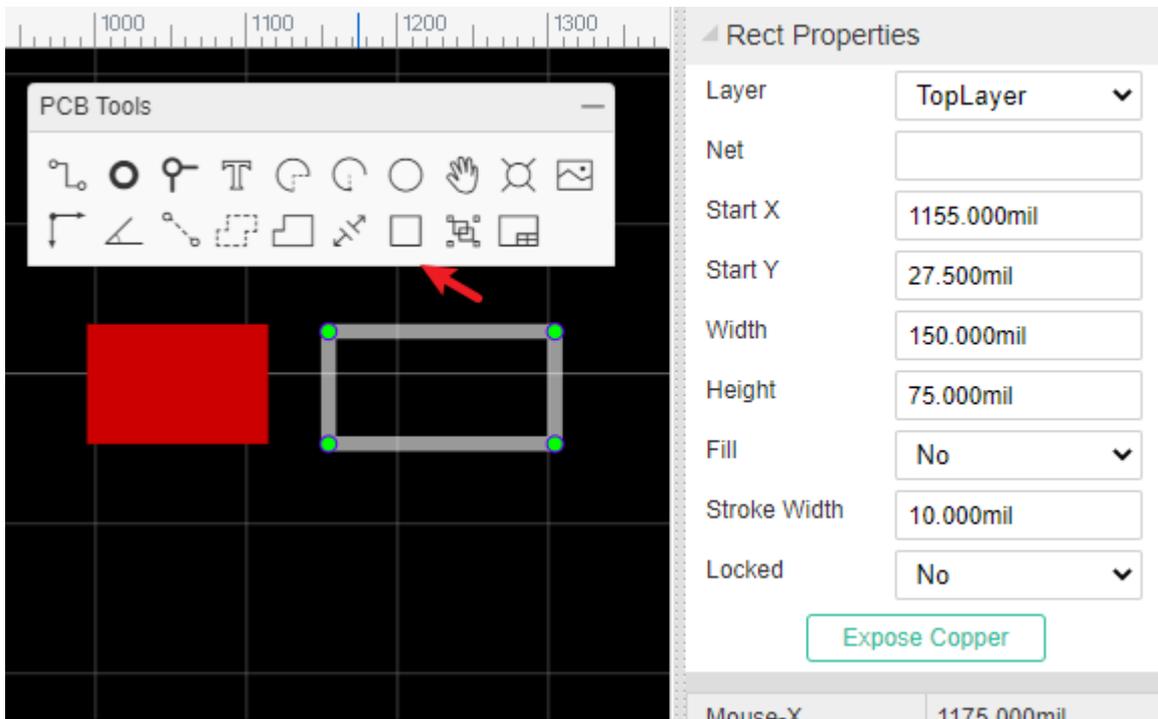


**Tips:**

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

## Rect

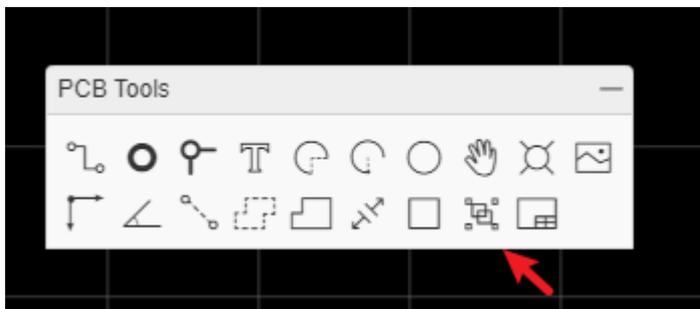
It looks like a Solid Region, but it can't be set Nets and you can't set the Layer as NTPH.



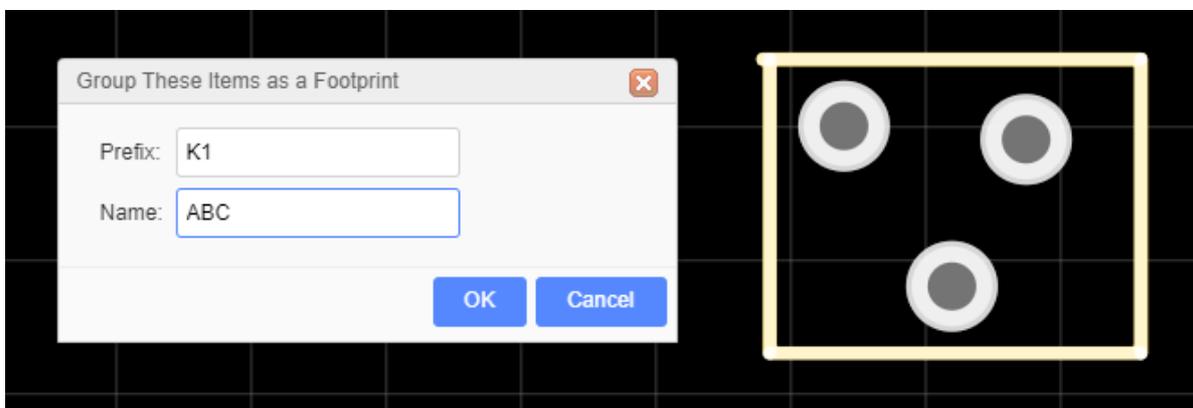
The rect doesn't rotate, you can change its width and height.

## 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

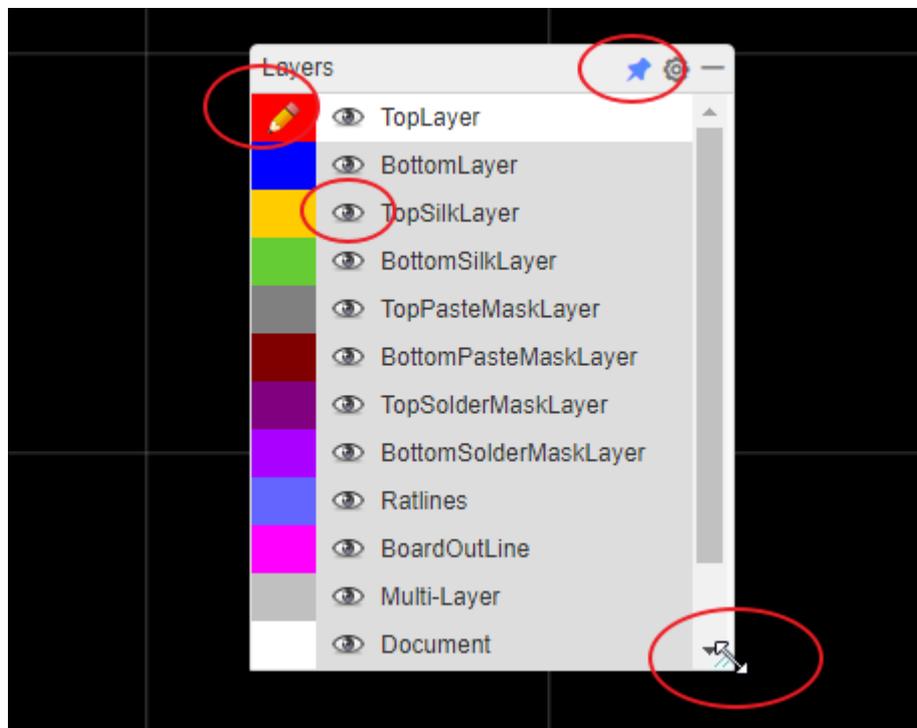
---

### 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:

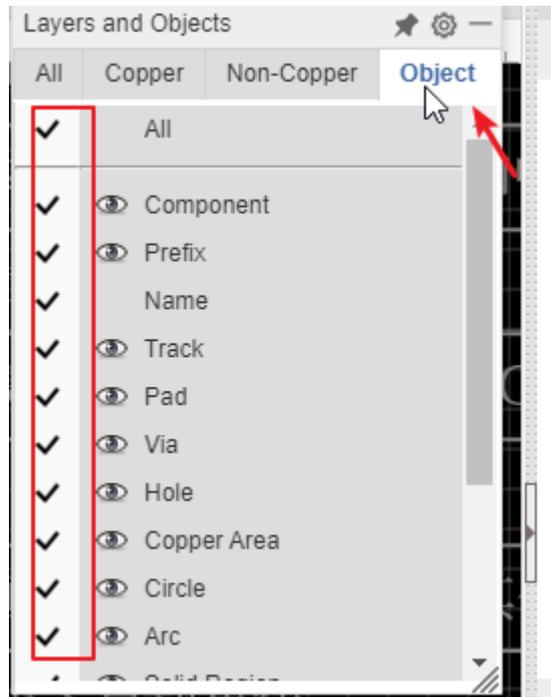
- **T**: Top Layer is active
- **B**: Bottom Layer
- **1**: Inner1 Layer
- **2**: Inner2 Layer
- **3**: Inner3 Layer
- **4**: Inner4 Layer

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

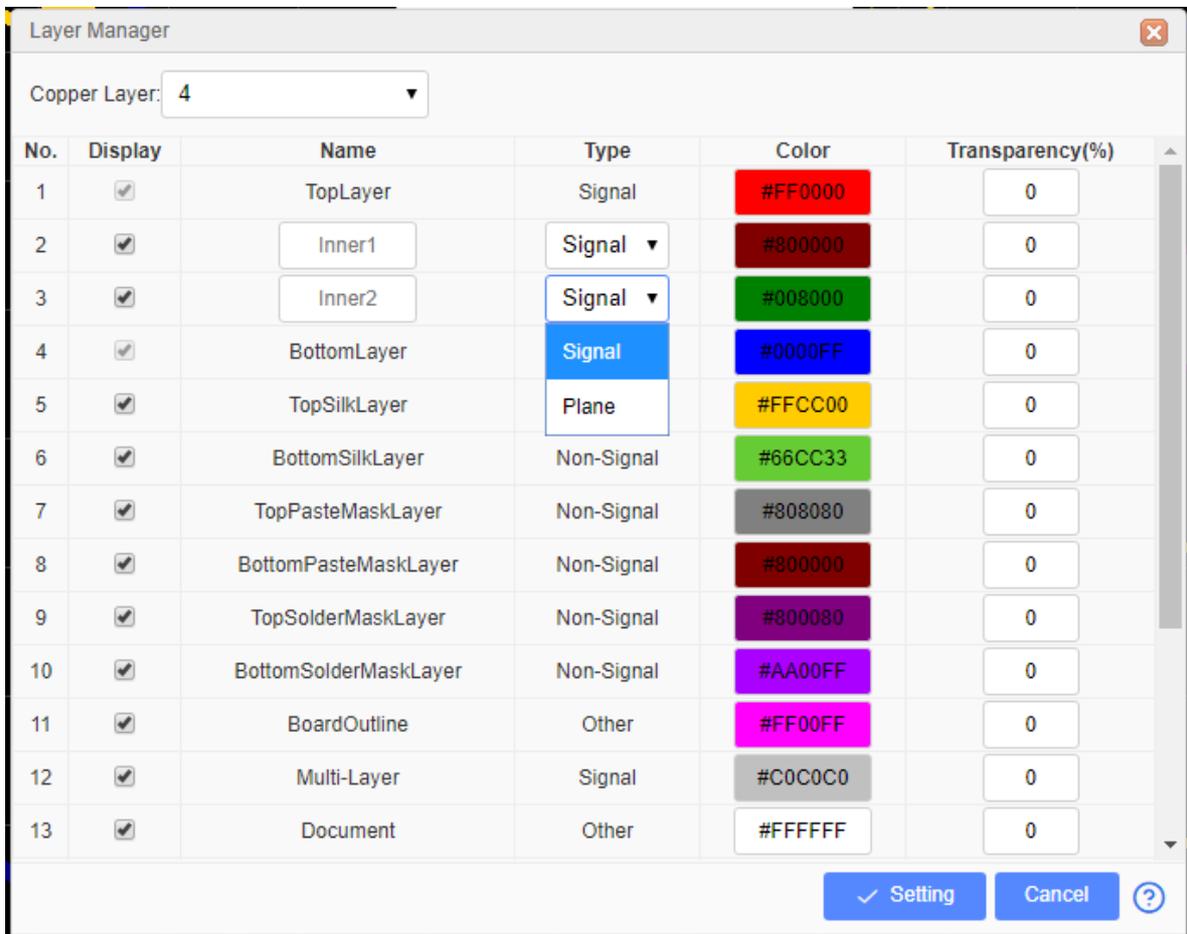
---

### 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:



The Layer Manager setting only works for the current editing PCB.

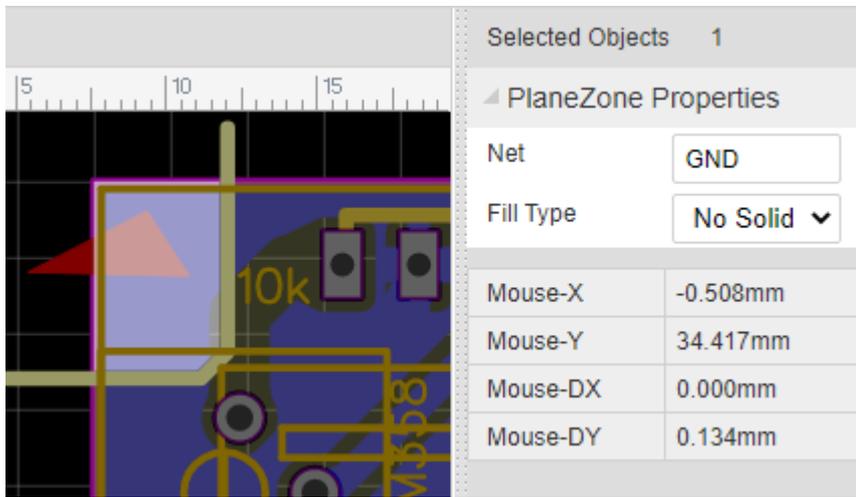
**Copper Layer:** The copper layer of your PCB. EasyEDA support 34 copper layers. The more copper layers the PCB will be more expensive. The TopLayer and BottomLayer is default layer, can not be disable. If you want change the copper layers from 4 to 2, you must delete the inner layers objects first.

**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 its 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 Defination:**

- **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. If the mechanical layer has closed wires, JLCPCB will give priority to using the mechanical layer as the shape of the board when producing the board. If there is no outer frame of the mechanical layer, GKO will be used as the frame (historical influence of Altium file). It is necessary to pay attention to the use of the mechanical layer in the design.

- **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 display.

## Layout Single Layer PCB

The PCB copper layers of EasyEDA are double, EasyEDA doesn't support layout a single 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:

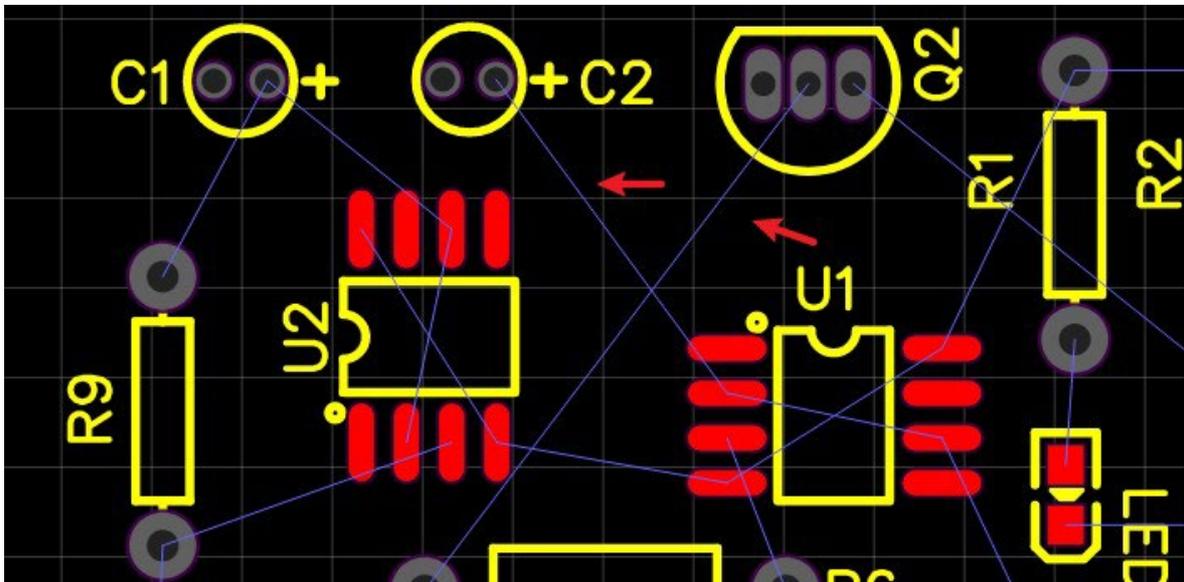
- Design your PCB at one side, if other side has pads etc, you don't need to deal with them.
- Generate the Gerber.
- Add the comment for mention that you need to order the single layer PCB when order the PCB.

---

## Ratline

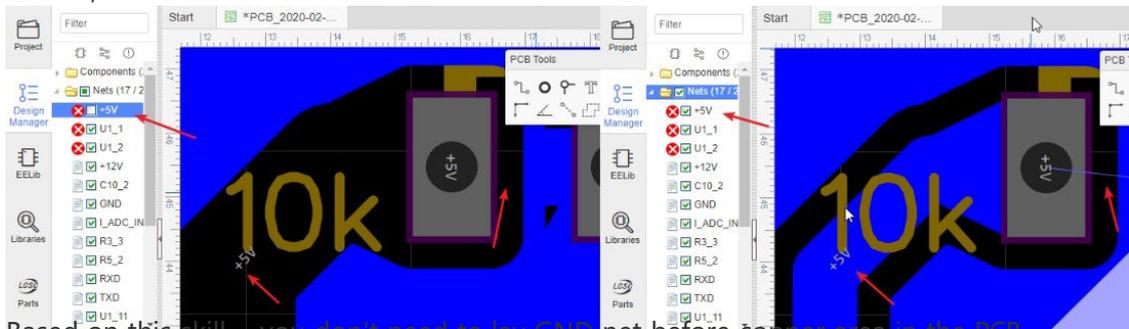
---

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.



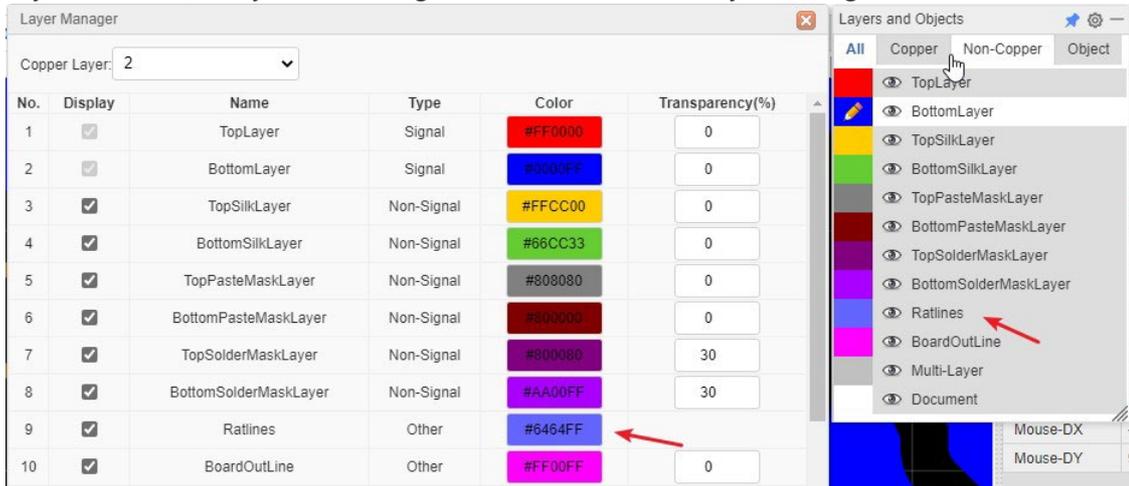
1. If you want one ratline do not show on the PCB editor, you can deselect the net in the design manager, as below deselect +5V :

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.



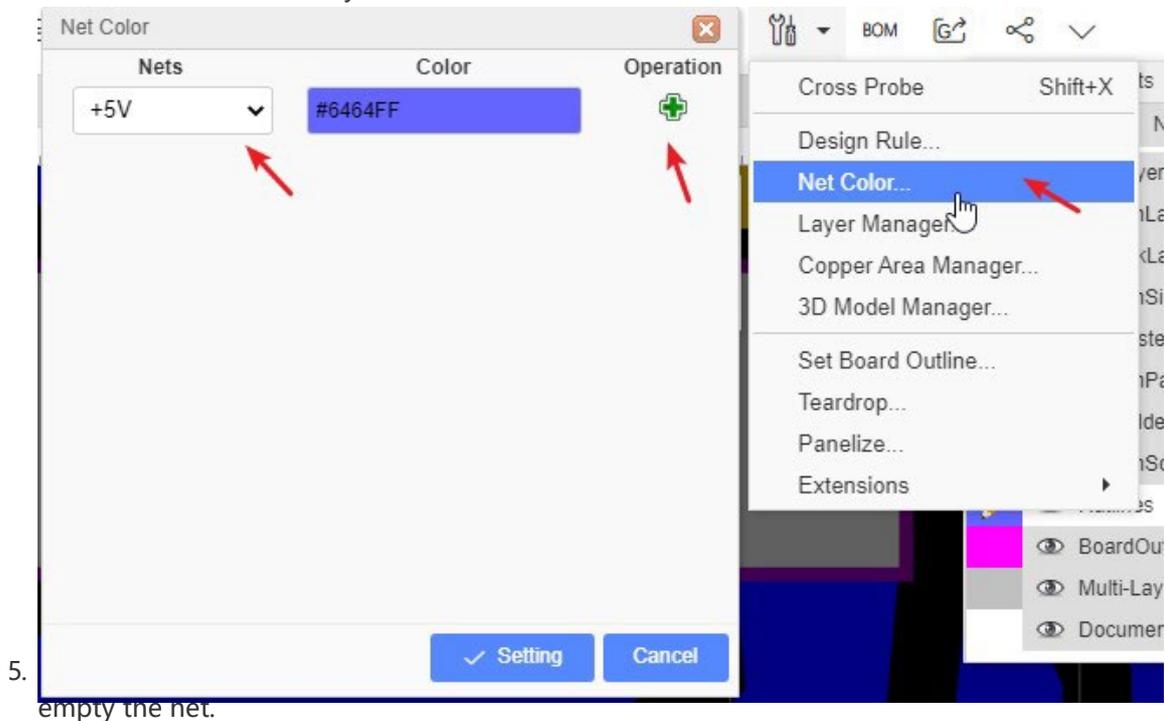
Based on this skill, you don't need to lay GND net before copper area in the PCB.

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.



3. If you want to highlight 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. empty the net.

---

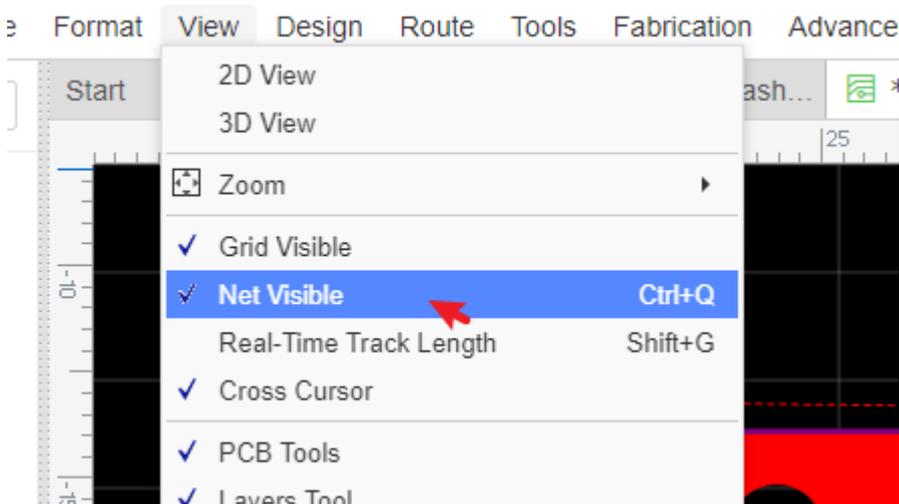
## PCB Net

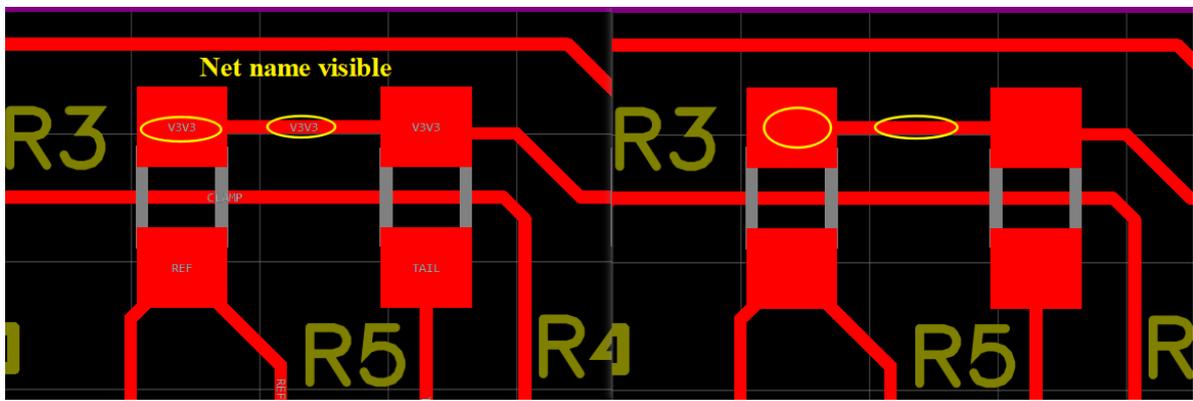
---

### 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:

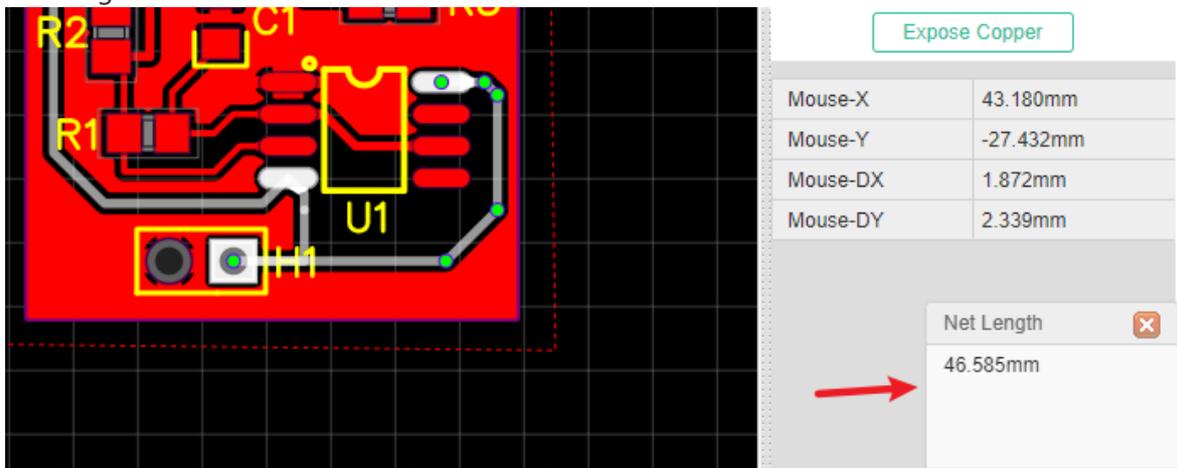
**Top Menu > View > PCB Net Visible**, or press hotkey **CTR+Q** .





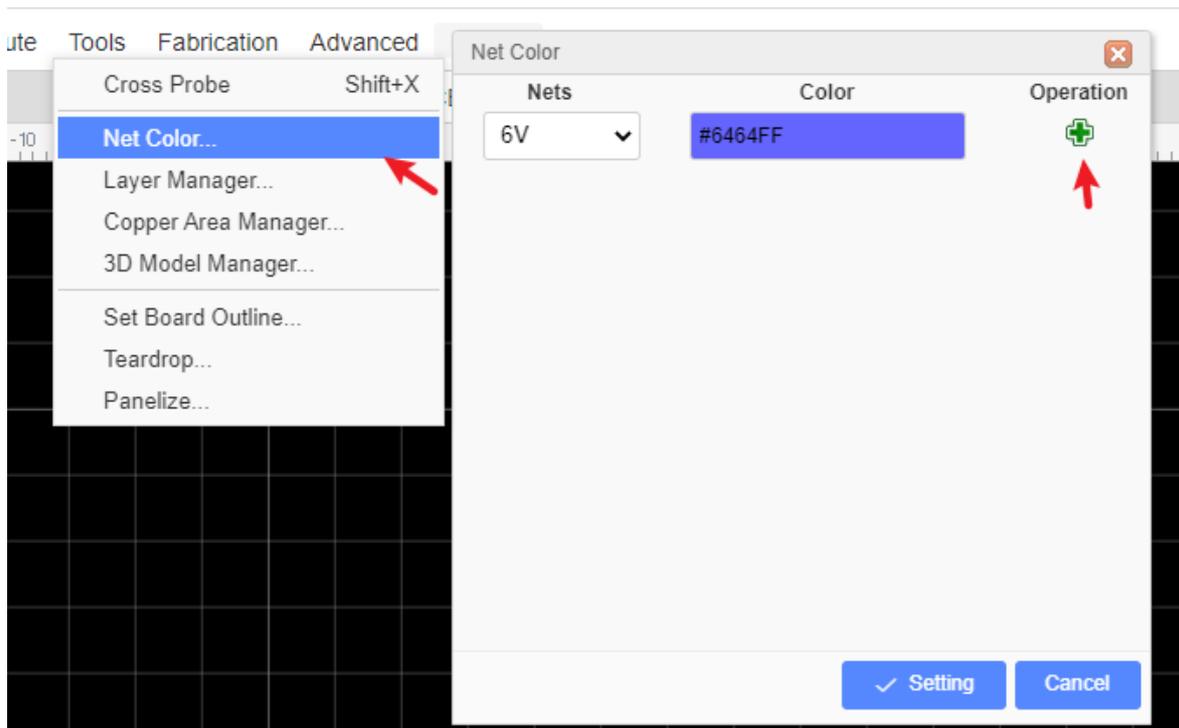
## Net Length

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.



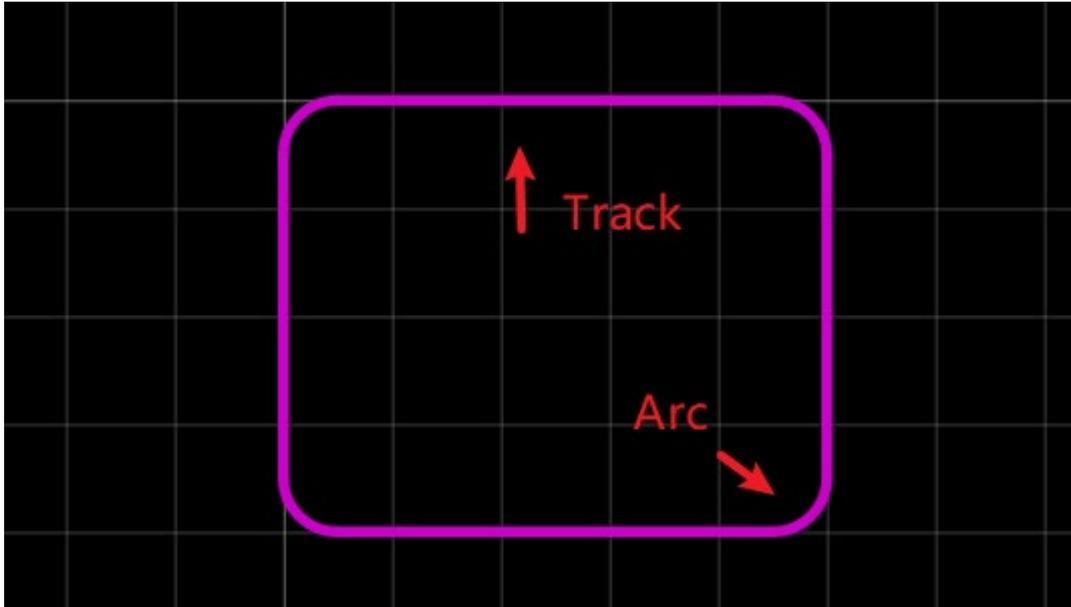
When you set a color for a net, you need to click the + button to make it work.

---

## Board Outline

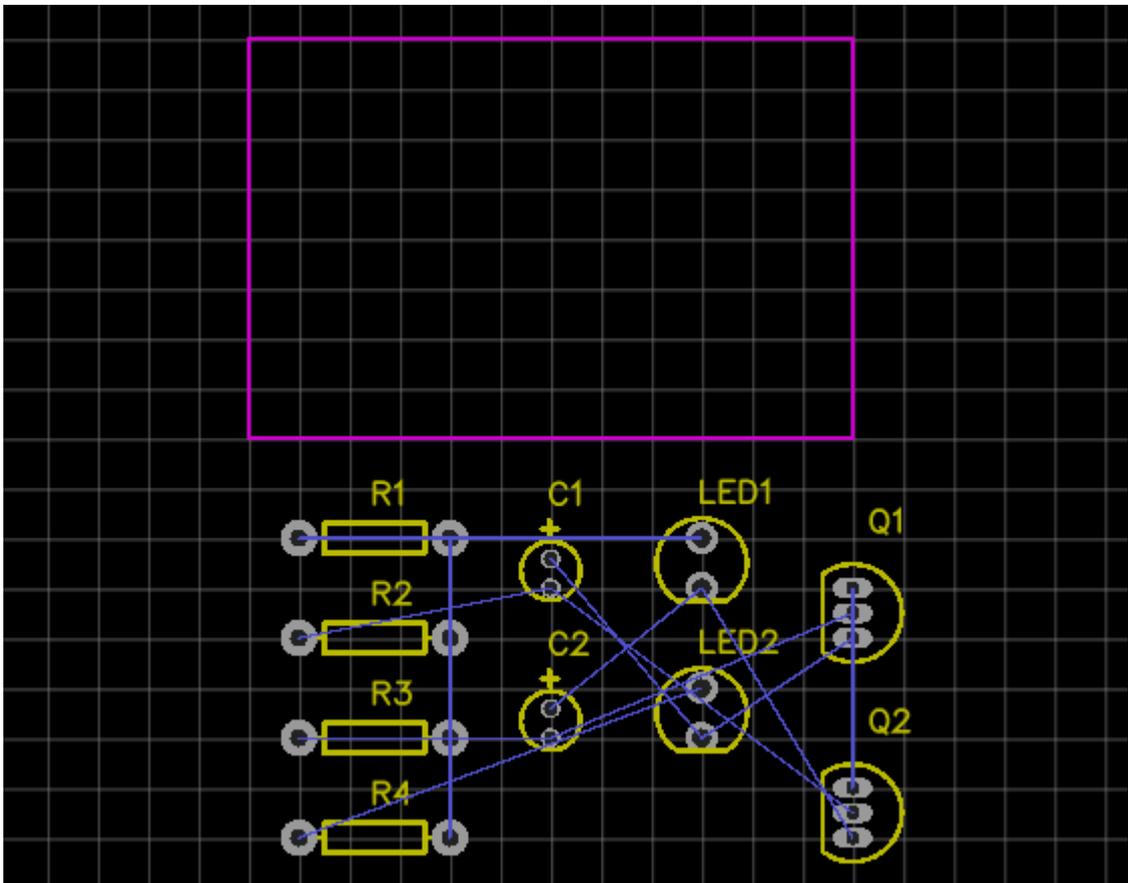
---

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.



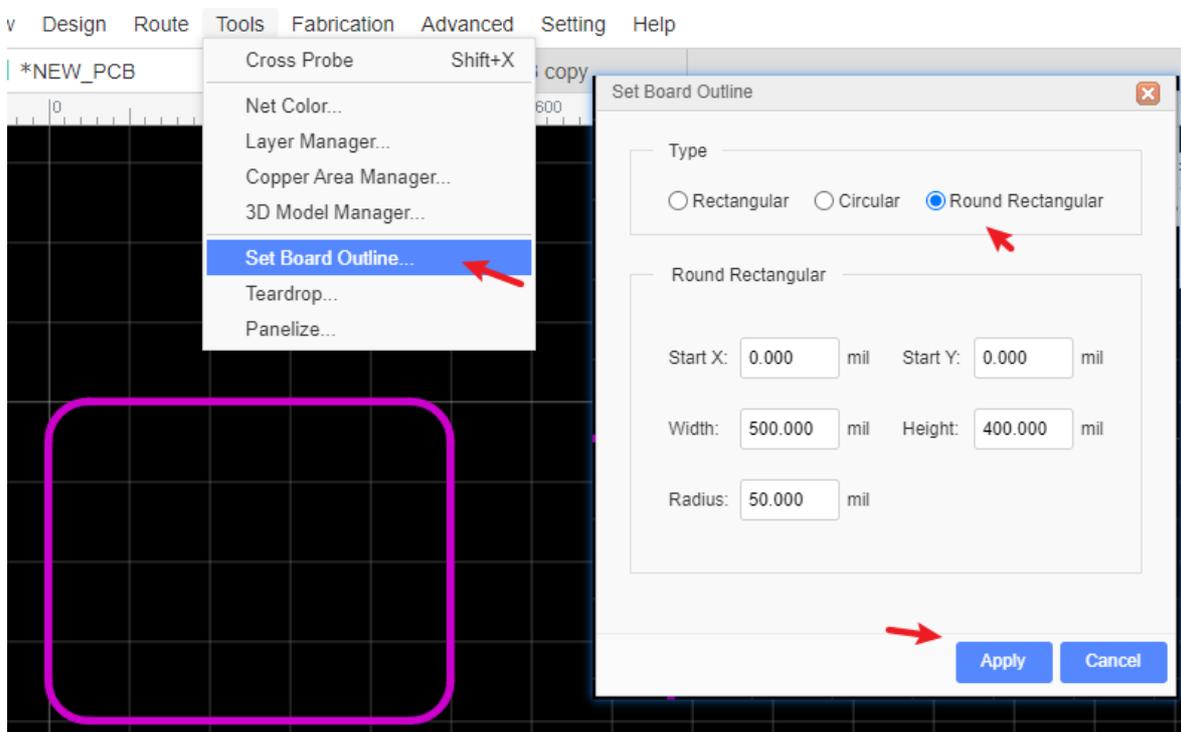
When converting a Schematic to PCB, EasyEDA will try to create a board outline for you.

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 cutout.
- If the board outline doesn't closed, the copper pour will not show up.

## Route Tracks

### Route Tracks

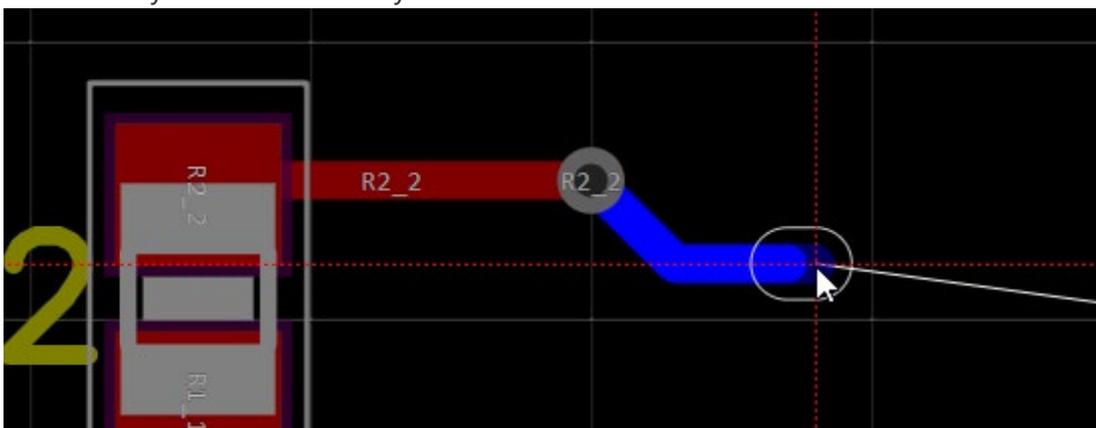
In the schematic editor, we use Wire or the `W` Hotkey to connect Pins, in a similar way in the PCB editor, we use Track to connect Pads. Track allows you to draw PCB tracks and can be found on the PCB Tools palette or using the `W` Hotkey (not T).

### Some Tips about Track

- 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.

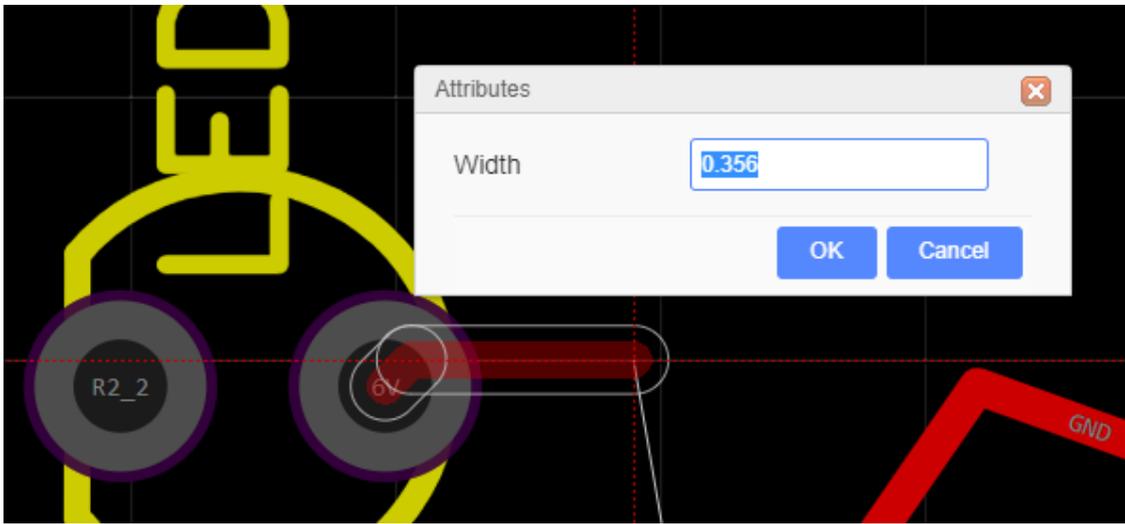


- Drawing a track at the same time as using a hotkey(for example hotkey `B`) for changing the active layer will automatically insert a Via:

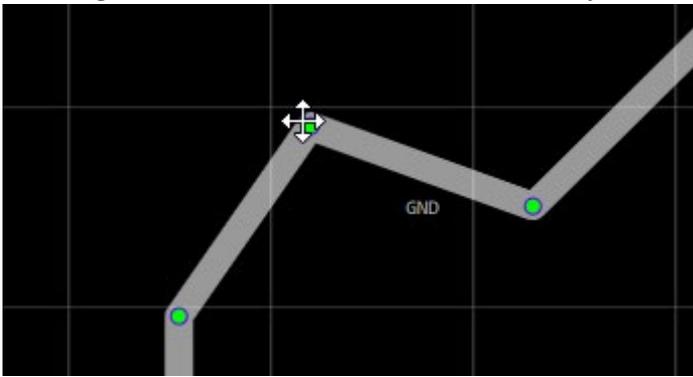


If you start drawing a track on the top layer, you will see it drawn in red, then press the B key to change to bottom layer and you will see EasyEDA insert a grey via and then the track will continue being drawn but now on the bottom layer in blue.

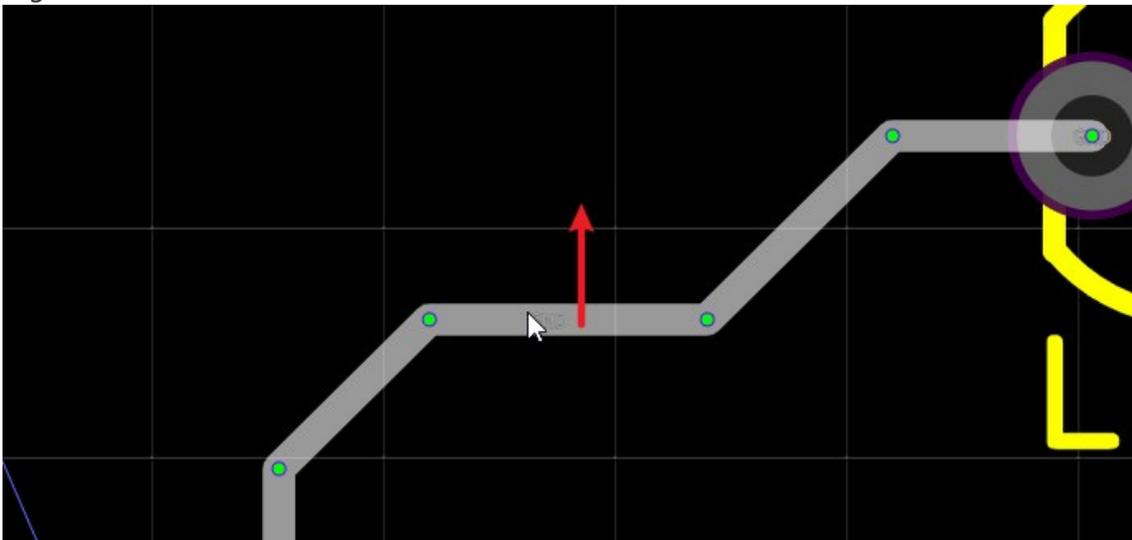
- Pressing the `+` or `-` Hotkeys when drawing the track will change the width of the track on the fly. Use the hotkey `TAB` to change the track width.



- 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.

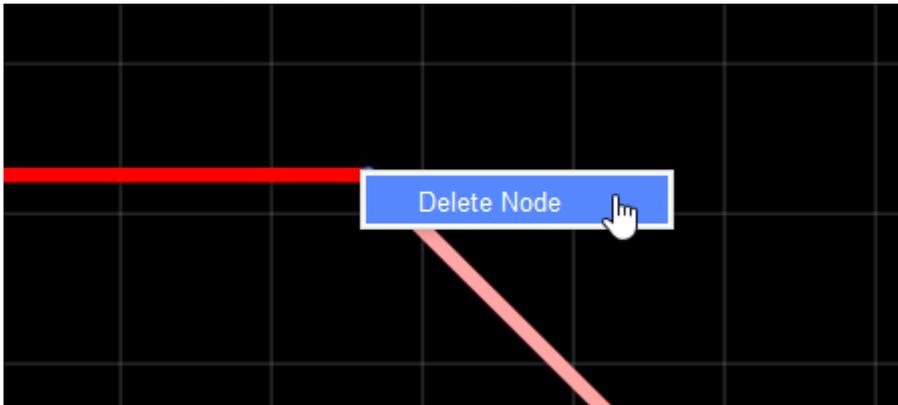
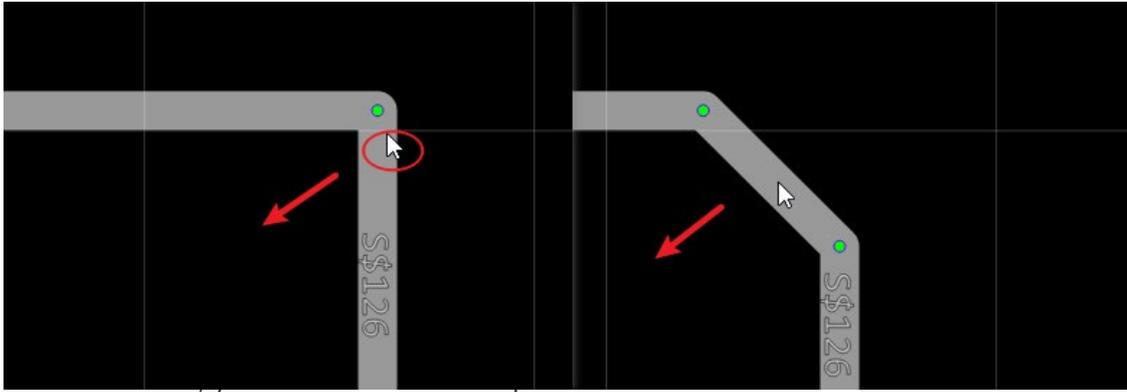


- Click to select the track and then Click and Drag on a segment of the track to adjust the segment between vertices.

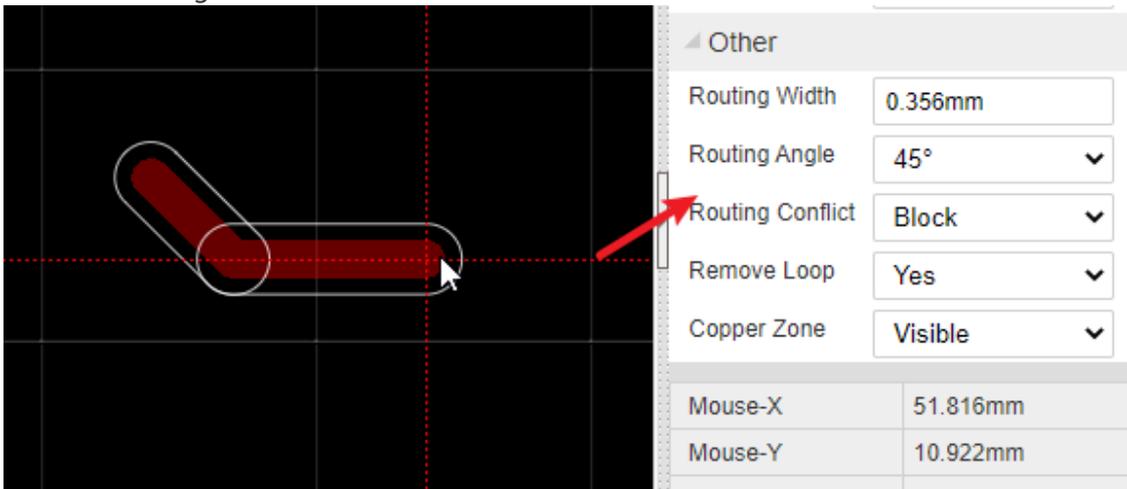


- When the track's corner is a right Angle and the routing corner is 45, drag the track next to the right Angle node to make a bevel. Instead of dragging a node directly, dragging a

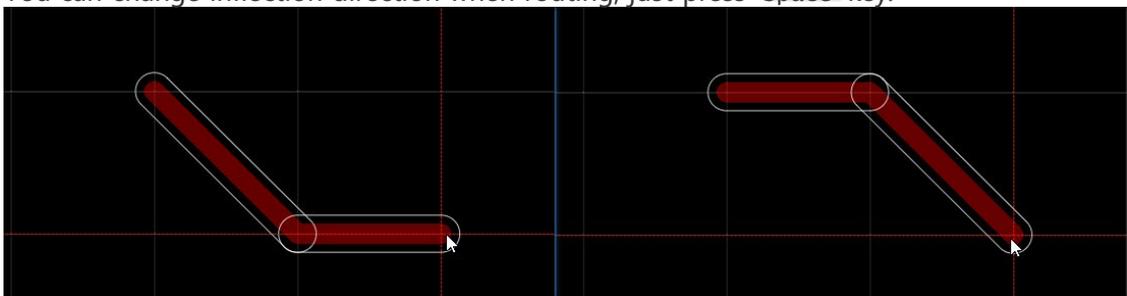
node directly will drag the entire track.



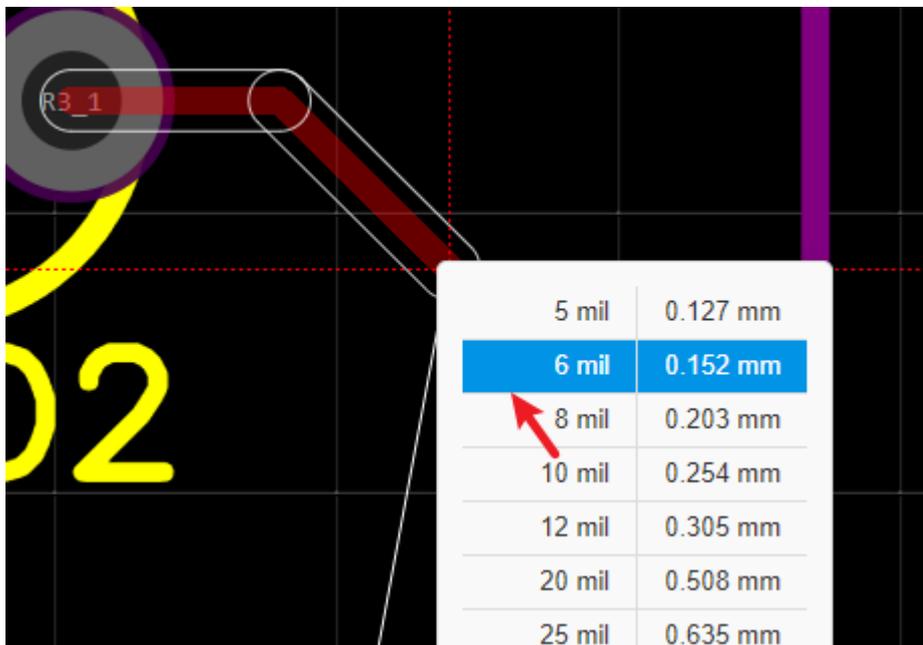
- Pressing the **L** Hotkey when drawing the track will change the track's Route Angle on the fly. And you can change Route Angle on the Canvas Attributes of the right panel before the next drawing.



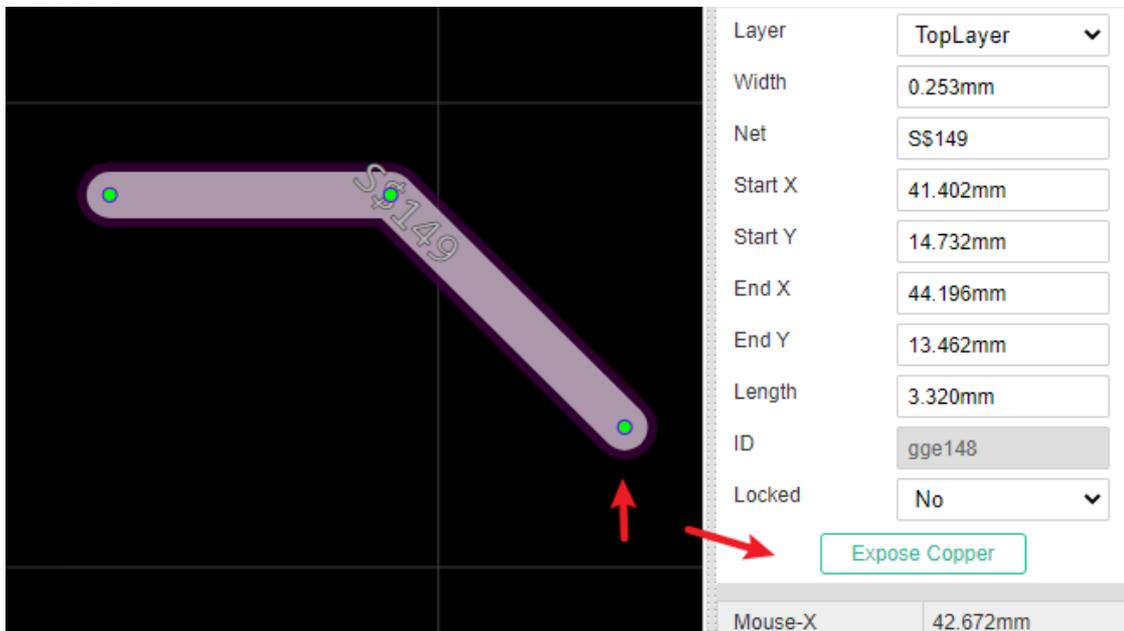
- You can change inflection direction when routing, just press **space** key.



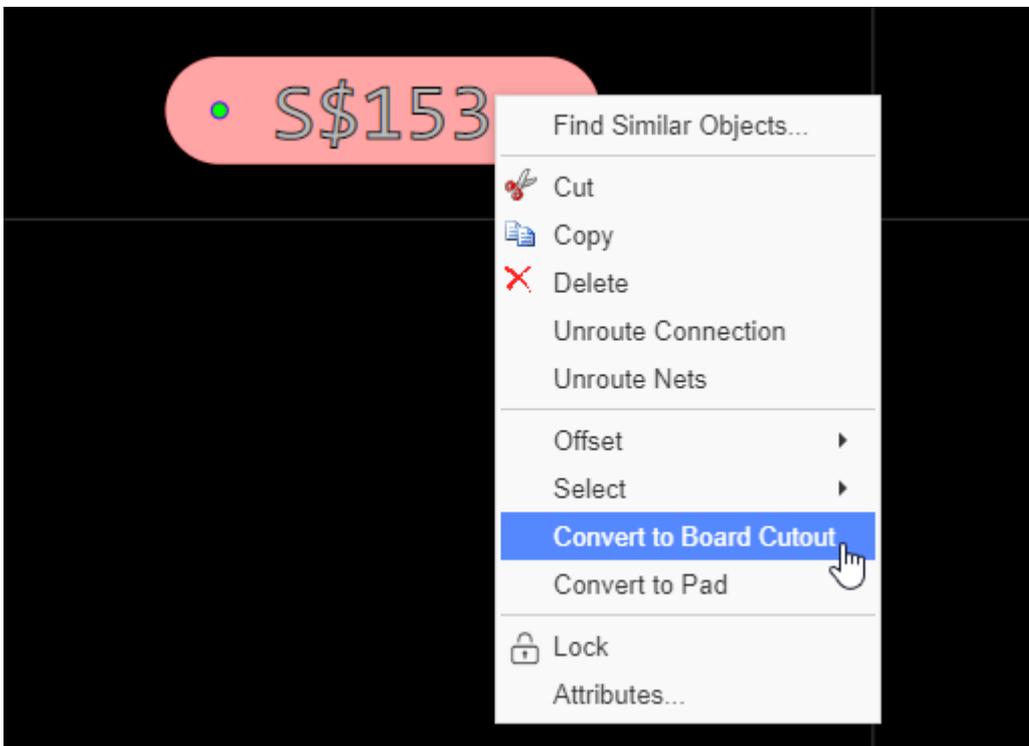
- If you want to route a track and use "L", and then press "+", you will get two different size track segments. or press "SHIFT+W".



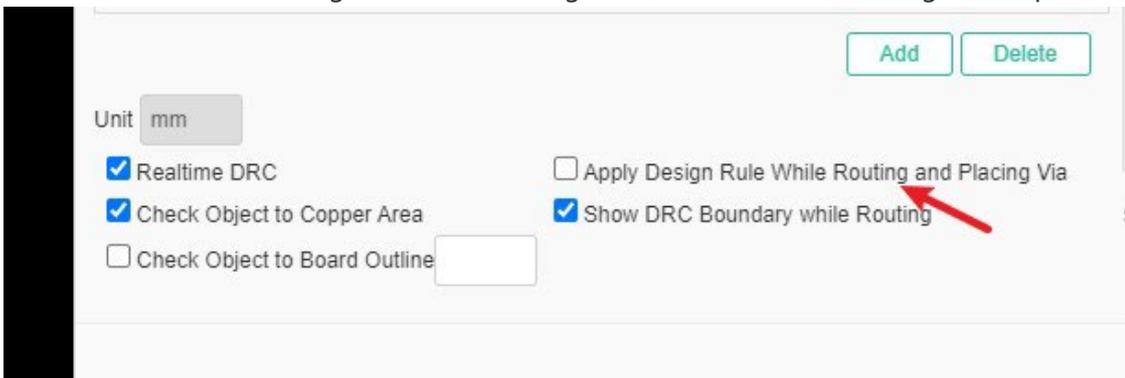
- If you want to create the solder mask aperture for the track, you can use "Expose Copper" when you select the track on the right-hand panel. The solder mask will be bigger 4mil than the track.



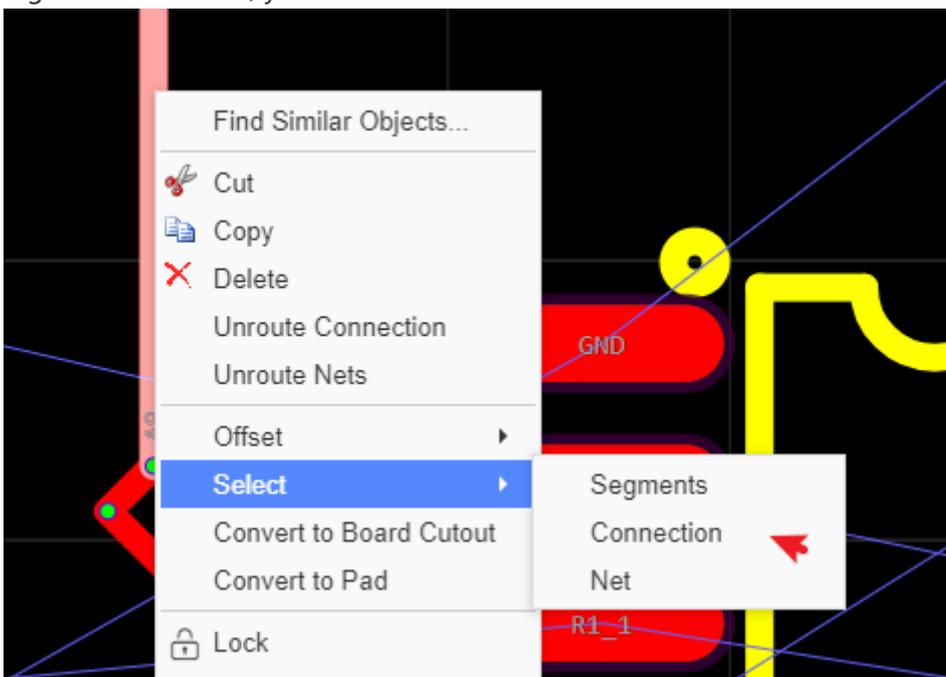
- And if you want to create the slot hole, you can route a track, and then right-click the "Convert to Board Cutout" menu.



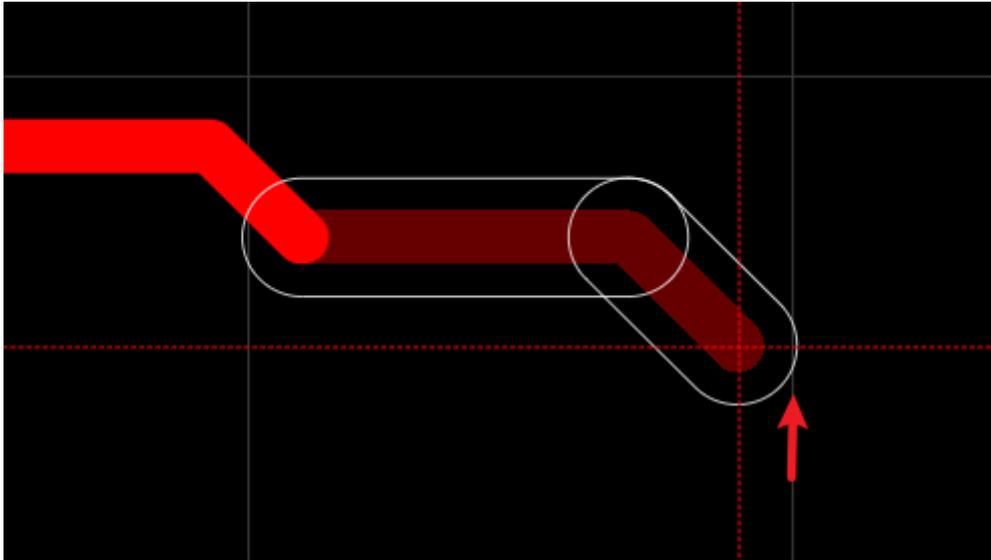
- You can make track routing width follow design rule, after enable the design rule option.



- Right-click the track, you can select the track connection or a whole same net tracks.



- You can disable the DRC boundary at Design Rule. The size follow the rule.



Unit mm

Realtime DRC  Apply Design Rule While Routing and Pl

Check Object to Copper Area  Show DRC Boundary while Routing

Check Object to Board Outline

System Settings

System Schematic PCB

Canvas Zoom Effect Speed Priority

Rotation Step 90

Favorite Track width Track Width Setting

Add Teardrop Automatically  Assign Net for Free Track/Arc/Circle

Terminate Routing Automatically  Open Wizard Dialog for New PCB

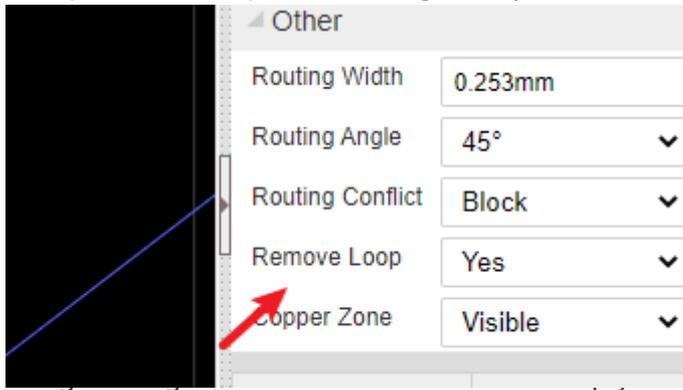
Rebuild Plane Automatically  Net Highlighting While Cursor Hover the Track

The Track's Routing Follows Component's Rotation  Display Pad's Number and Net

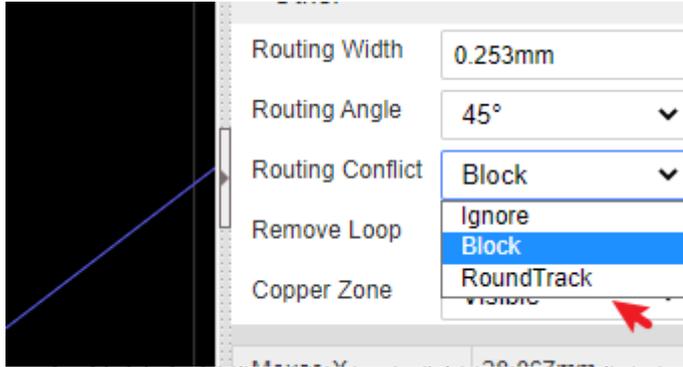
Cursor Snap to Component's Origin or Pad Center While Dragging Component

Apply Cancel

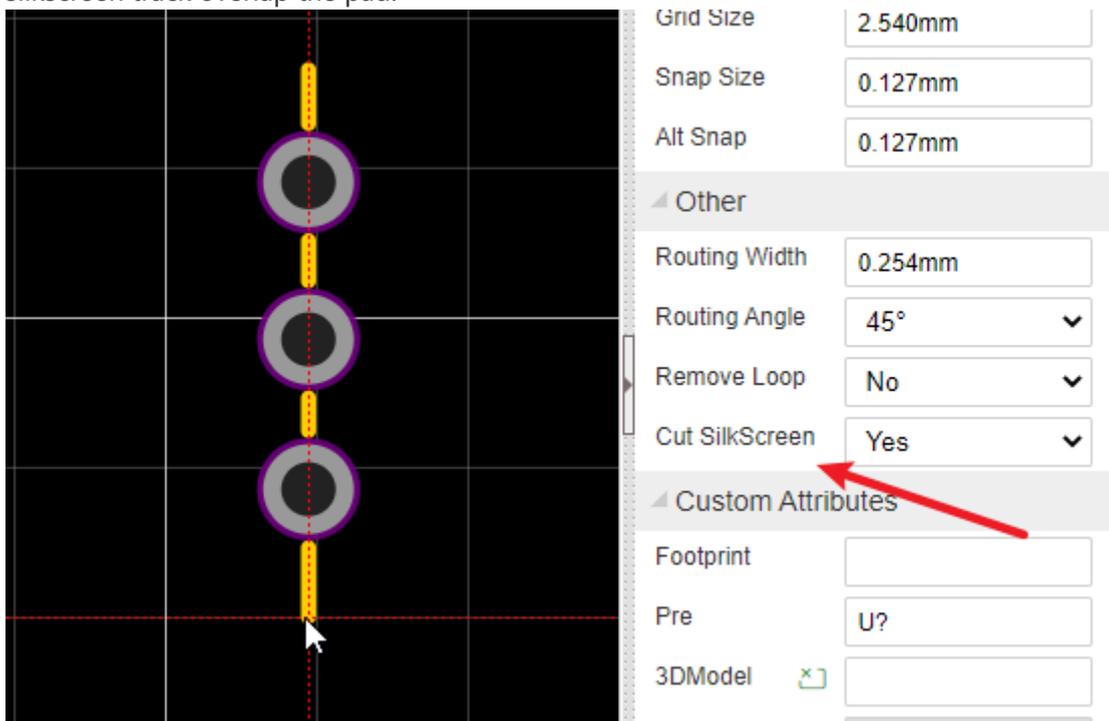
- Set up Remove Loop while routing, it only works on copper layer.



- Press the 'F' key to finish routing quickly.

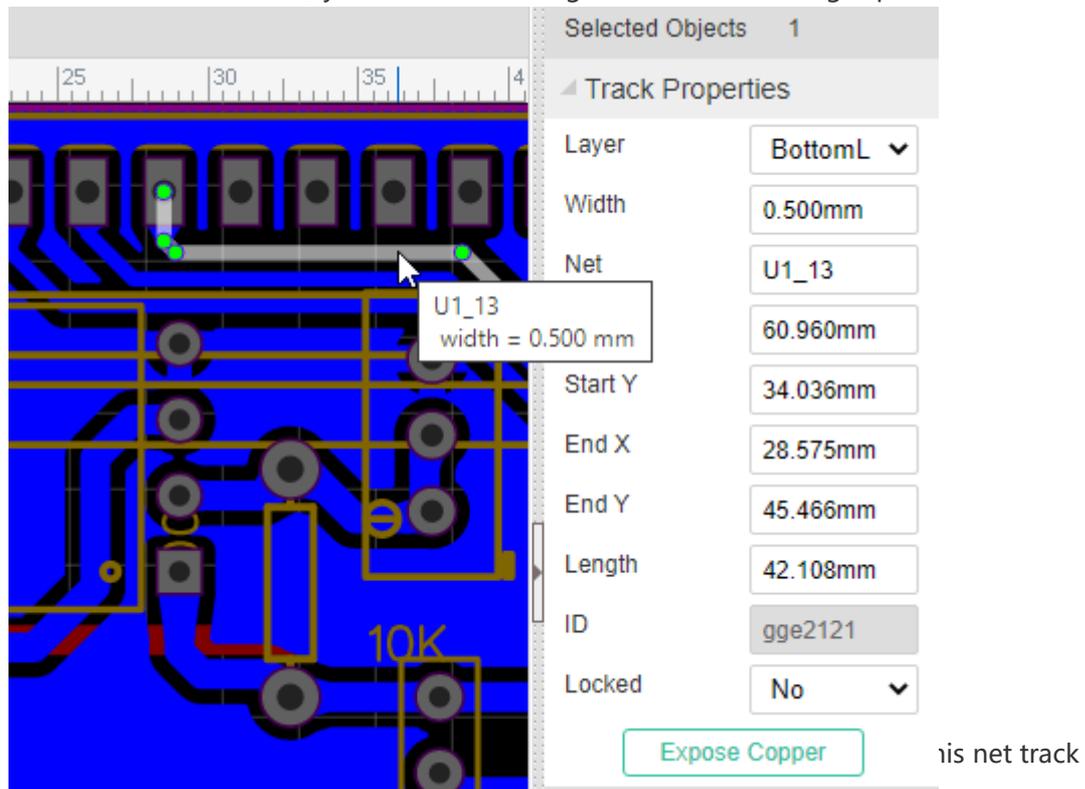


- Press the 'S' key to "Cut SilkScreen" to avoid the silkscreen track overlap the pad.



**Track Length**

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



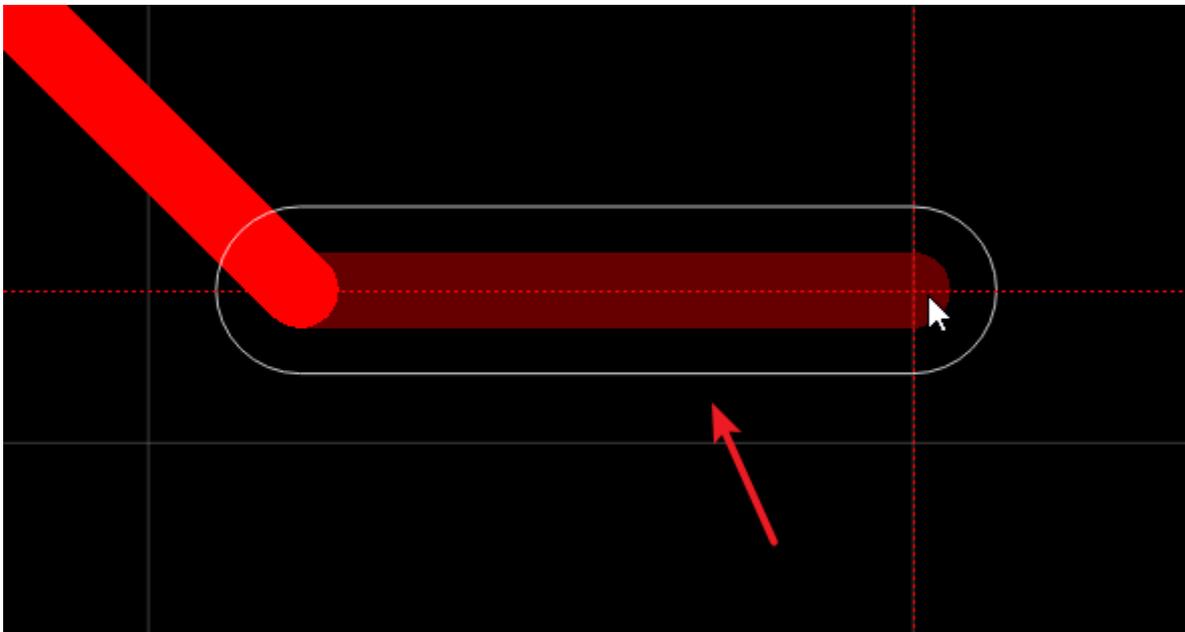
- Click a track, press hotkey H will keep highlight this track and net, and show this net's length.

## 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

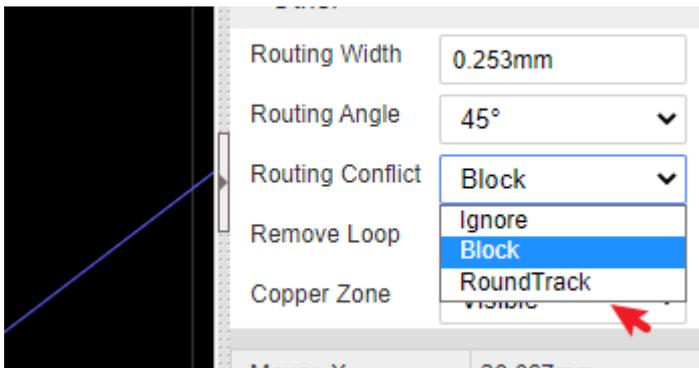
When you routing a track on the signal layer, you will see an outline around the first track, it is the DRC outline, the clearance from outline to the track edge depends on your Design Rule(DRC) clearance setting.



## 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 around 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

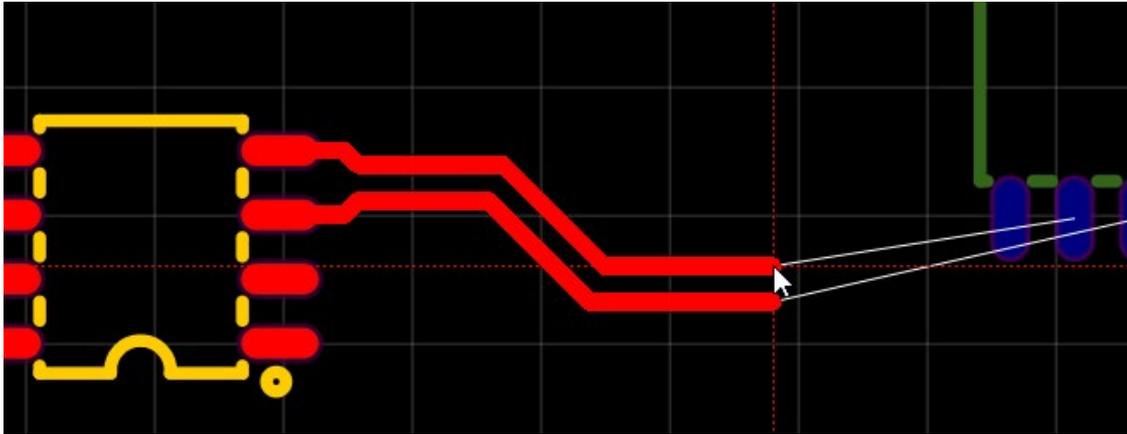
You must make sure the Differential Pair net names must be `XXX_N`, `XXX_P` or `XXX+,XXX-`.

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

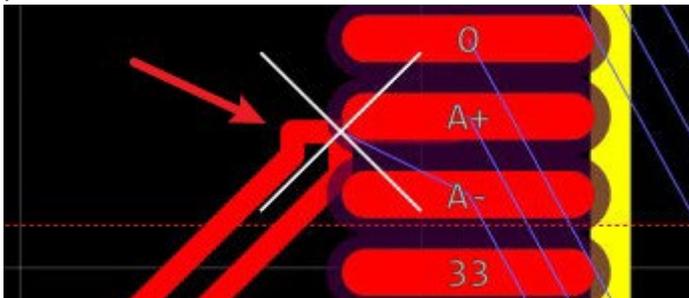


Notice:

- 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:

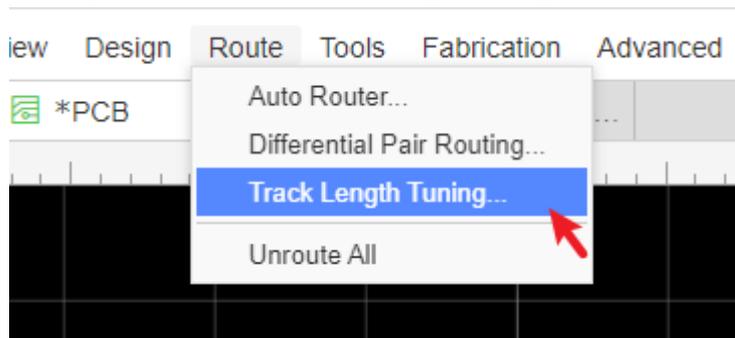
- When finish previous routing location too close with the finish pads, the track will generate the extra segments, please finish the previous location far away from finish pads.

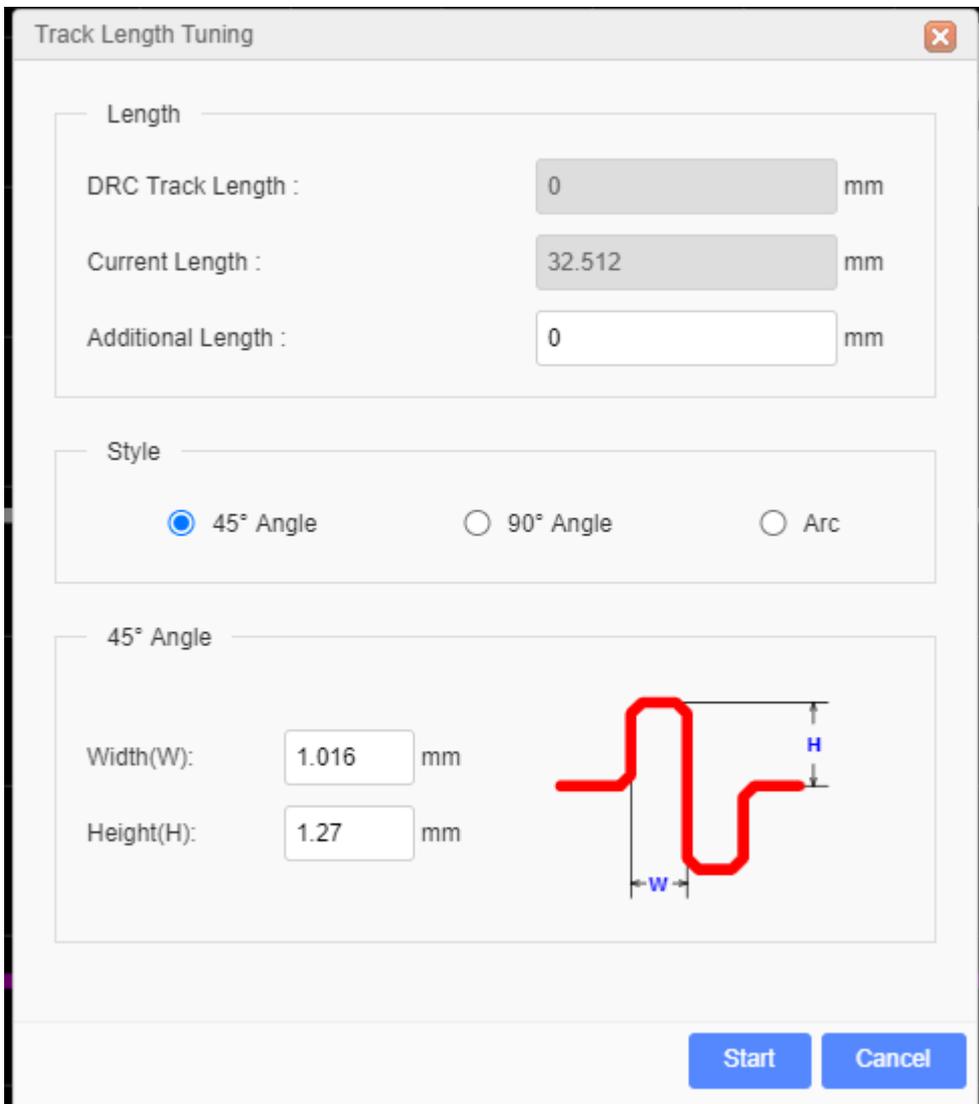


## Track length Tuning

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.

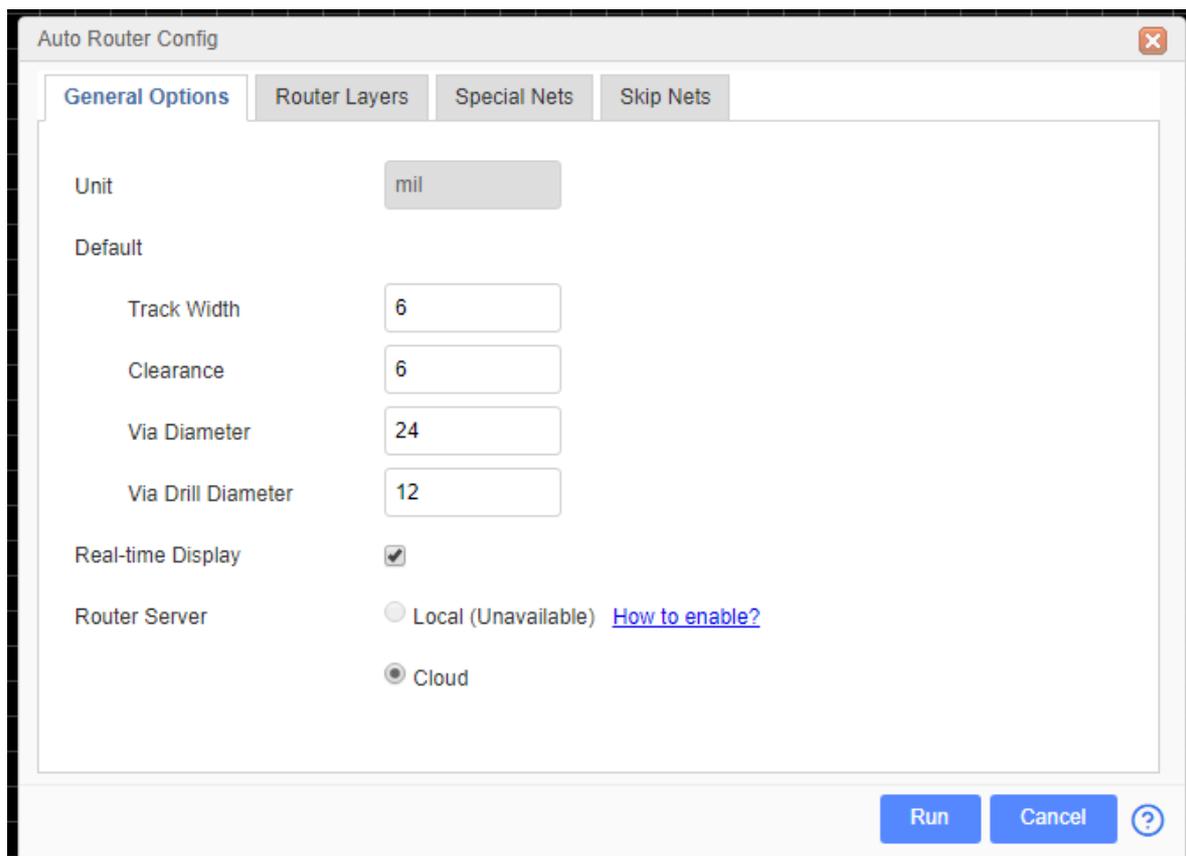
**Auto router is not good enough! Suggest routing manually! You can use "RoundTrack(Walk Around)" option to route tracks, via right-hand panel - Routing Conflict.**

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

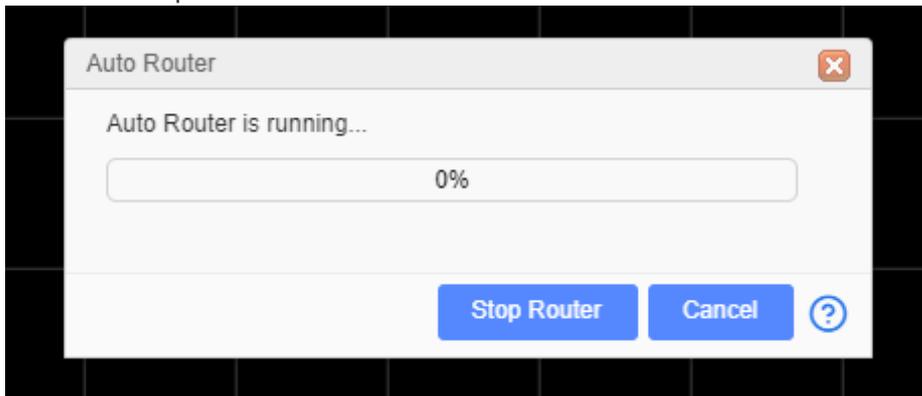
- ◆ **Unit:** The unit follows PCB canvas unit.
- ◆ **Track width:** The auto-route track width.
- ◆ **Clearance:** The clearance of the objects.
- ◆ **Via Diameter/Via Drill Diameter:** The via placing by auto-router.
- ◆ **Realtime Display:** when you select it , the real time routing status will show 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.

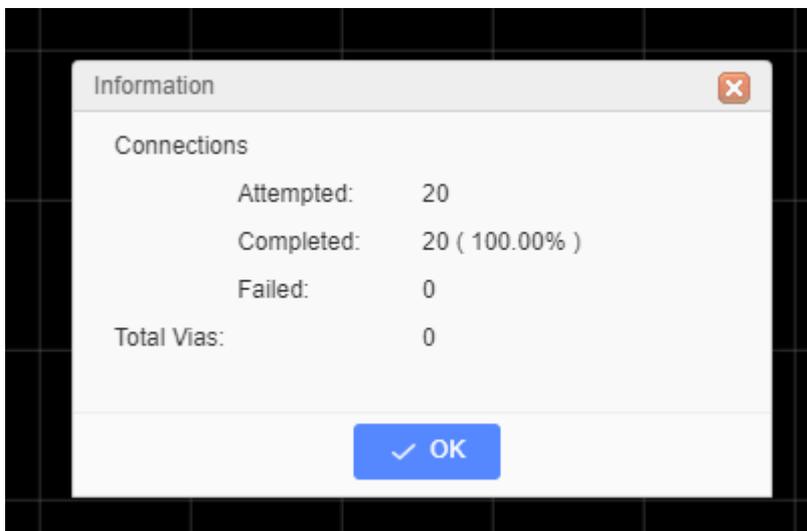
### 3 Run it

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.

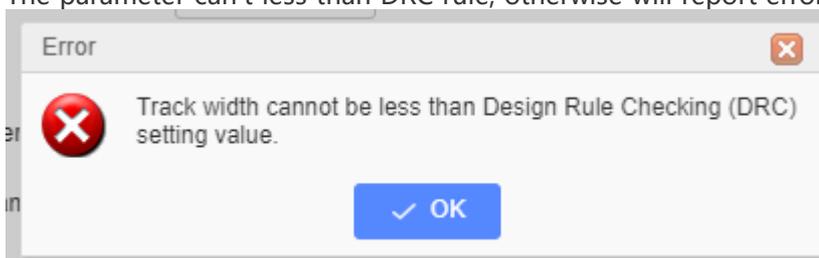
When finish, will pop up a window.



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.**

Download EasyEDA Router:

[easyeda-router-windows-x64-v0.8.11.zip](#)

[easyeda-router-linux-x64-v0.8.11.zip](#)

[easyeda-router-mac-x64-v0.8.11.zip](#)

Supported OS:

- windows7(x64) or later 64bit windows
- Ubuntu17.04(x64) or other 64bit Linux, Linux recommend [Deepin] (<https://www.deepin.org>)
- 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**

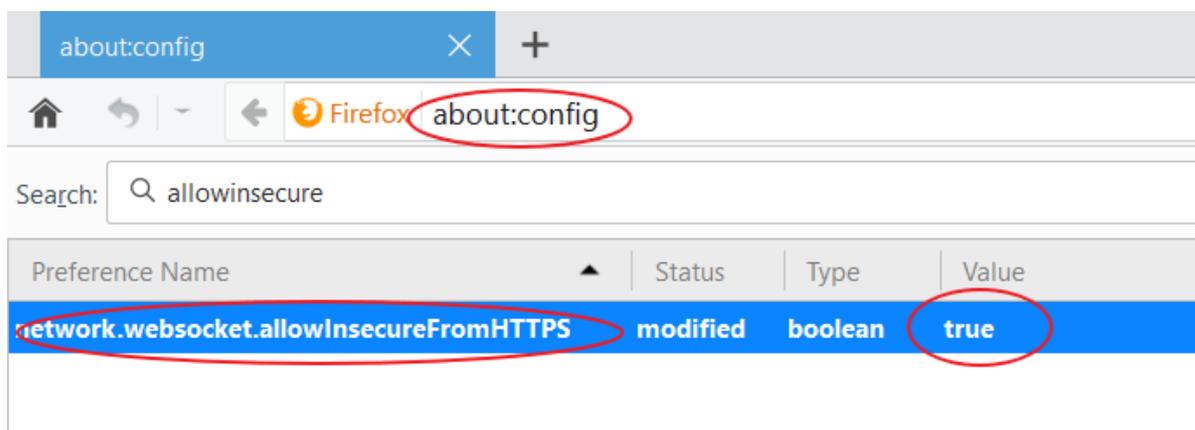
The Chrome Browser don't need to be configure, If the local auto router is unavailable, you have to upgrade Chrome to version 60.0.3112.78 or later.

- **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

Auto router is not good enough, suggest routing manually, recommended "Routing Conflict - RoundTrack" at the PCB right-hand panel.

Sometimes, if you can't get it done, try the tips below.

- Use local auto router rather than cloud server.
- Make sure the net of PCB doesn't contain the special characters, such as `` { } ^ ; ~ \ / [ ] =` etc. the character `-` 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 package.
- Make sure no footprint outside the board outline.
- Make sure your canvas Grid size is set to 10mil, make sure the components align with the grid via: top menu - format - align grid.
- Place some vias at suitable position, and modify the vias' net as you want.
- 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 more cost.
- Change the components layout, make them have more space between each other.
- Don't make any via/pad overlap the different objects which can be set the net.
- Stop the auto-router, close the script, and re-open the script to run local auto-router server again.
- Sometimes, The auto-router won't work at the first time, please modify the layout and try several times.
- If only a few remaining networks cannot be completed, it is a normal phenomenon. Please manually route the remaining networks.
- Tell the error detail to us and send your PCB file as EasyEDA Source file to [support@easyeda.com](mailto:support@easyeda.com).  
<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.

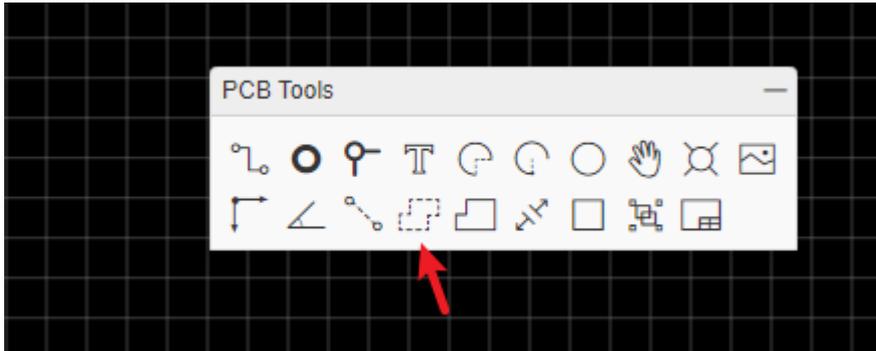
---

## 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).

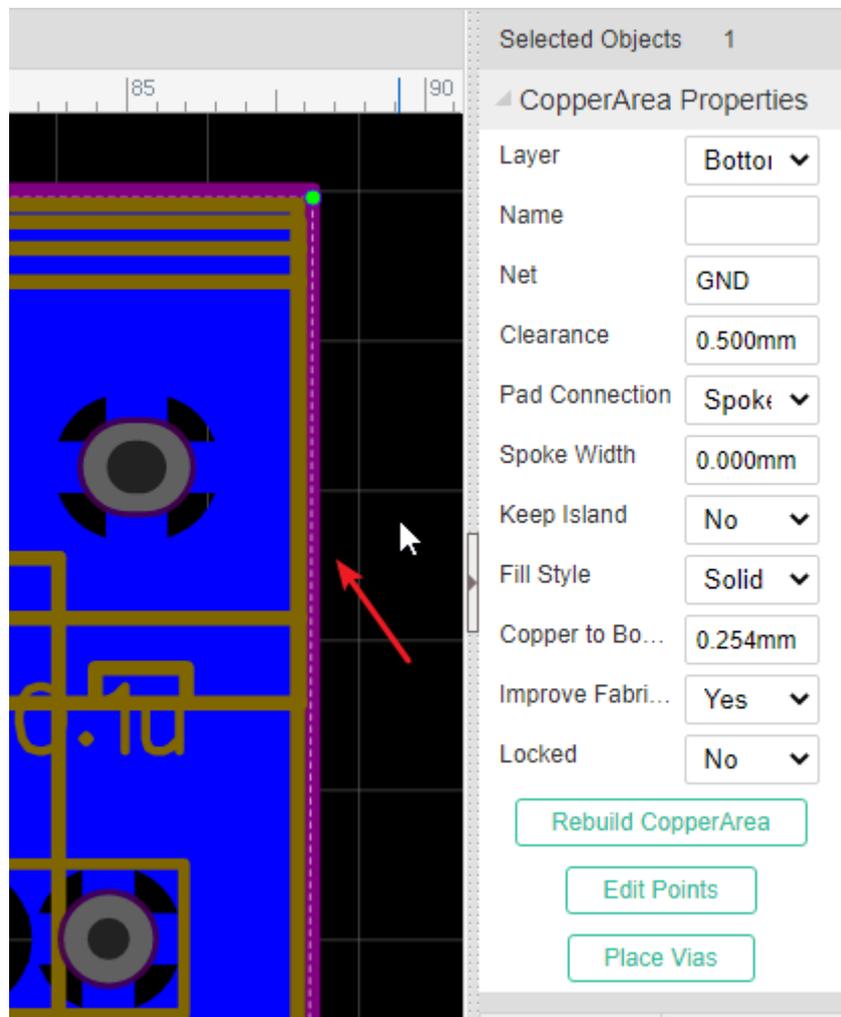


**Before using Copper Area, please make sure your PCB has a closed board outline!**

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.

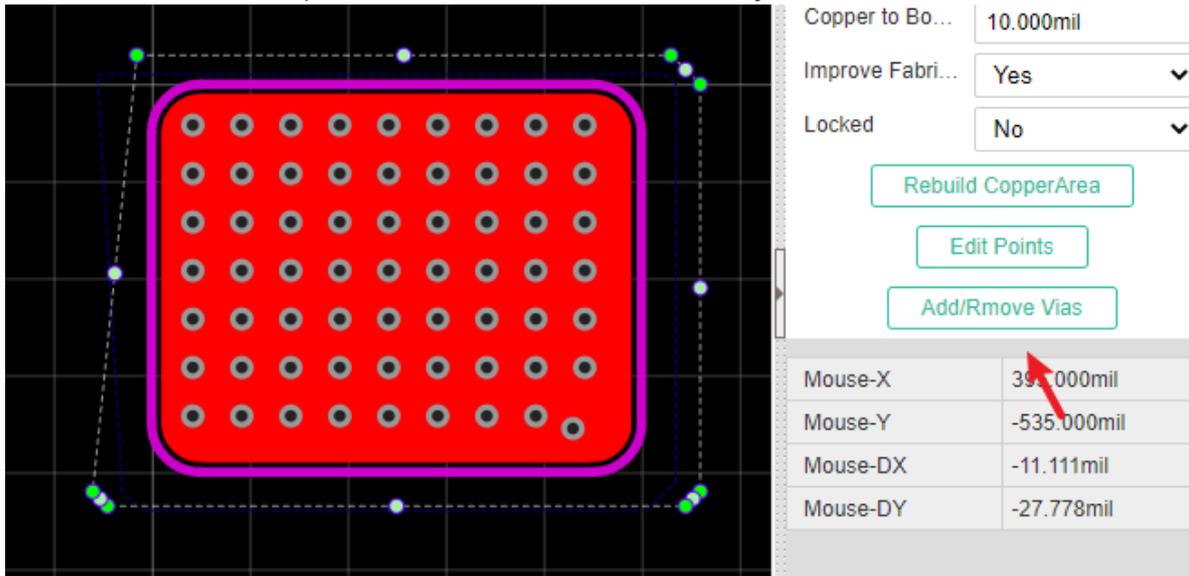
**Copper to Board Outline:** Setting the clearance between copper with board outline.

**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.

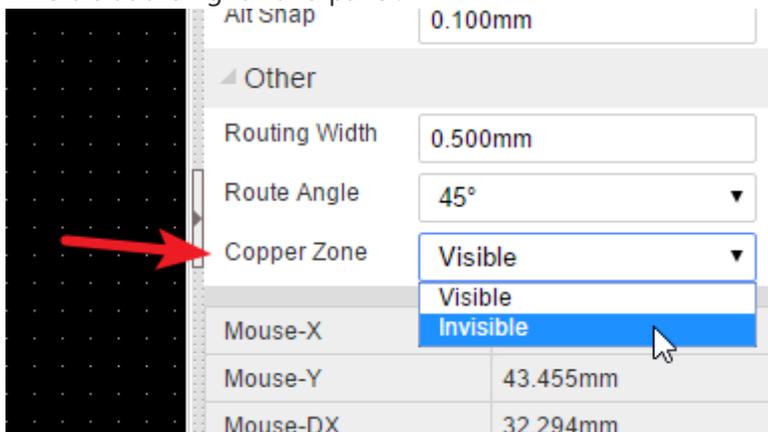
**Edit Points:** You can edit the copper area shape manually, any shape as you want.

**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.

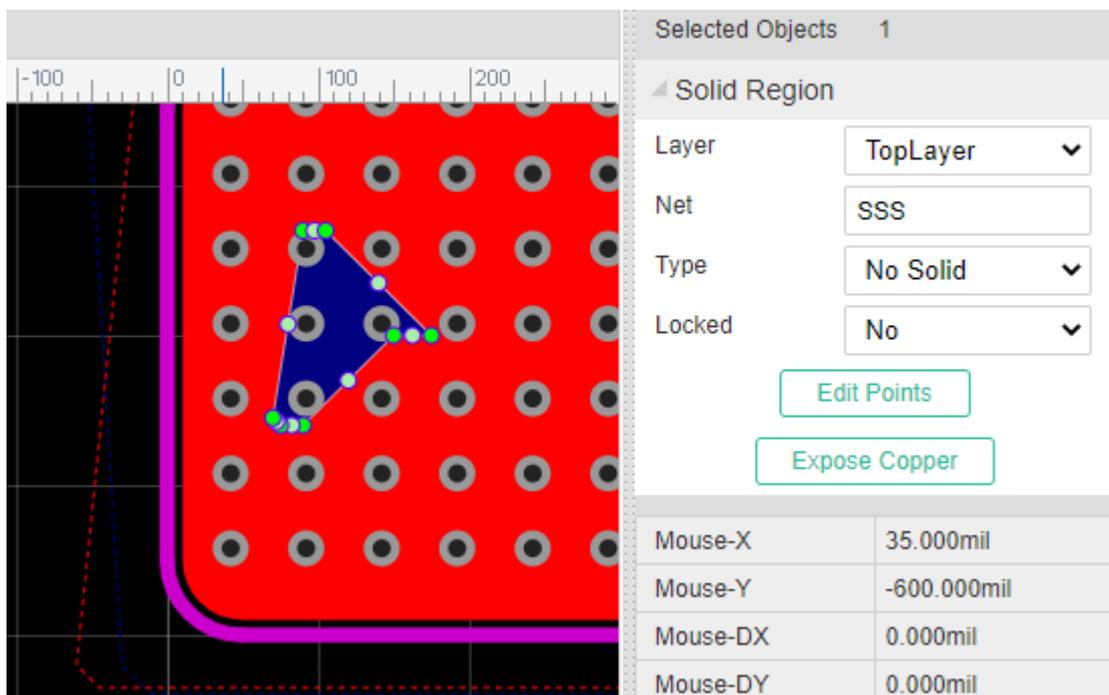


## Tips

- 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

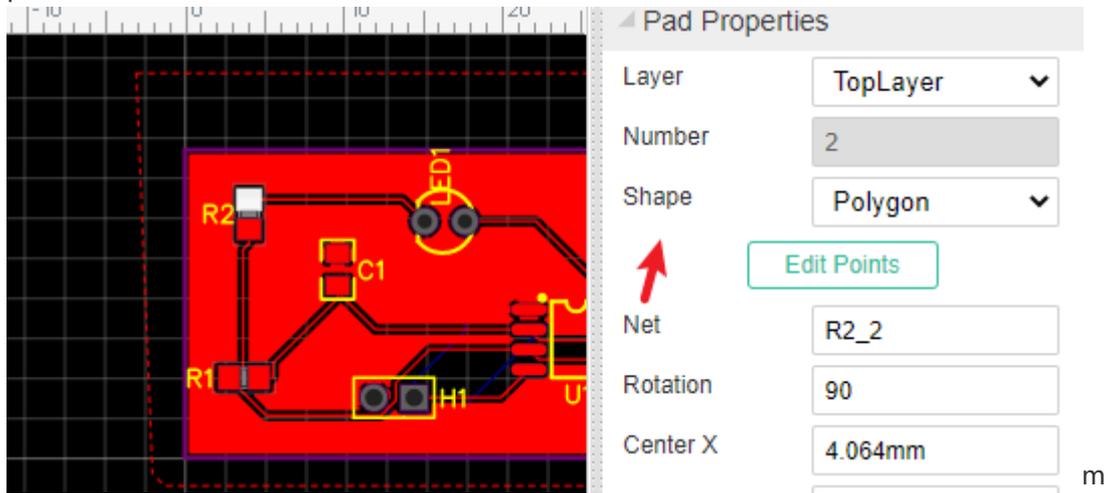
- Because of the browser's performance issue, EasyEDA doesn't support the real-time copper pour, after PCB modifying, please rebuild copper area via Hotkey `Shift+B`.
- EasyEDA doesn't support click the copper zone, you need to click the copper outline to select it.
- **If the PCB size is more than 15MB, 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. If the PCB size is less than 15MB, the copper filled data will be saved into file, and you can open the PCB in another PC without rebuild copper area.**

## FAQ

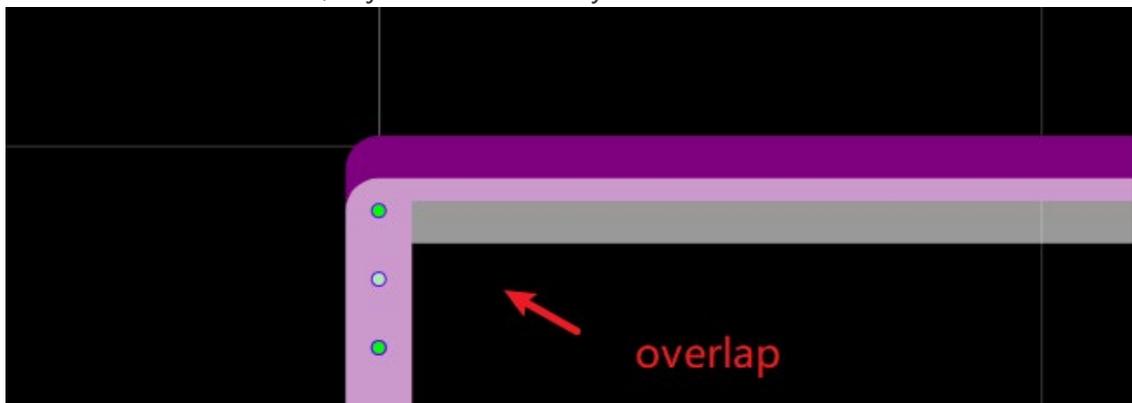
### Why sometimes it takes a long time to copper pour

- 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.

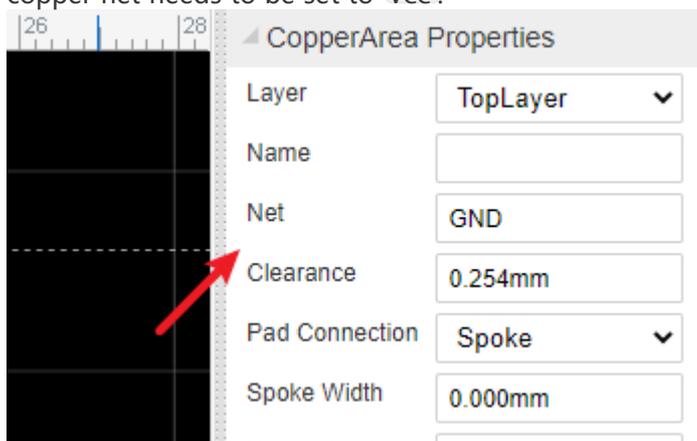


- ◆ Design PCB, Altium Design picture is a large number of track segments combined, need to be manually removed.
- ◆ Check that the board outline is complicated, with overlapping board outlines, or a large number of board outlines, adjust them manually to reduce the number of board outlines.



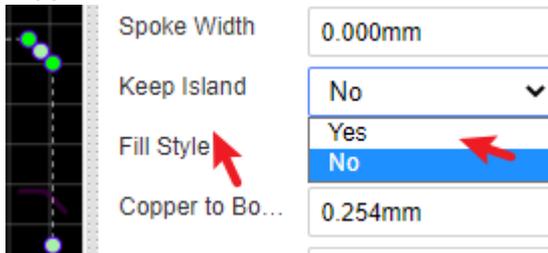
### Why did I not show the copper fill after copper poured

- ◆ Your copper area net must have the same pad or via same 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`.

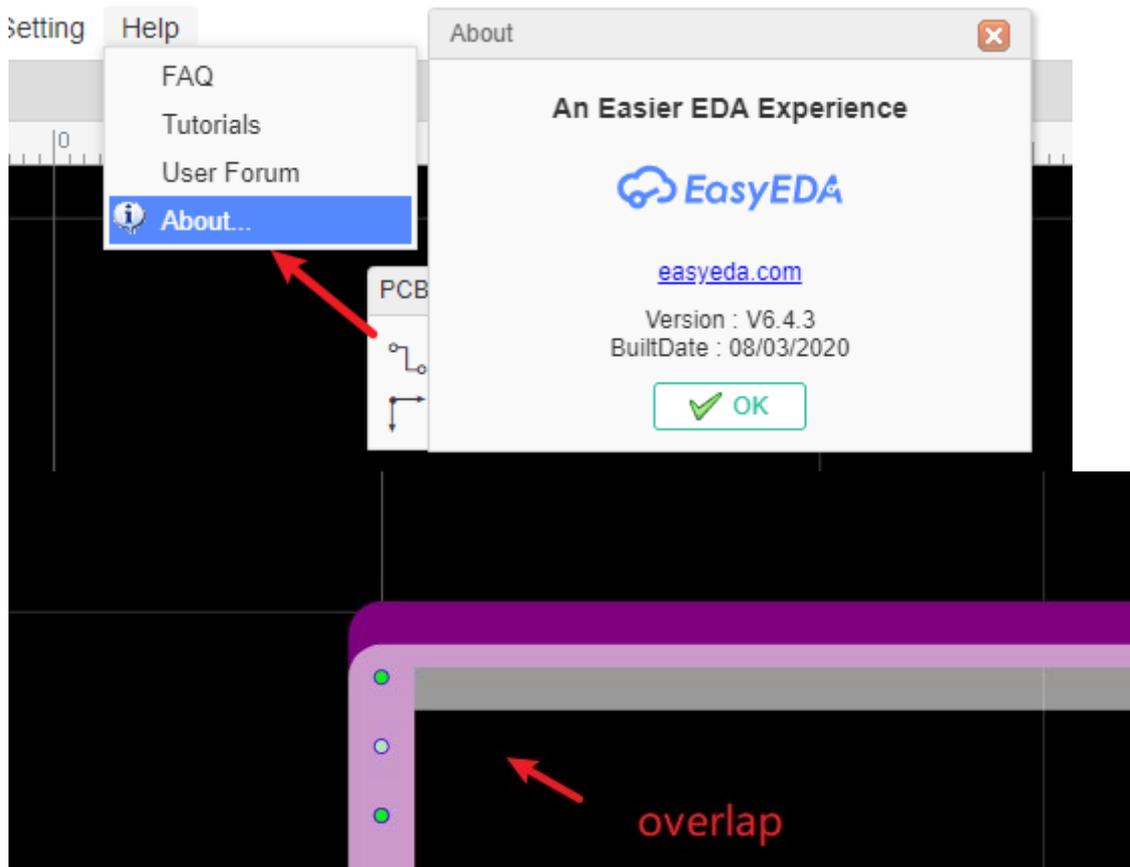


- ◆ If you don't change the copper area net, you can click on the copper outline and modify the property "Keep Island" to Yes in the right property panel. The copper area logic of the EasyEAD is based on whether there is a connection or not to decide whether it is an island, and if there is no element connection to the same net, the

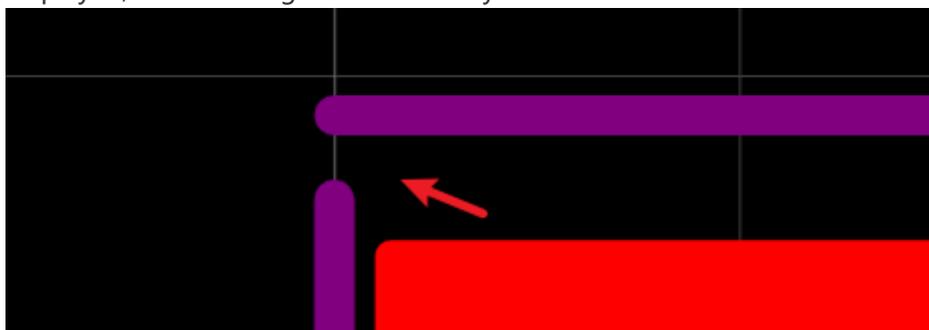
copper area will be considered an island.



- ◆ 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.



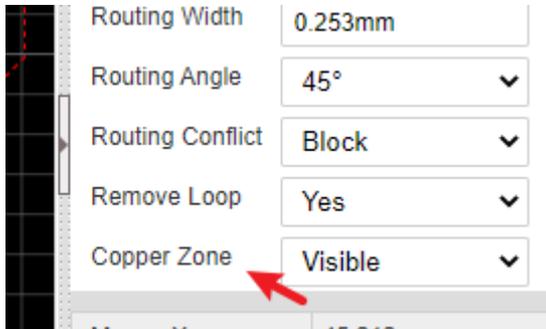
- ◆ Check that the board outline is closed and that endpoints need to be closed between the tracks, and that there are overlapping segments of the board outline (usually inside the imported PCB). Once you can hide all layers, only the board outline layer view is displayed, and each segment is carefully examined.



- ◆ Check that the copper area property is set to type No Solid and needs to be set to Solid or Grid.



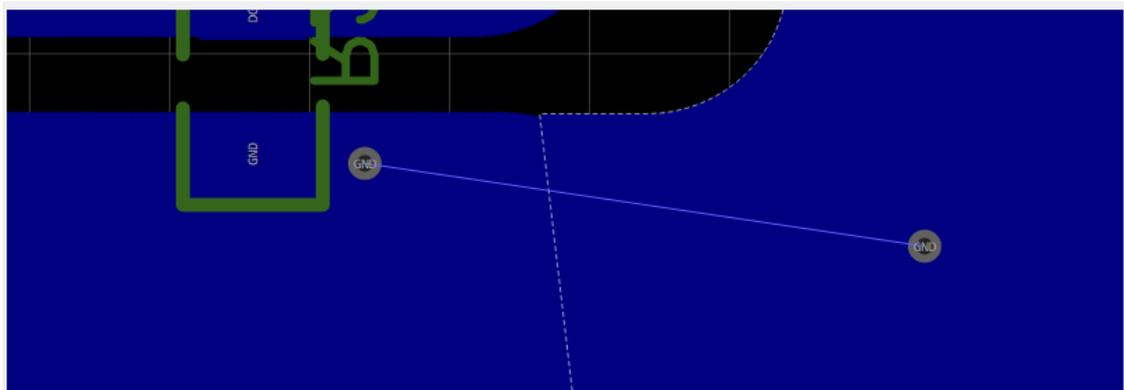
- Whether to make the copper area invisible, on the right side of the canvas, set the copper zone to Visible.



- Still unable to copper pour may be an editor bug, please contact us.

### Why the ratline doesn't disappear when two copper area overlap which are the same net

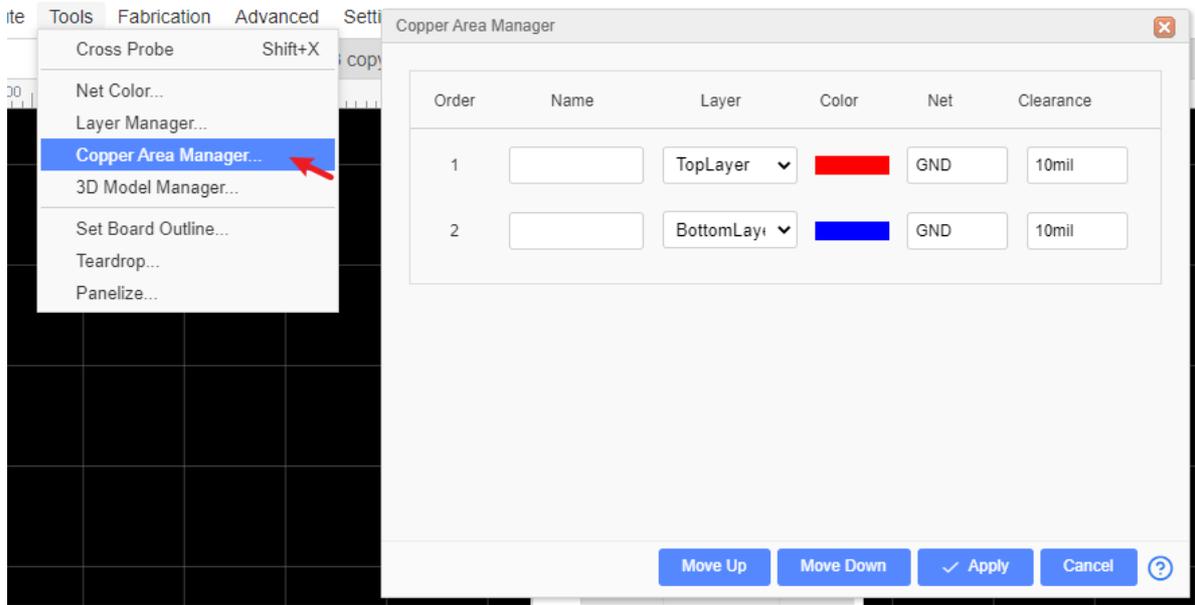
- At present, doesn't support this, that will make the ratline generation very slow. Please route a track manually to connect these copper areas.



## Copper Area Manager

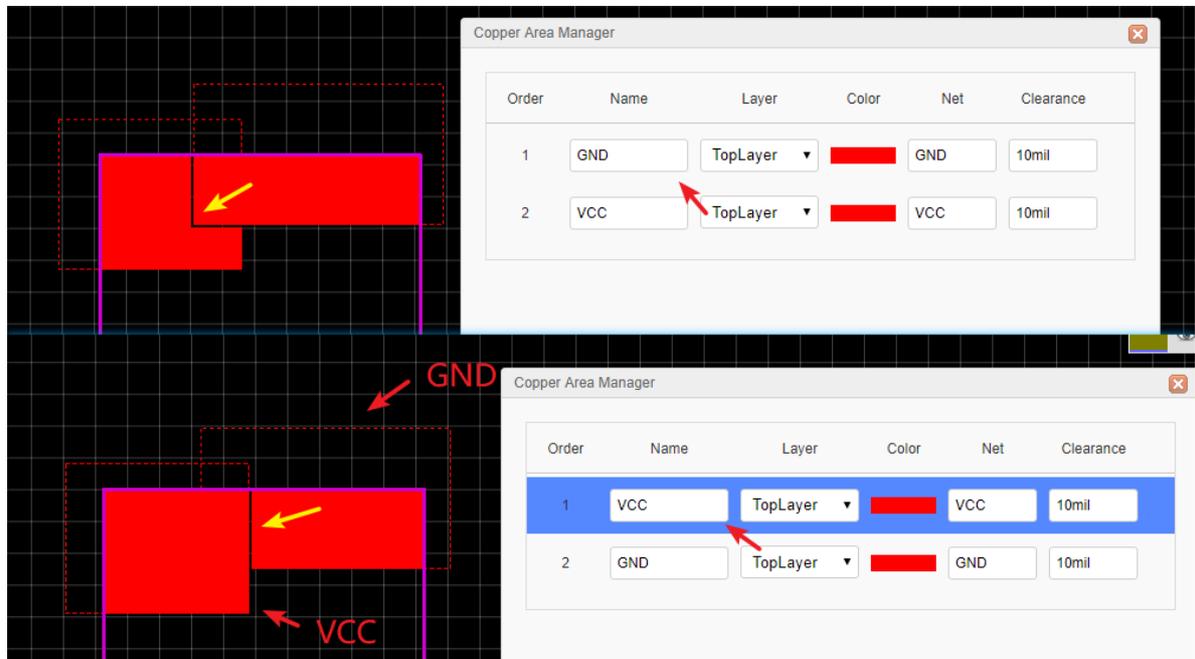
EasyEDA support copper area manager now, you can set the copper order and apply, the forward copper area will be poured first.

Via: Top Menu - Tools - Copper Area Manager



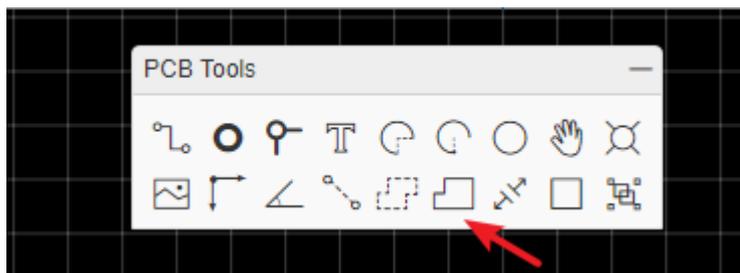
For example:

The GND on the top and VCC on the top, you can see the clearance is different.



## Solid Region

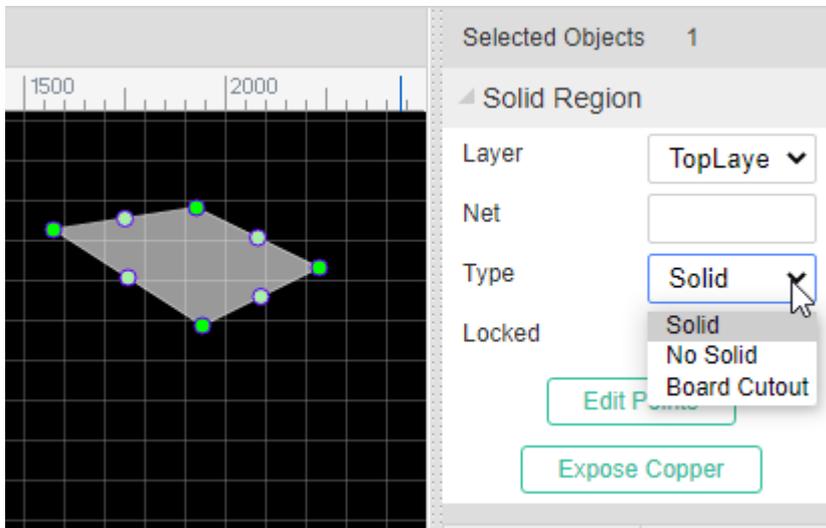
EasyEDA has added a new tool Solid Region for PCB design



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).



- **Edit Points:** You can edit the solid region's outline points as you want.
- **Expose Copper:** you can create an aperture in the solder mask by one click. It's very easy to do.

The outline of the solid region can not be self-intersection, when it happens, please delete the self-interaction point at "Edit Points".

---

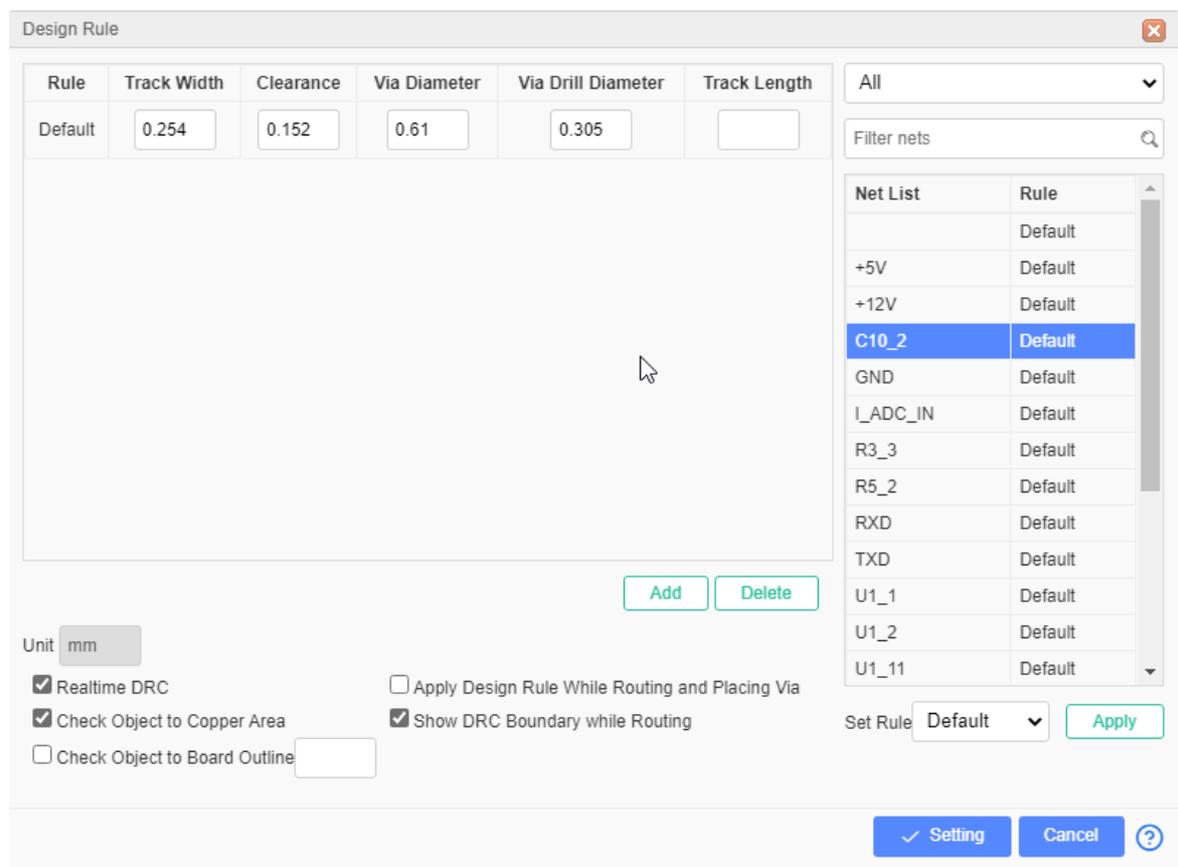
## Design Rule Check(DRC)

---

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.

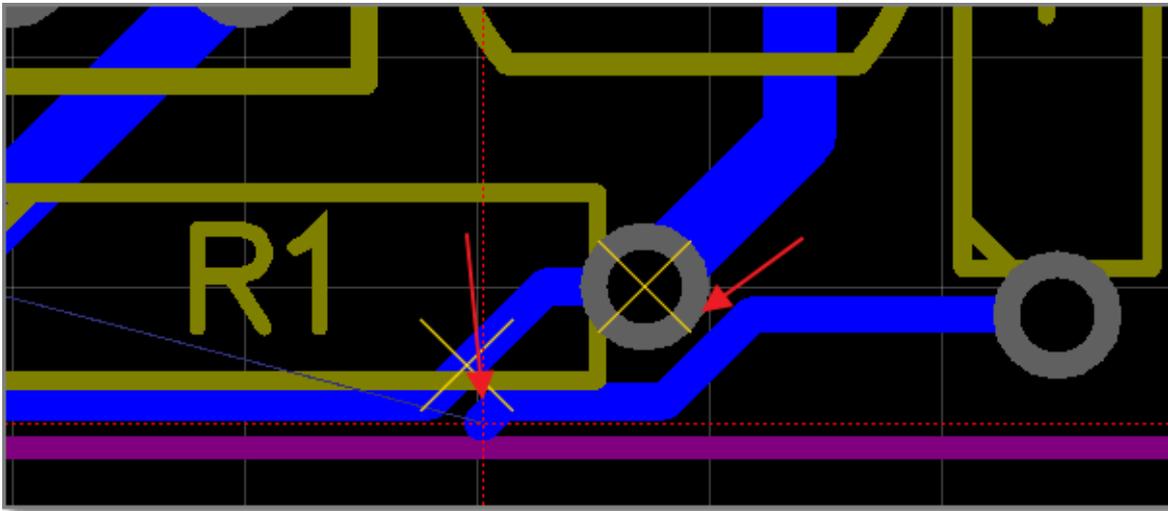
**Clearance:** The clearance of different objects which have different net. The clearance of the PCB can not less than this value.

**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.

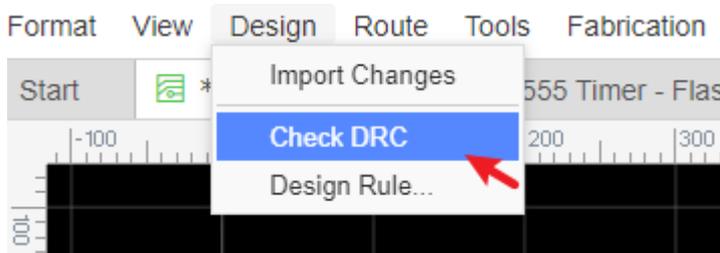
**Show DRC Boundary while Routing:** When routing you will see a outline around the track. Its diameter depends on design rule.

## Set Rule for a Net

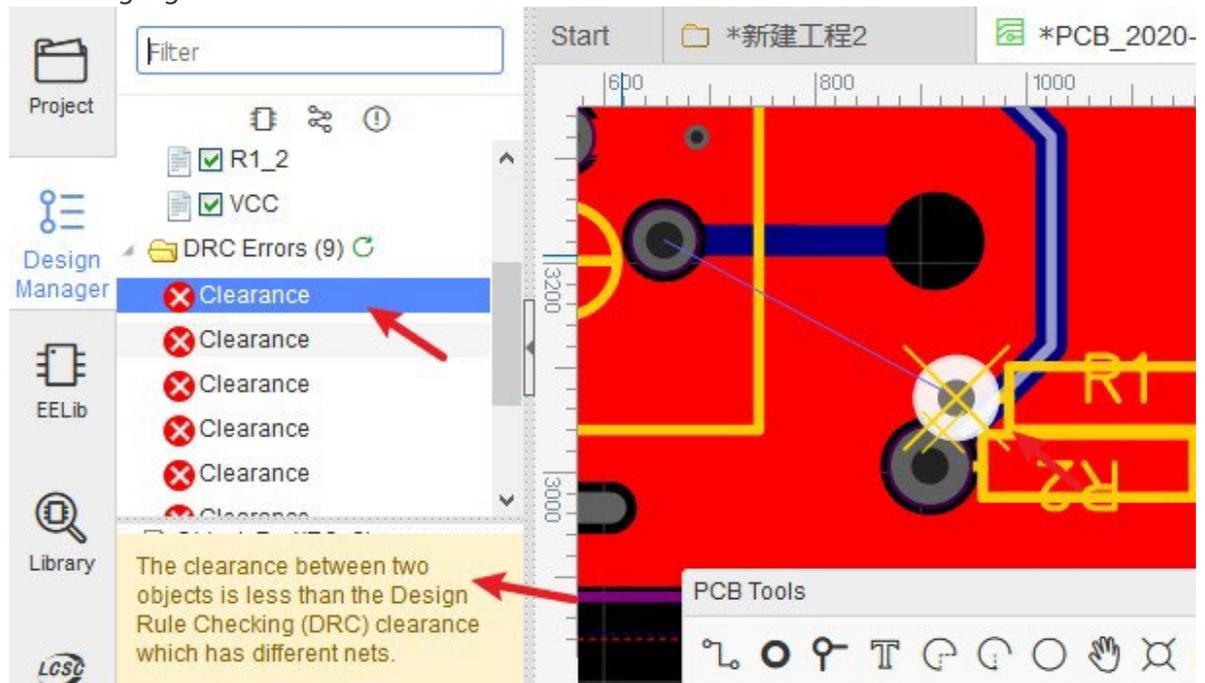
1. Click the "new" button to create a rule, or use the default rule
2. Select one or more networks on the right, support holding down the CTRL key for multiple selection, and also can perform keyword filtering and rule classification filtering
3. Then select the rule you want to set in the "set rules" 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.

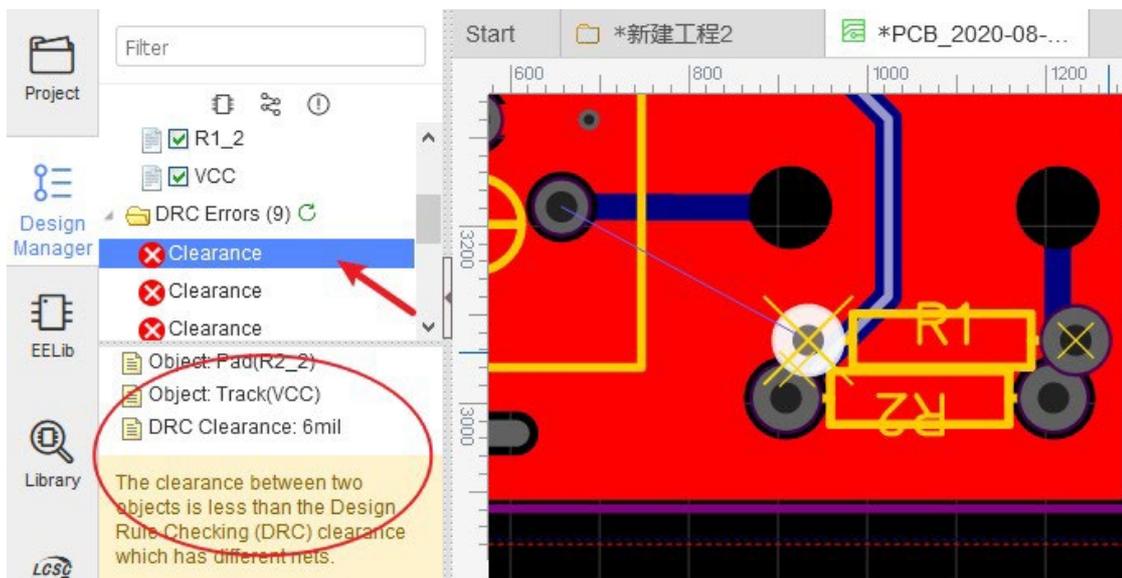


After checking, you can view all the error at the "DRC Error", click the error the related objects will be highlighted.

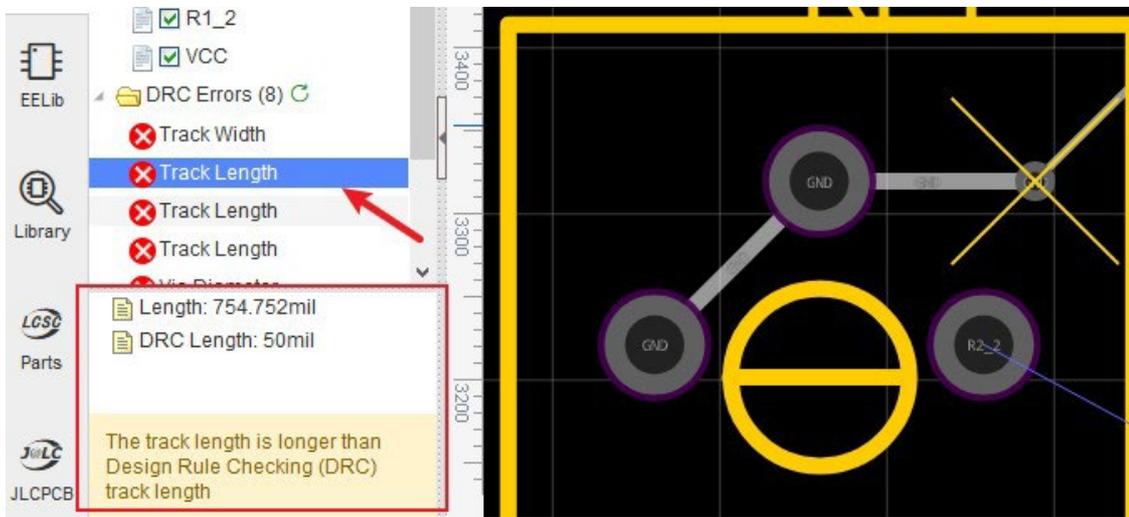


## DRC error type

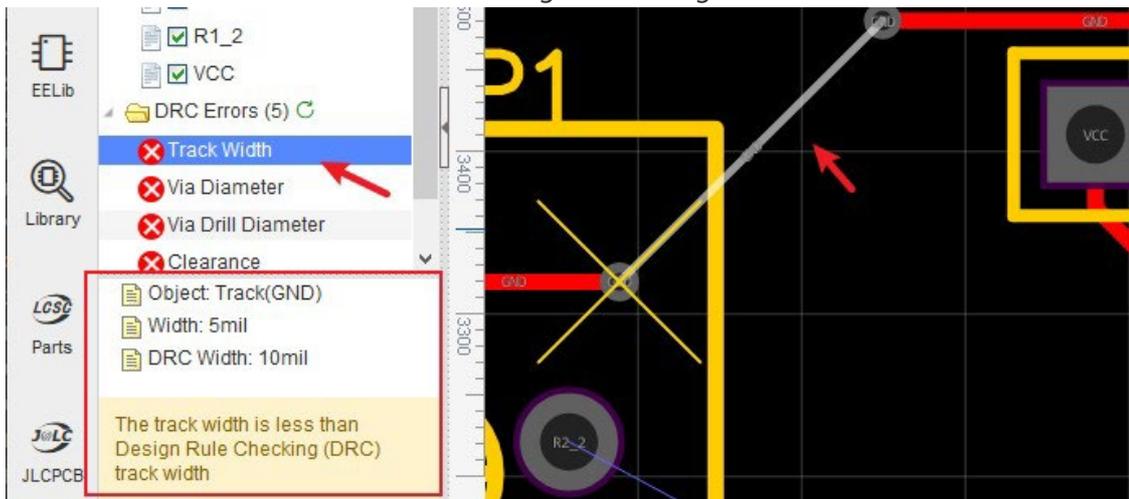
- Clearance: Object to Object. If two different net objects too close, and the distance less than the Design Rule clearance, it will show the Clearance error.



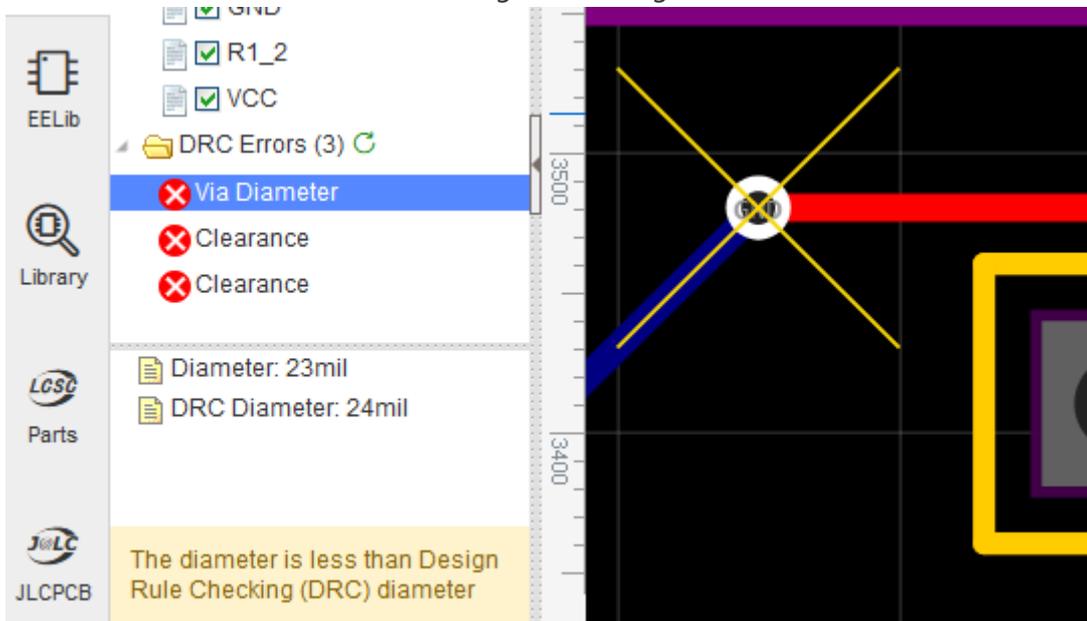
- Track Length: The track Length of the all same net tracks must less than Design Rule track Length.



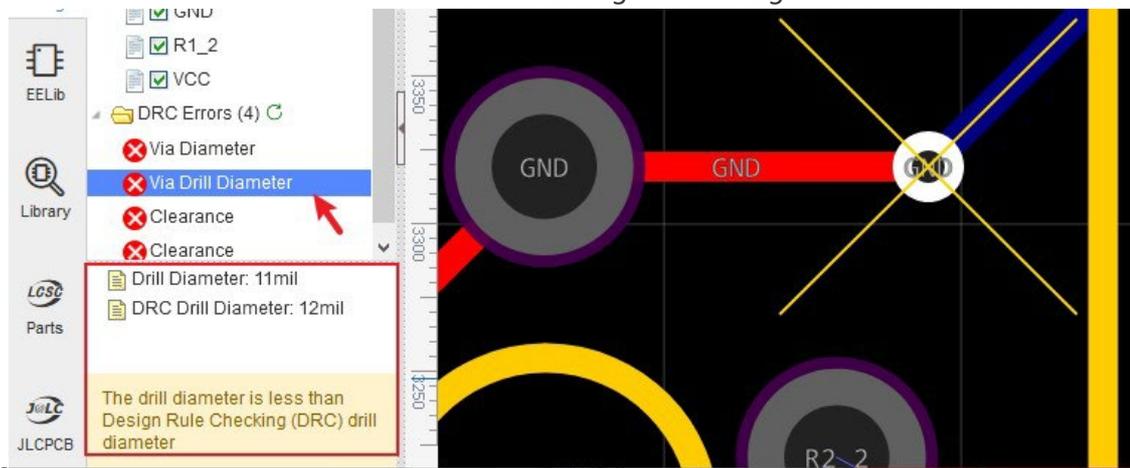
- Track Width: The track width must be larger than Design Rule track width.



- Via Diameter: The via diameter must be larger than Design Rule diameter.



- Via Drill Diameter: The via drill diameter must large than Design Rule drill diameter.

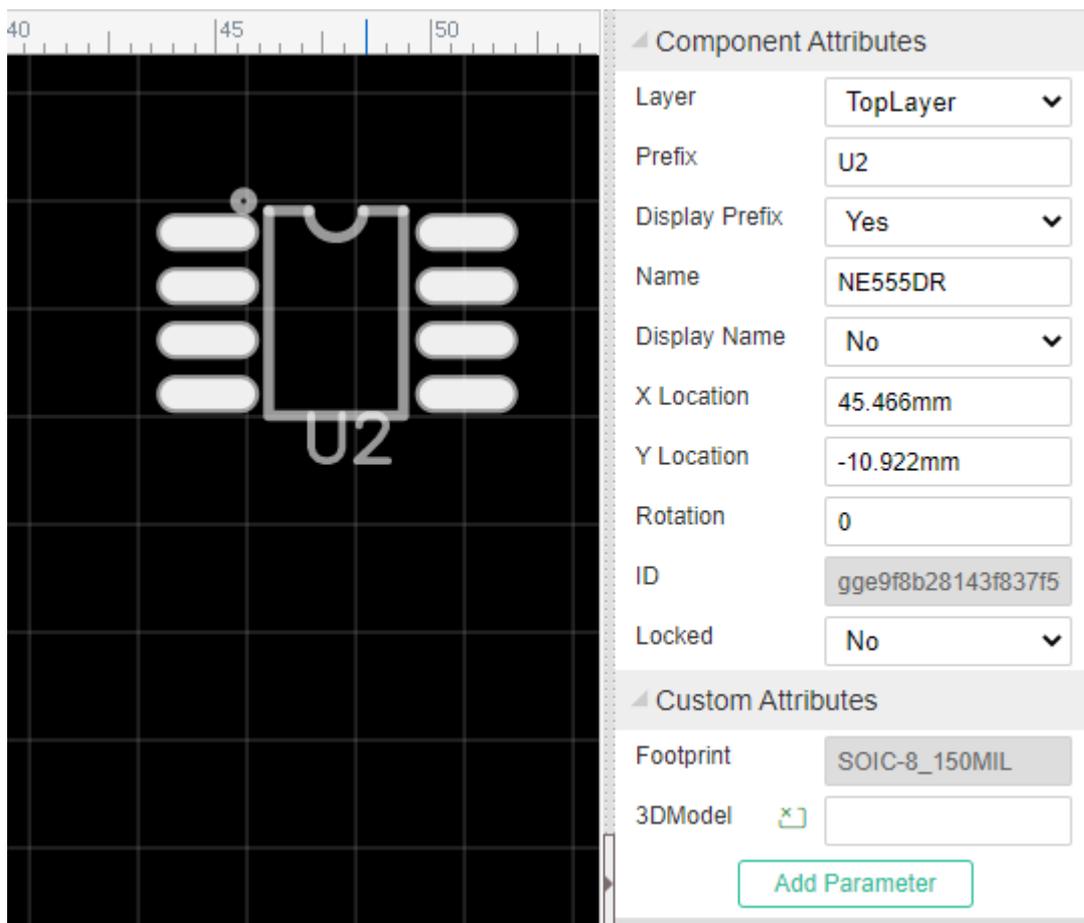


**Note:**

- 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 1o 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.

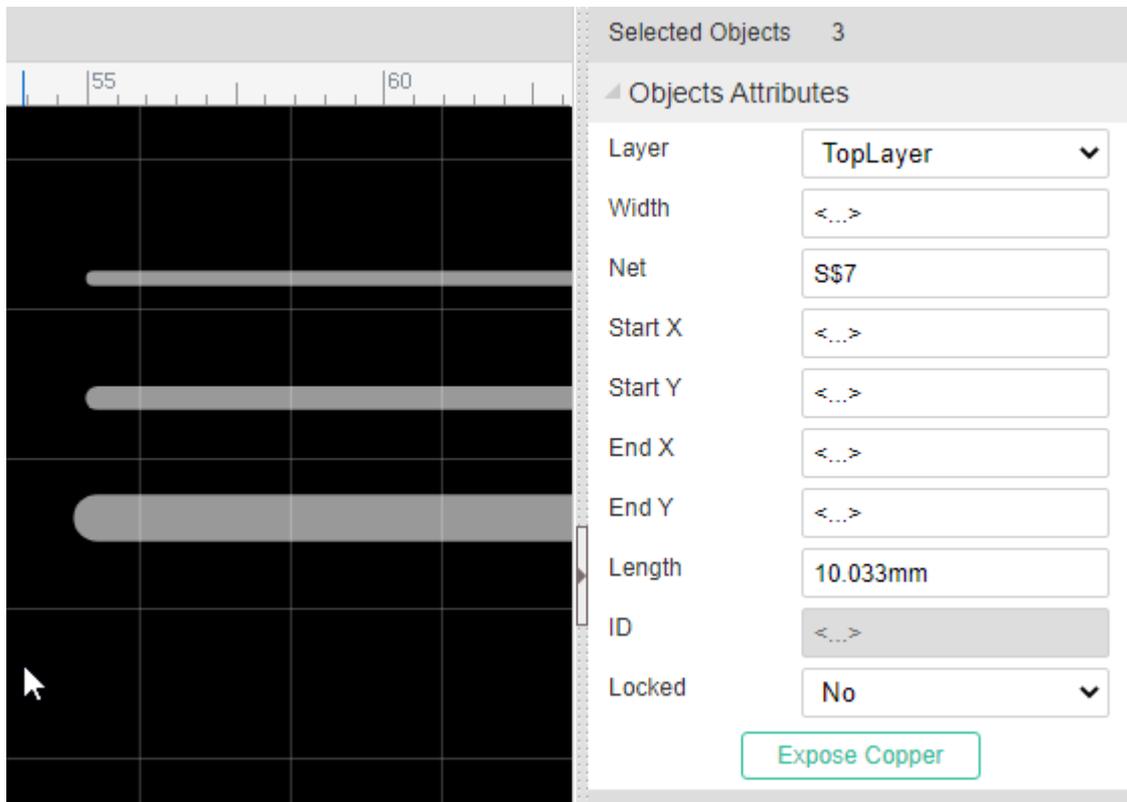
**Lock:** when locked the object, you can not move it by mouse, but if you set the X Y at the right-hand property panel, the object still moves, and the Locked will lock the object layer too.

### 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 seleted objects.



---

## Design Manager

---

Just like Schematic's Design Manager, PCB's Design Manager can be found via:

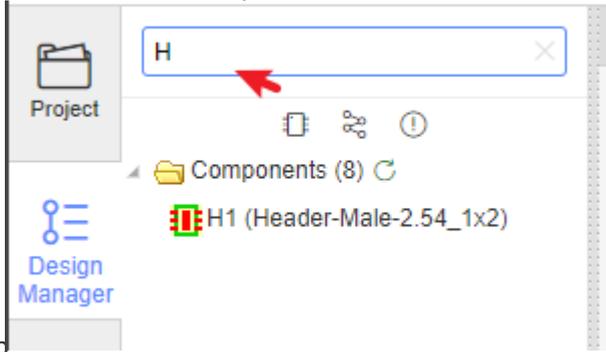
**Left Navigation panel > Design**

or just press the **CTRL+D** hotkey to open the Design Manager dialog.

Design Manager function:

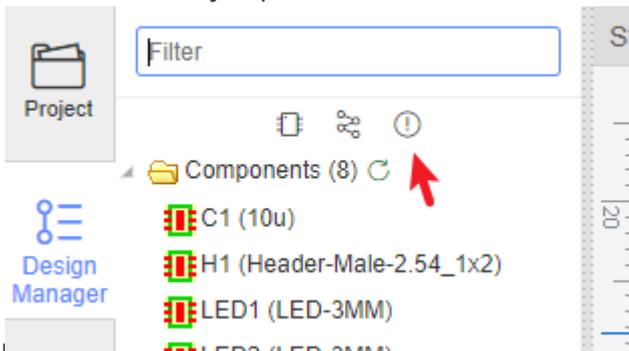
- Filter

- Filter to find a component or net.



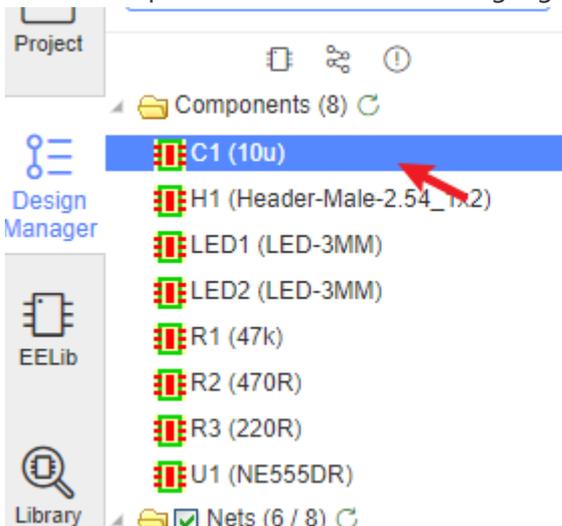
- Jump

- Click the icon to jump to folder.

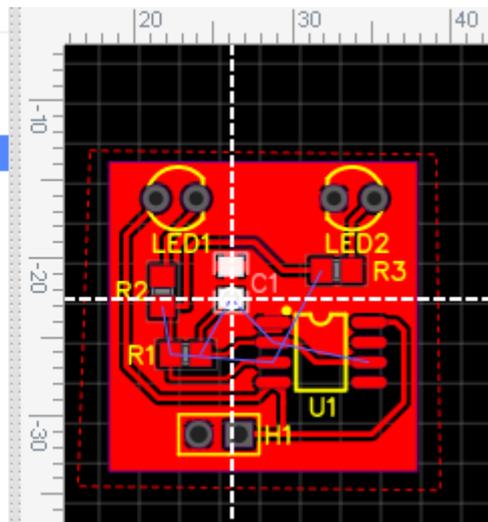


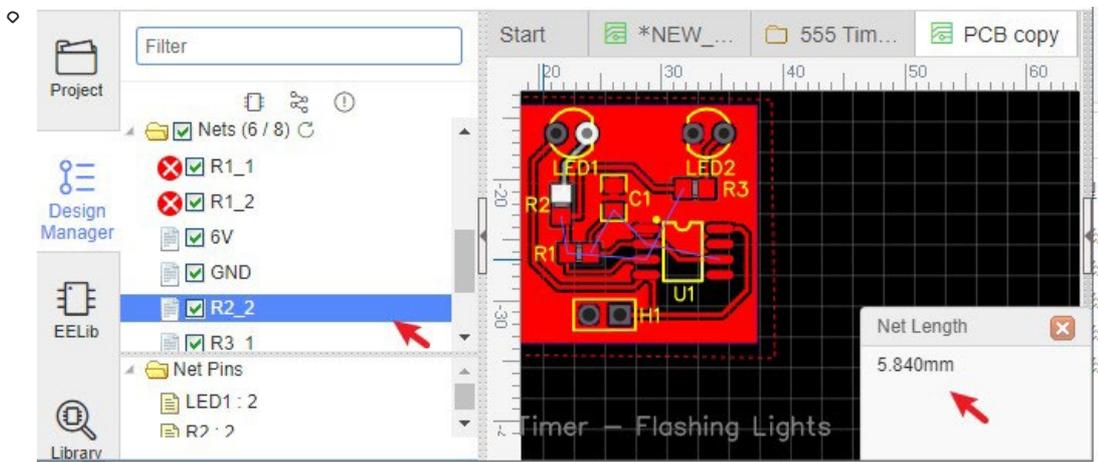
- Component

- Click a component/Net/DRC Error to highlight it.

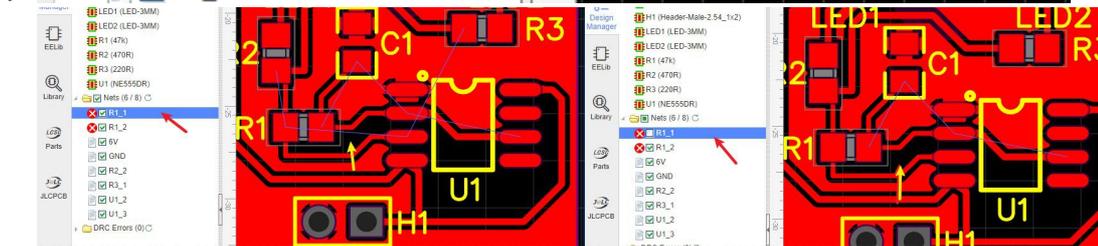
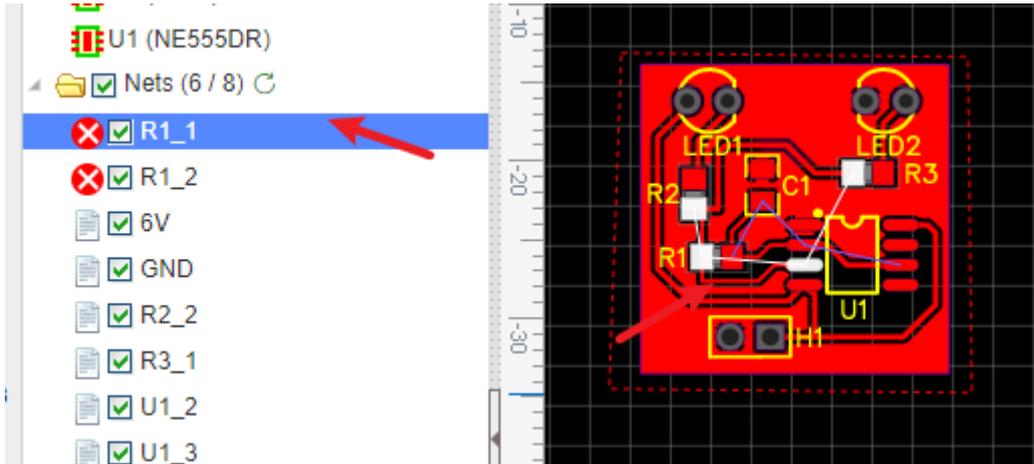


- Net

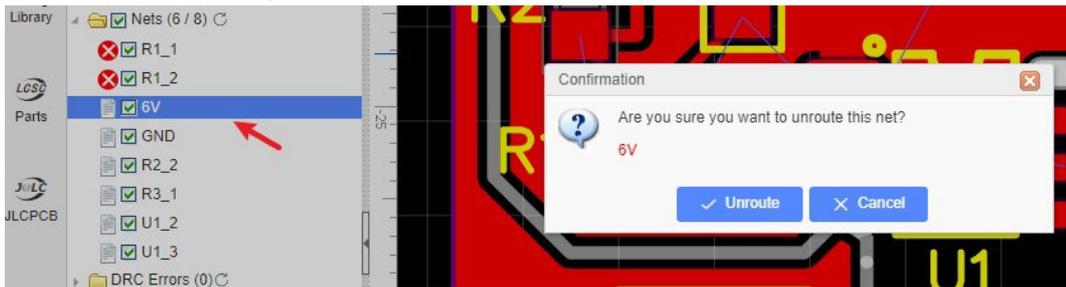




- Click a incomplete net will highlight the ratline and objects.

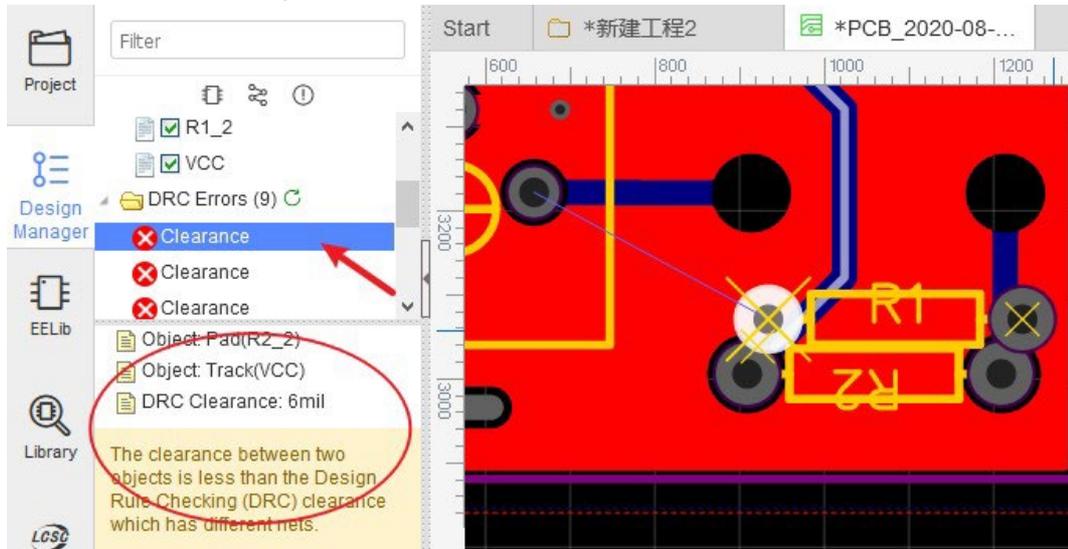


- Double click the net to remove all of the tracks and vias with the net name. If you want to reroute a net, this is the recommended method to use to un-route it first.



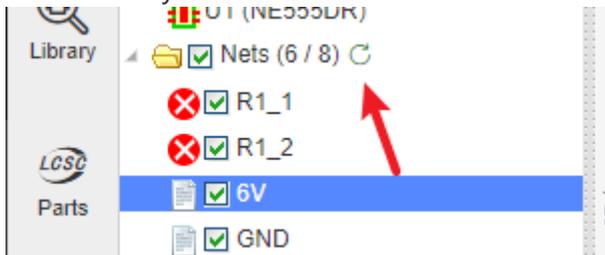
- DRC

- o Click the DRC list, will position the DRC mark on the canvas.



Notice:

- Design Manager list doesn't support to refresh automatically, you must click the refresh icon manually.



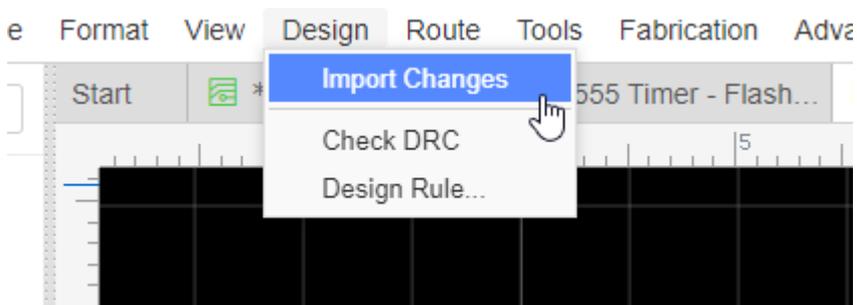
## Import Changes

### Import Changes

Sometimes, while working on a project, you need to make changes to the schematic and then update your board, to incorporate them.

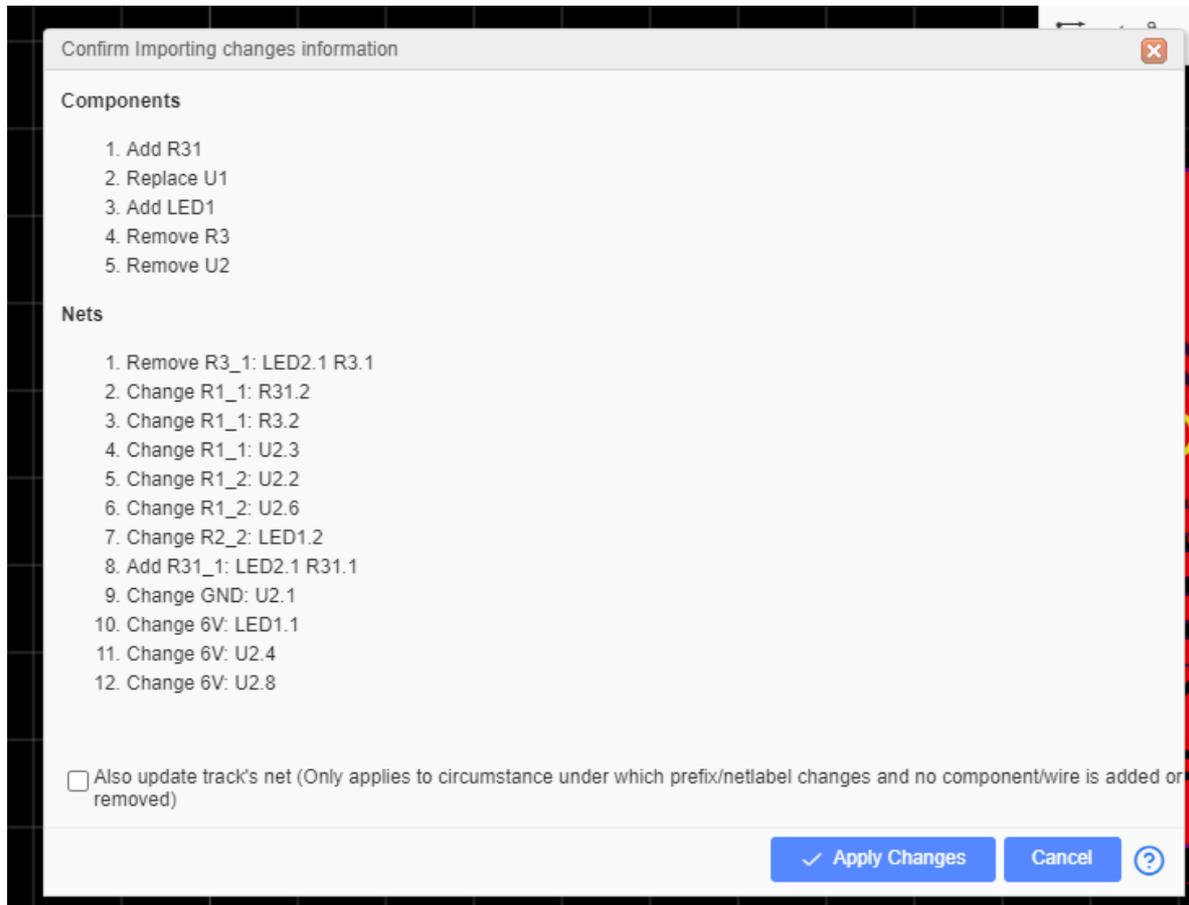
It's easy to do this with EasyEDA.

Go to the **PCB Editor**, via: **Top Menu > 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 schematic, 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 seleted, you can change them net at the right-hand property panel.
- After Import Changes, there are some action can not be undo.

---

## Panelize

---

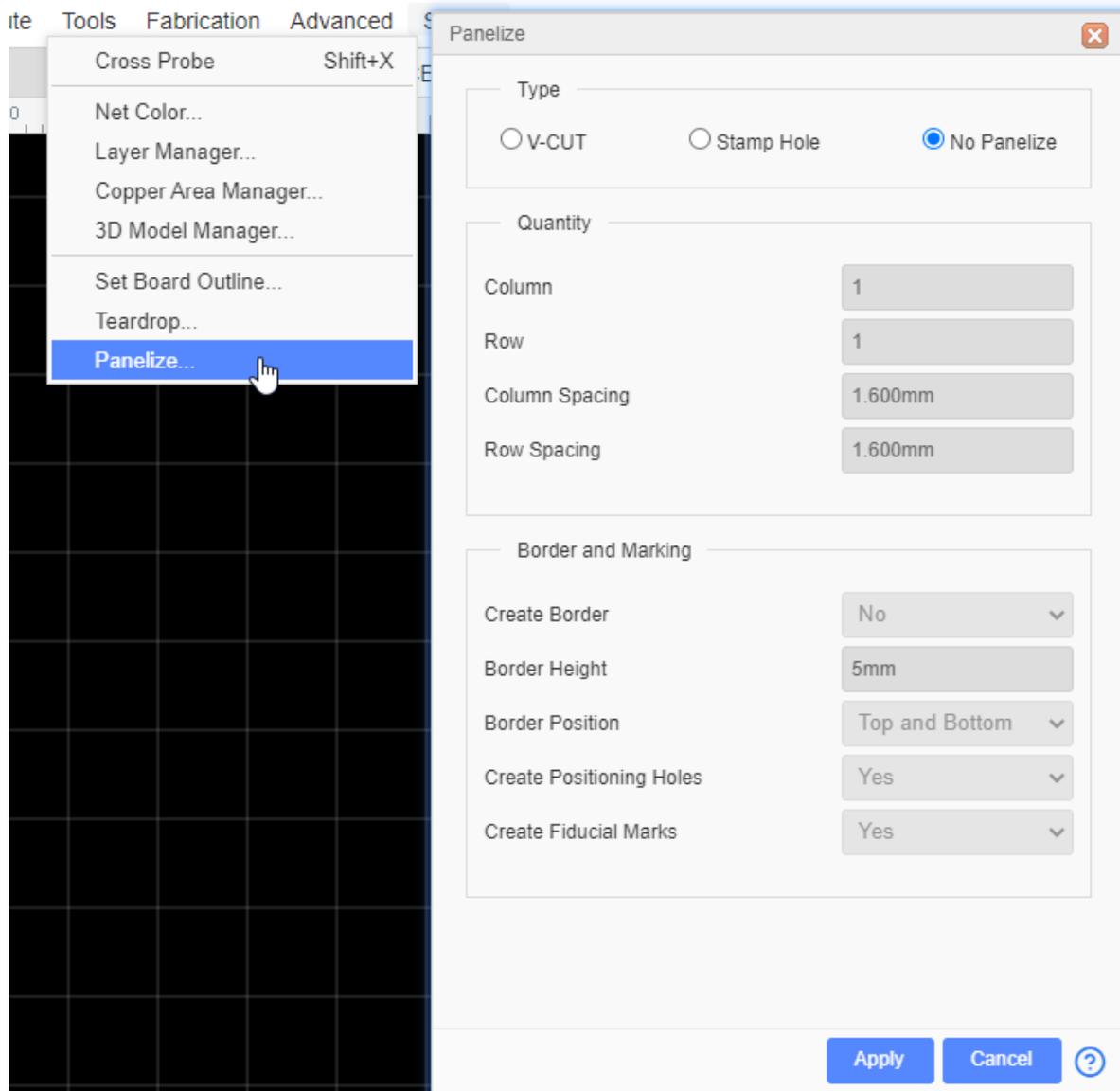
via: Top Menu - Tools - Panelize

## 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.

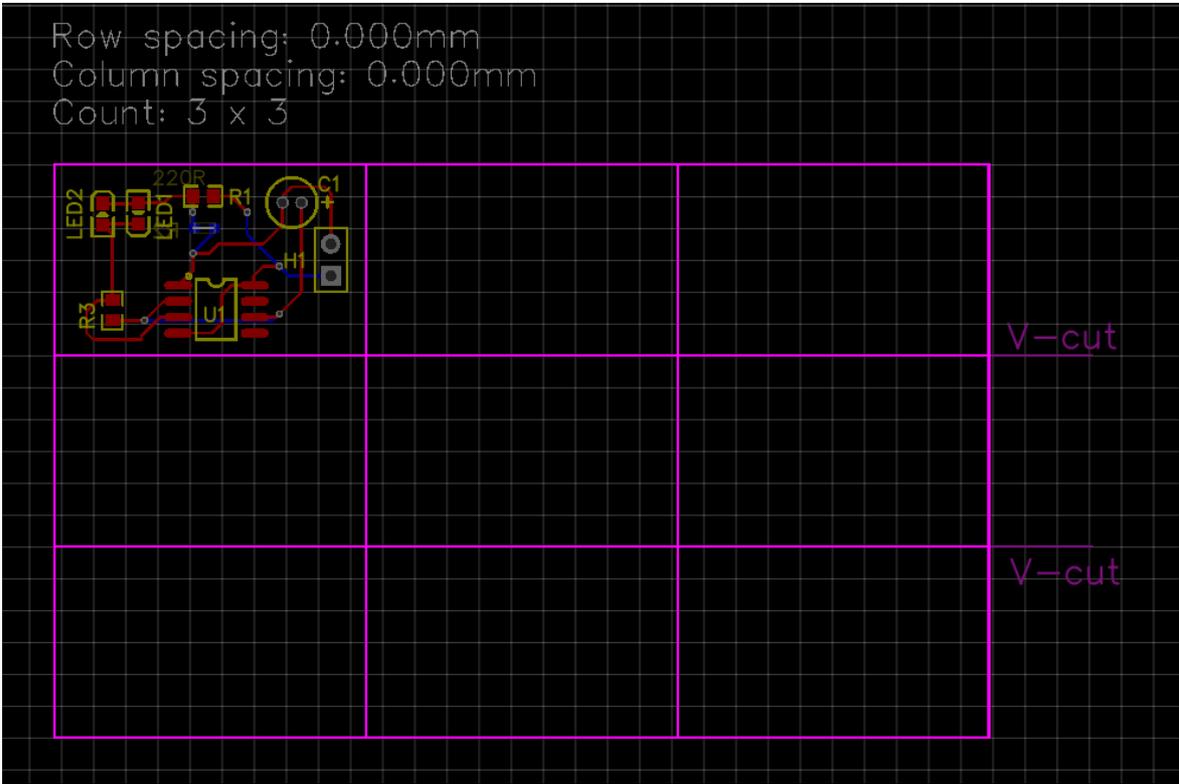
via: **Top Menu - Tools - Panelize**



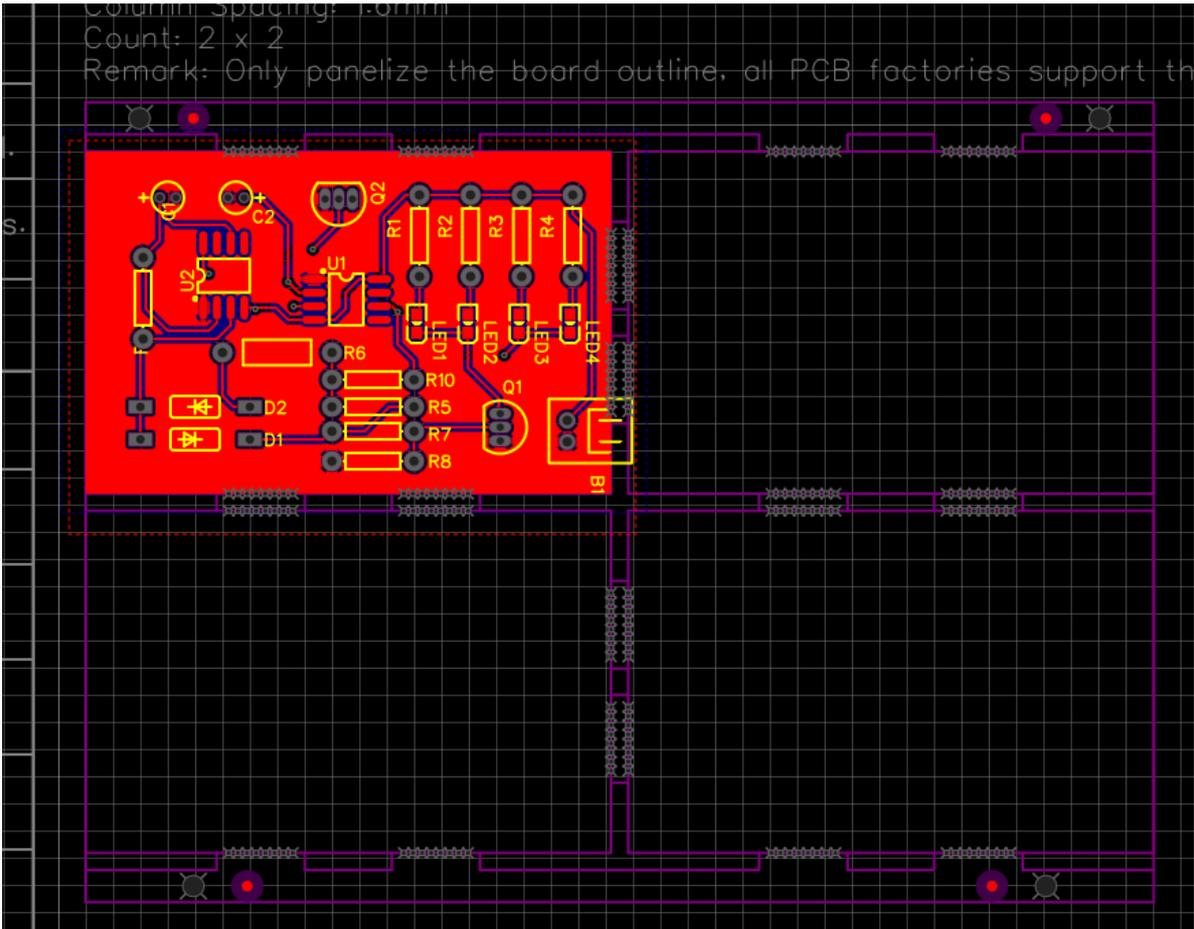
The Border height can not less then 3mm.

#### **V- cut:**

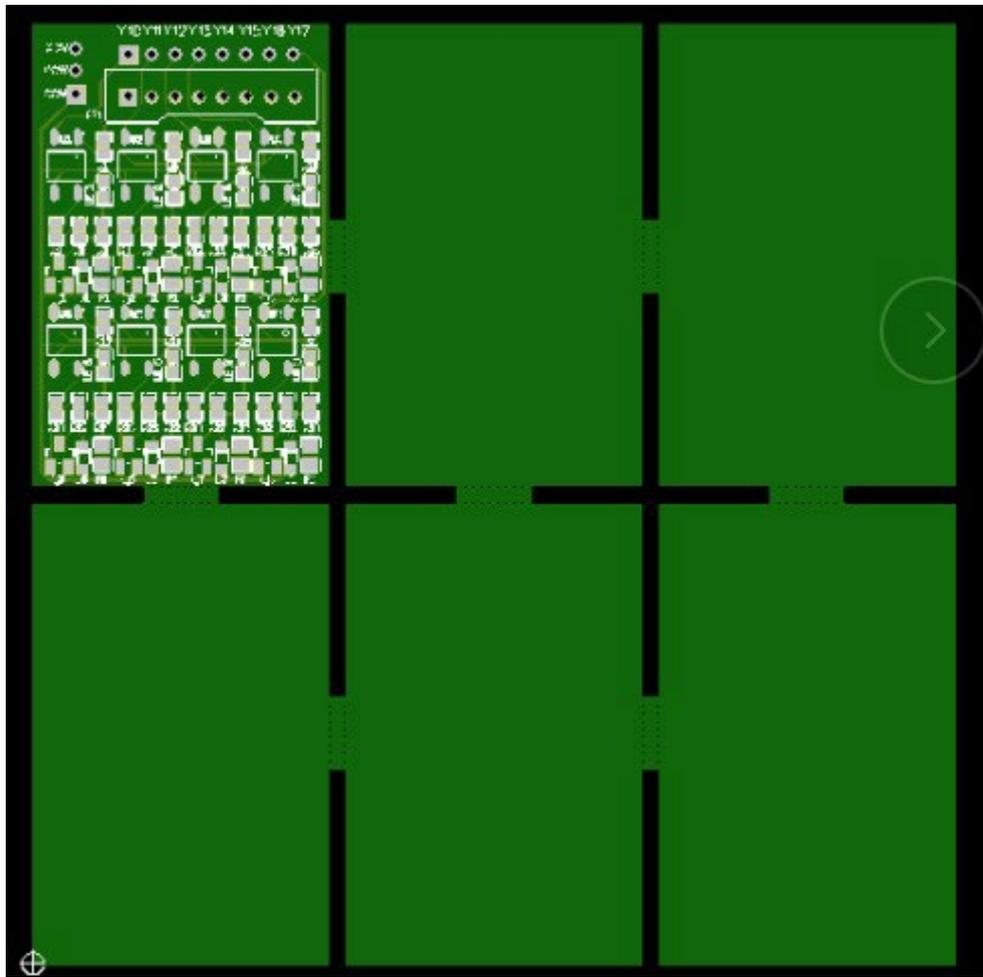
If you choose V-Cut, the editor will add the v-cut indication track on Board Outline layer at the Gerber.



**Stamp Hole:**



When you preview the Panelize Gerber at JLCPCB.com, you will get the image like below:



JLPCB 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`. You can only copy and paste the board outline to become the panelize board.
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.

---

## Layout a PCB Without Schematic

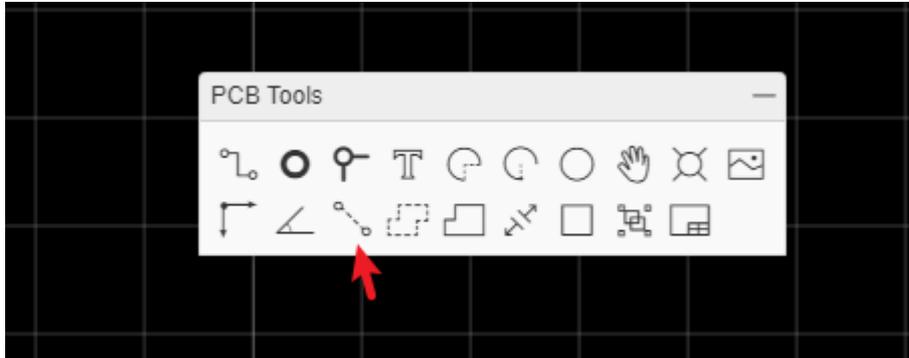
---

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.



---

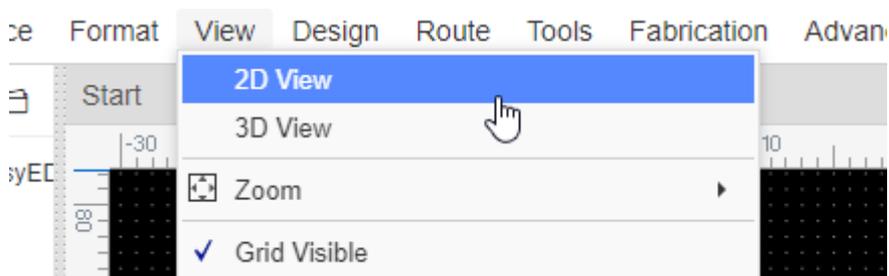
## PCB Preview

---

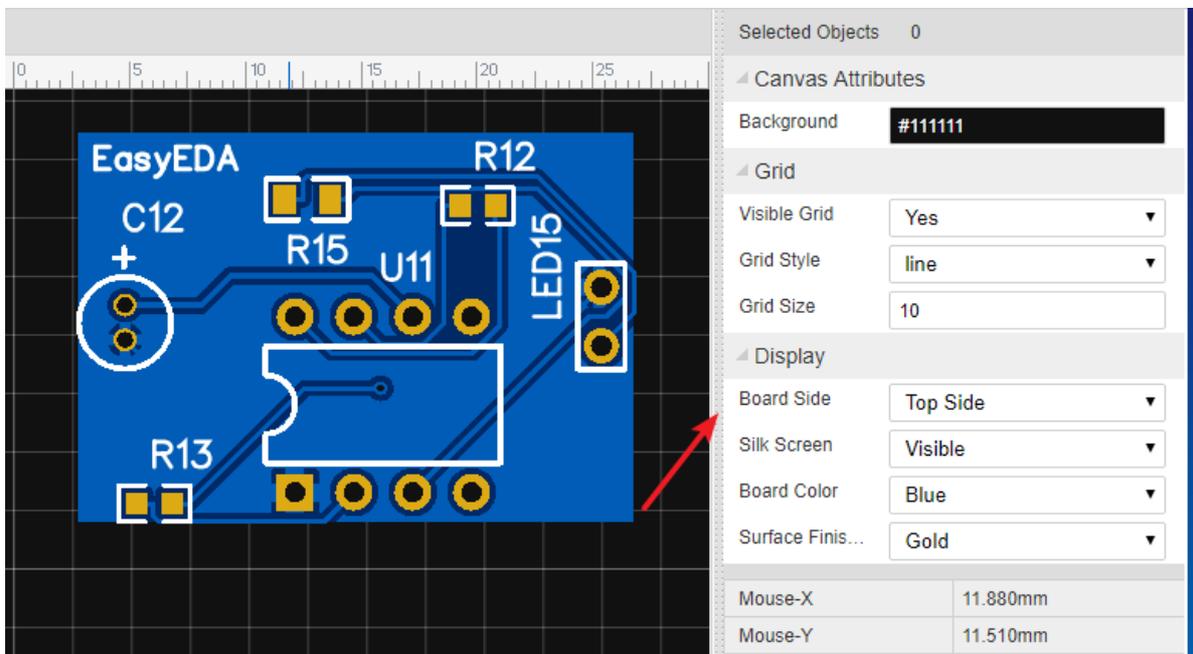
### 2D View

EasyEDA provide a nice Photo View to help you to check the PCB.

Via: Top Menu - View - 2D View.

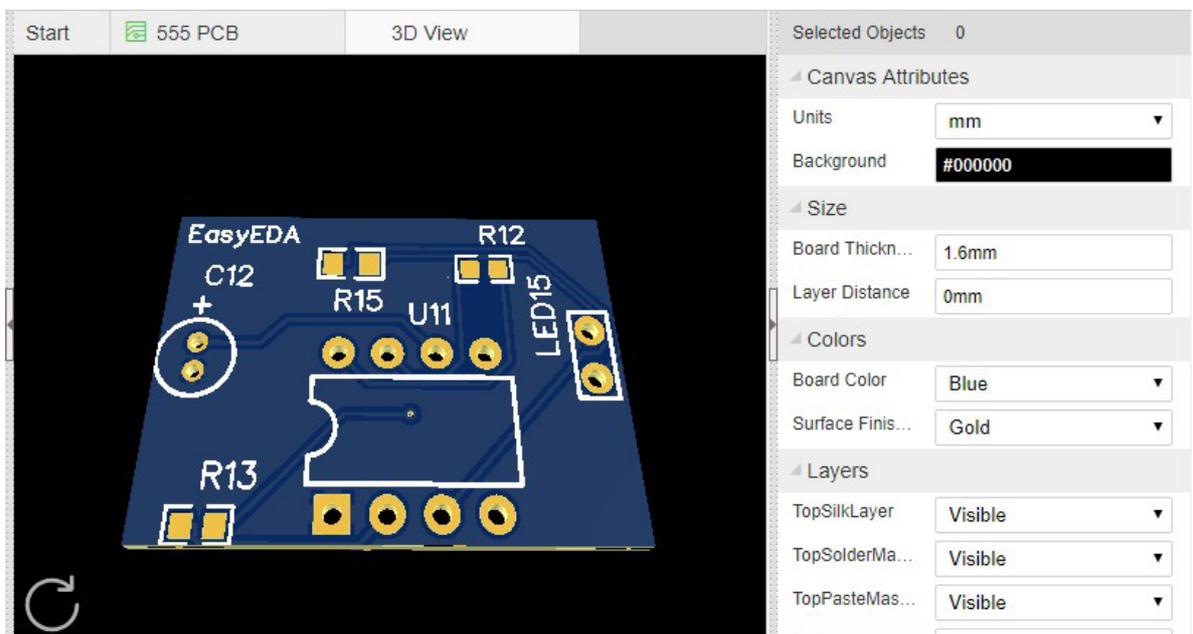


After converting the PCB to Photo View, you can see the result as in the image below.



## 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.



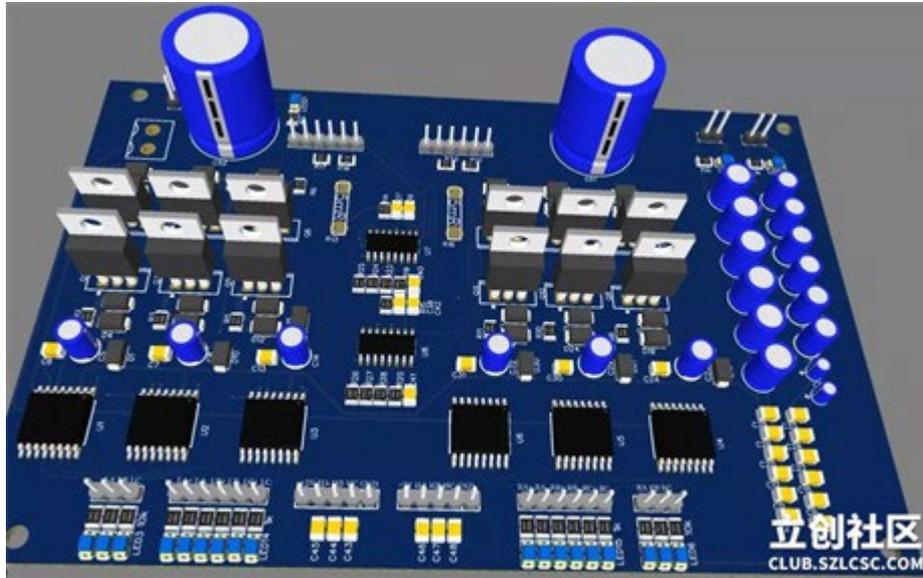
- Change 3D view attributes at the right-hand panel;
- Reset the 3D PCB position at the left-bottom corner icon;
- Keep left-click and drag the canvas can change the view direction;
- Keep right-click and pan can change the 3D PCB position.

3D model view of the component please check "PCB - 3D Model Manager" and "Footprint - Import 3D Model" chapter.

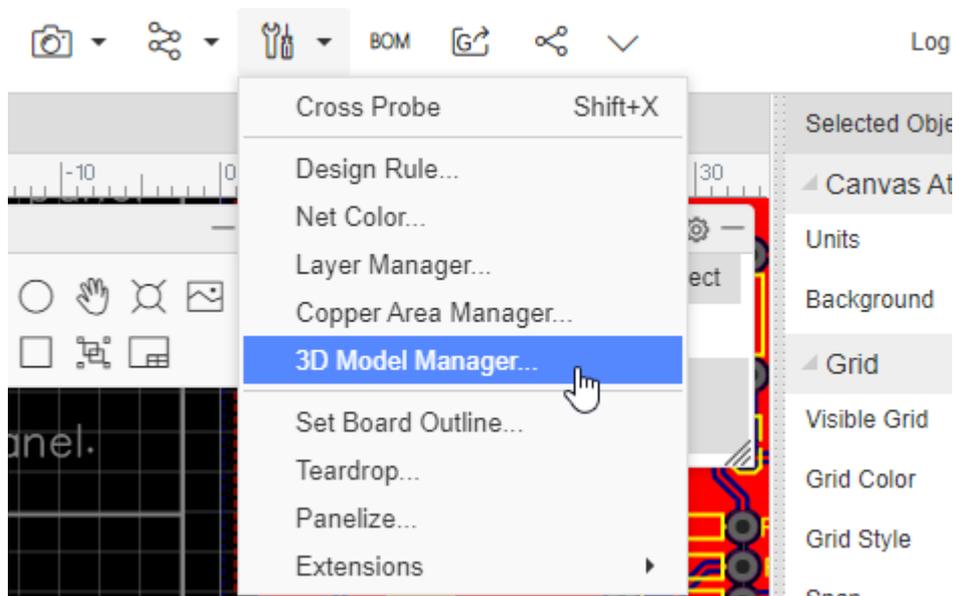
# Experiment no: 16

## Experiment Title: 3D modeling in EasyEDA

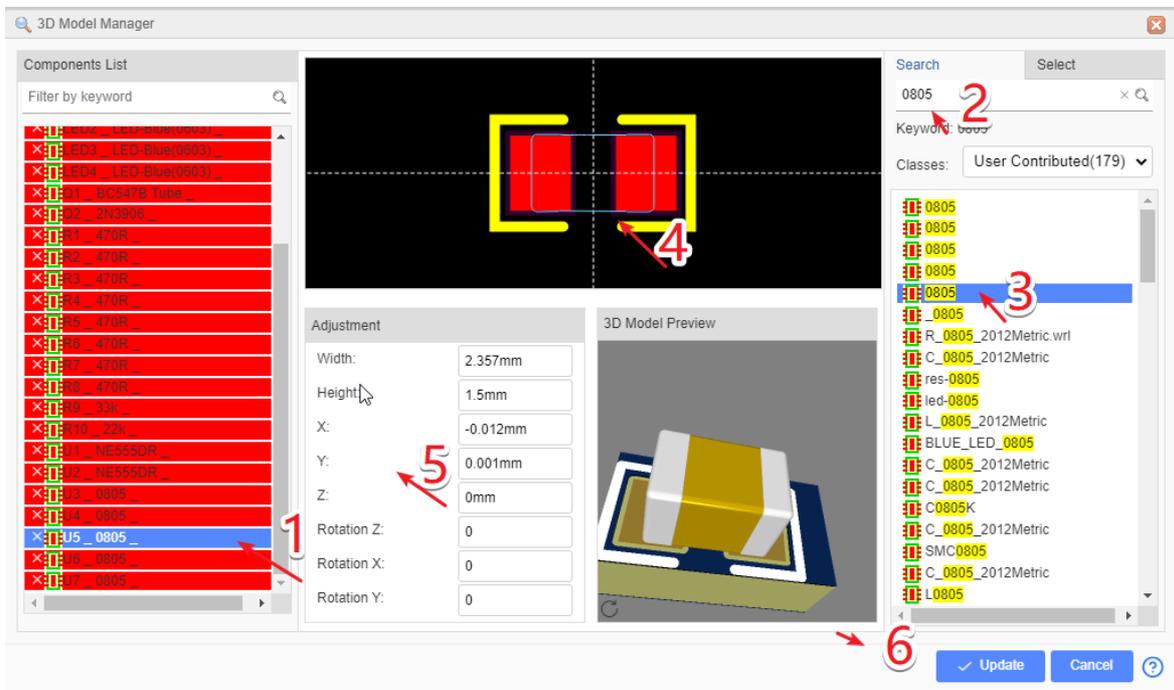
EasyEDA supports for importing 3D models, PCB can view cool 3D models when doing 3D preview. Exporting PCB 3D model files is not supported yet.



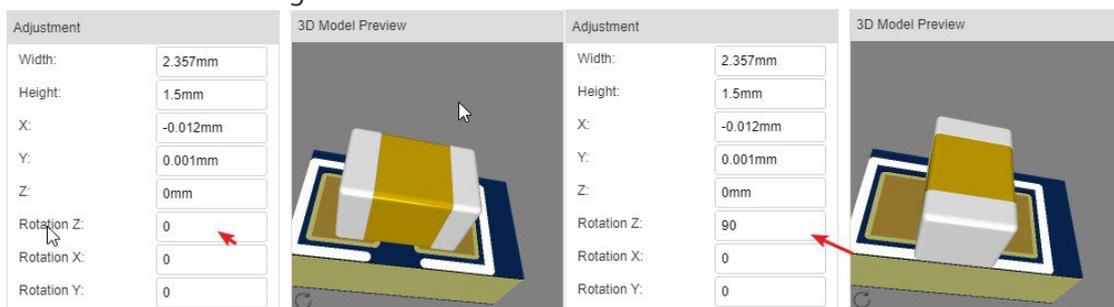
Open 3D model manager: - tools - 3D model manager



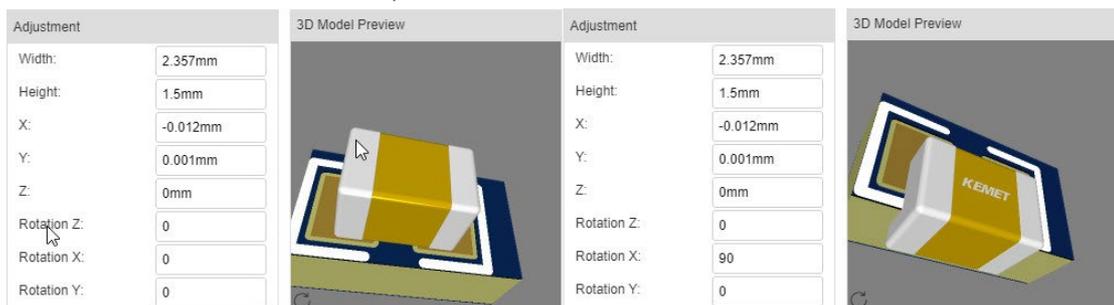
When you open it, you can bind 3D models for Footprint, and you can import or search for user-contributed 3D models. Import tutorial please see: [Footprint - Import 3D Model](#)



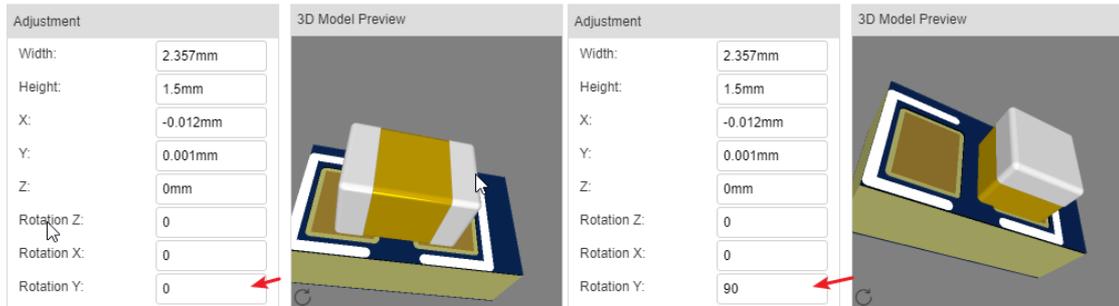
1. Click the footprint in the list of footprints on the left, and the preview of footprints will be displayed in the central area. Support multiple selection: hold down CTRL + Mouse click selection; Hold SHIFT + click select.
2. Select the imported 3D models from the list on the right or directly search the 3D model uploaded by the user, and search by keyword.
3. Select a 3D library from the list of 3D model libraries and display 2D outline of the top view of the 3D model in the middle preview area.
4. Adjust parameters:  
 drag the mouse to align the 2D boundary line with the footprint shape; You can also adjust the way you enter parameters below. The right side of the parameters can be previewed directly, and it supports a long left - click drag 3D preview interface.  
**width/height:** the width/height of the 2D shape of the 3D model  
**X and Y:** the X and Y coordinates of the 2D shape of the 3D model  
**z-axis rotation:** in the overhead view, the 3D model rotates anticlockwise. The editor automatically recognizes the width and height of the footprint and automatically sets the z-axis rotation to 90 degrees.



X-axis rotation: in the side view, the 3D model rotates counterclockwise



Y-axis rotation: when facing the image, the 3D model rotates anticlockwise

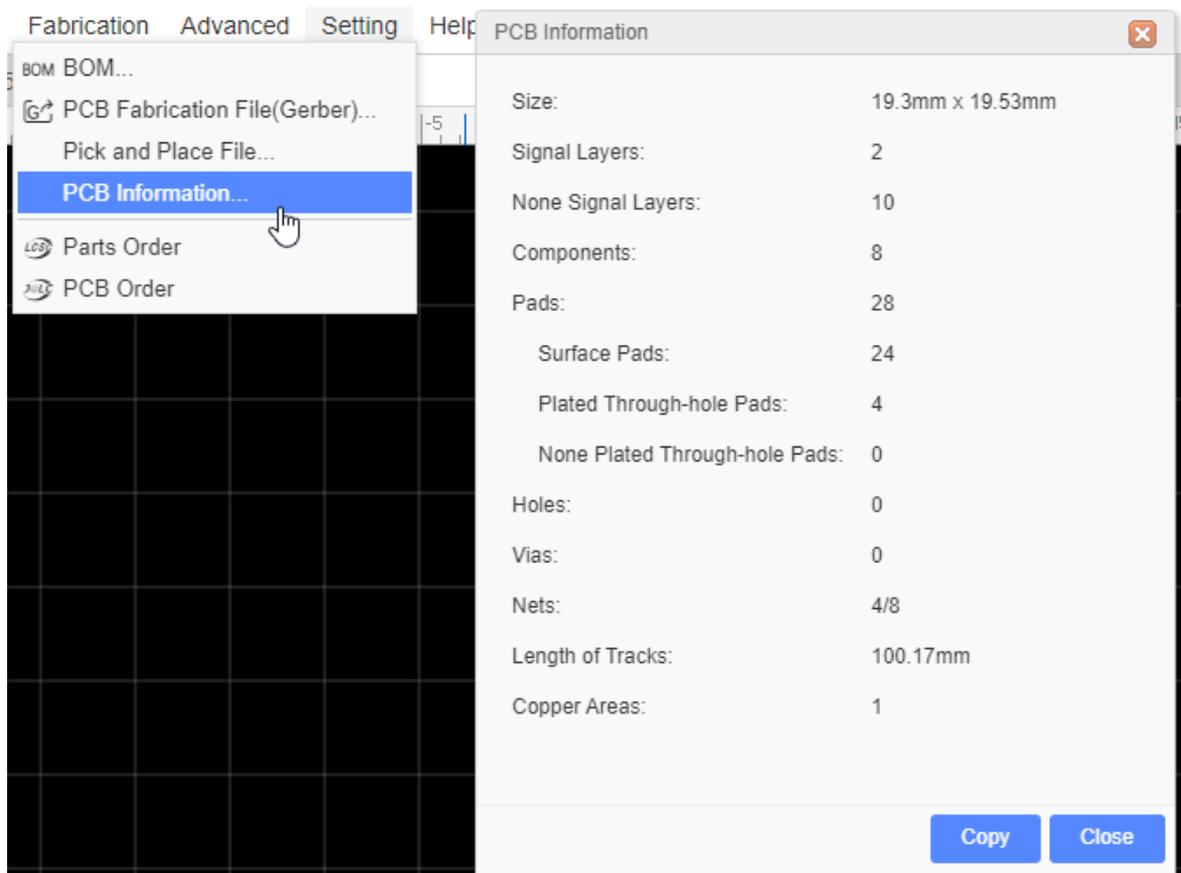


5. After the adjustment, click the update button to complete the 3D model binding.
6. Click 3D preview from the preview menu at the top to preview the 3D model.

## PCB Information

PCB design information can be easily obtained by checking PCB information.

Entry: Top Menu - Fabrication - PCB Information



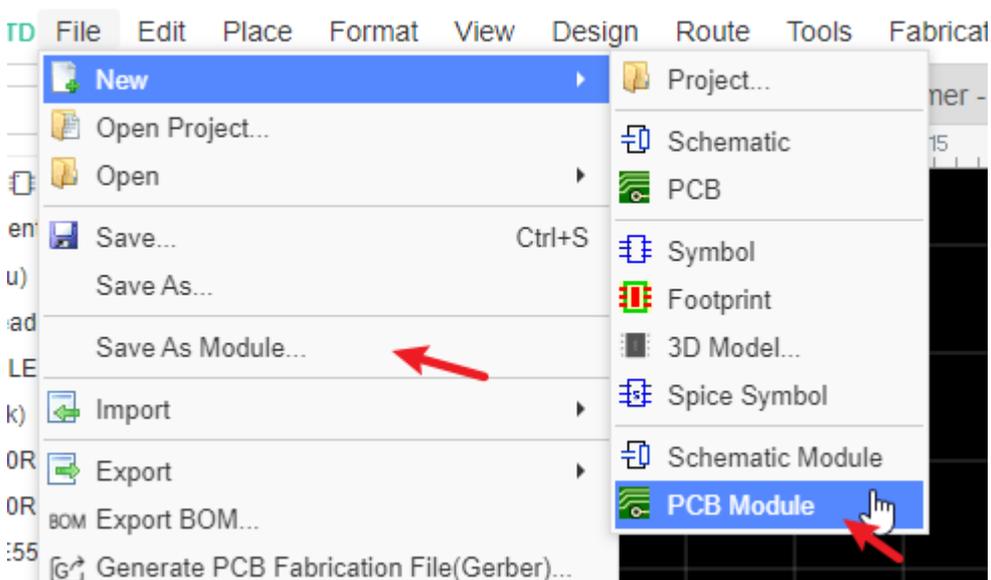
Nets shows: routed nets/total nets.

## PCB Module

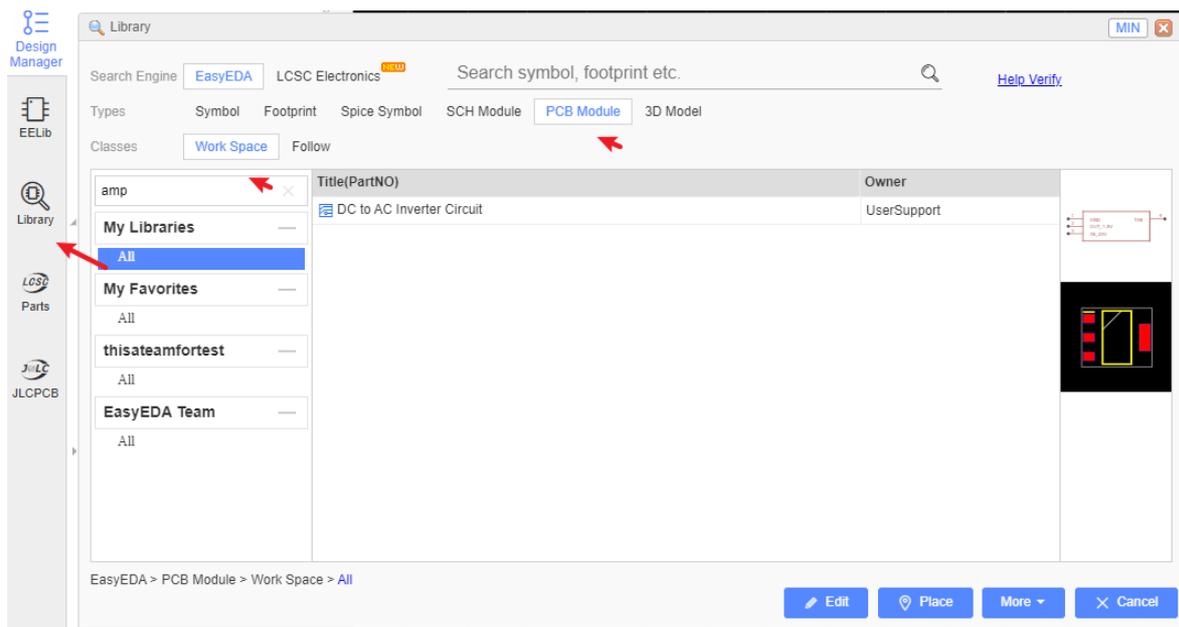
EasyEDA support create the PCB modules, it seems schematic module.

## 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**



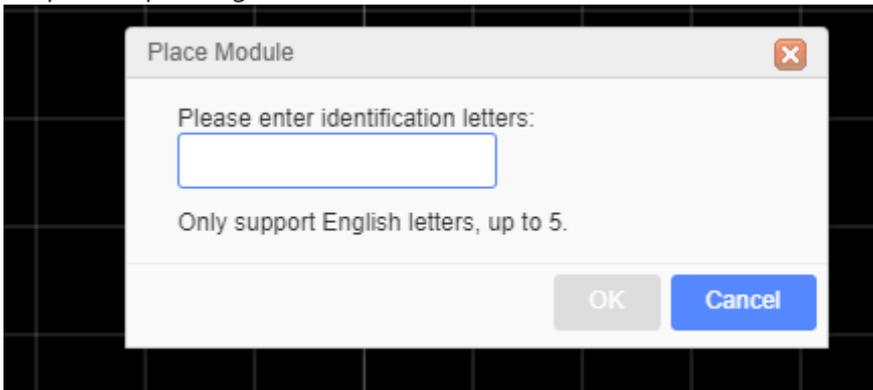
## How to use

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 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.

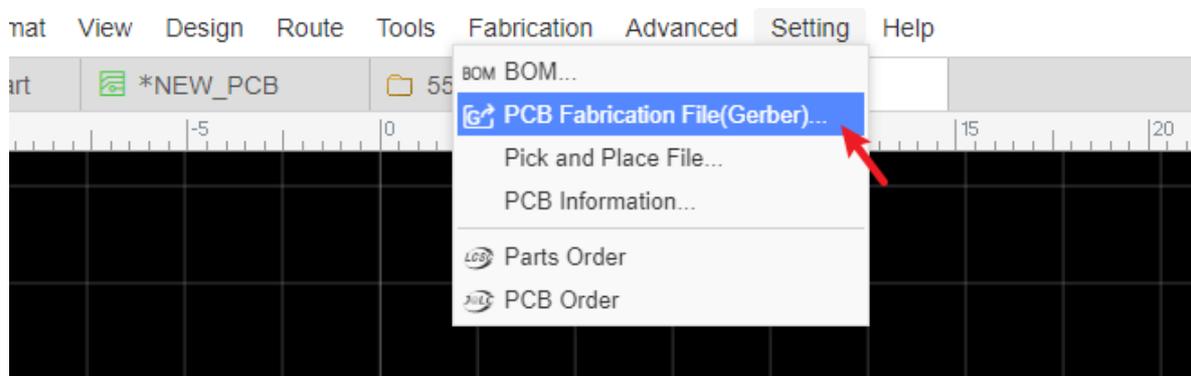
---

## Generate Fabrication File(Gerber)

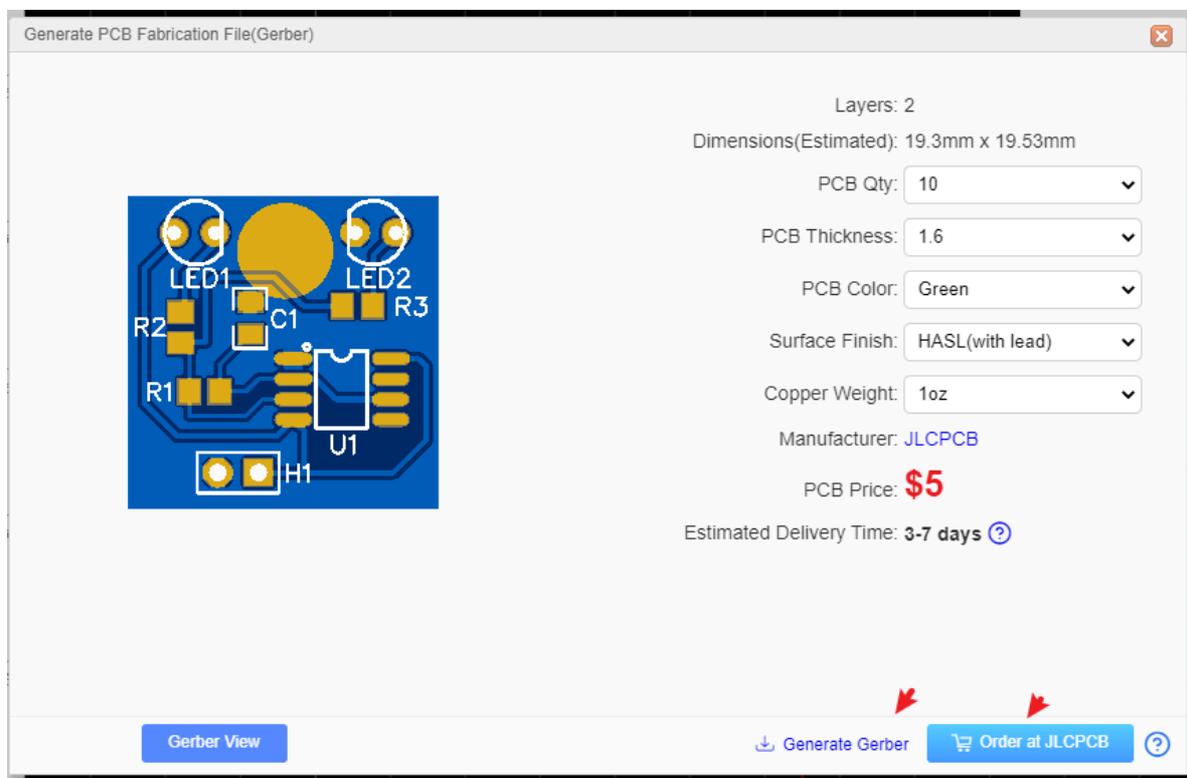
---

### 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\_BoardOutlineLayer.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:** Inner copper layer, signal type.
- **Gerber\_Inner2.GP2:** Inner copper layer, plane type
- **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.
- **Drill\_PTH\_Through.DRL:** Plated drill through hole layer. This document shows the location of the hole where the inner wall needs to be metallized. Old name: Gerber\_Drill\_PTH.DRL
- **Drill\_NPTH\_Through.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. Old name: Gerber\_Drill\_NPTH.DRL
- **Gerber\_TopPasteMaskLayer.GTP:** Top Paste Mask, for the stencil.
- **Gerber\_BottomPasteMaskLayer.GBP:** Bottom Paste Mask, for the stencil.
- **Gerber\_TopAssemblyLayer.GTA:**Top Assembly, read only, doesn't affect the PCB manufacture. Old name: ReadOnly.TopAssembly

- **Gerber\_BottomAssemblyLayer.GBA:** Bottom Assembly, read only, doesn't affect the PCB manufacture. Old name: ReadOnly.BottomAssembly
- **Gerber\_MechanicalLayer.GML:** Record the information on the mechanical layer in the PCB design, and only use it for information recording. Old name: ReadOnly.Mechanical. 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. If the mechanical layer has closed wires, JLCPCB will give priority to using the mechanical layer as the shape of the board when producing the board. If there is no outer frame of the mechanical layer, GKO will be used as the frame (historical influence of Altium file). It is necessary to pay attention to the use of the mechanical layer in the design.

**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!
- The coordinates of the Gerber file follow the canvas coordinates
- When exporting Gerber, the coordinate format accuracy defaults to 3:3. When the PCB size is out of range, it automatically uses 4:2 format. If you view the Gerber as such as CAM350, found that the Drill hole has been offset the location, you can modify the drill coordinate format to fit the location

## 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.

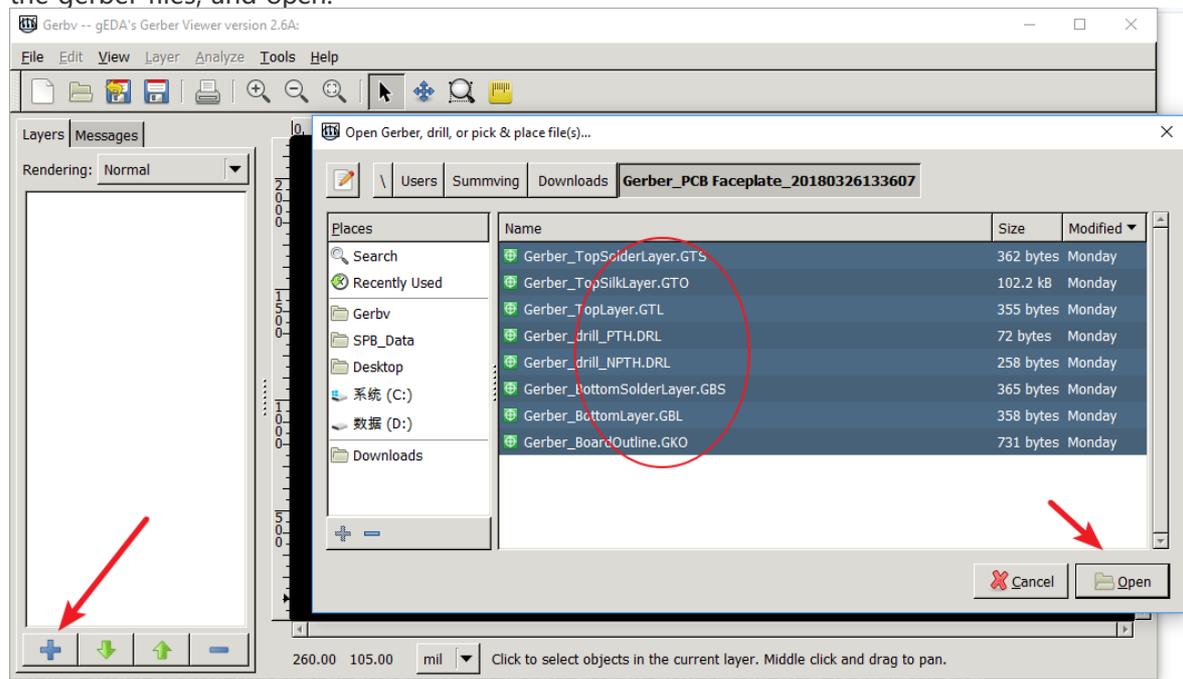
Gerber viewer recommend Gerbv:

- 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.



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:

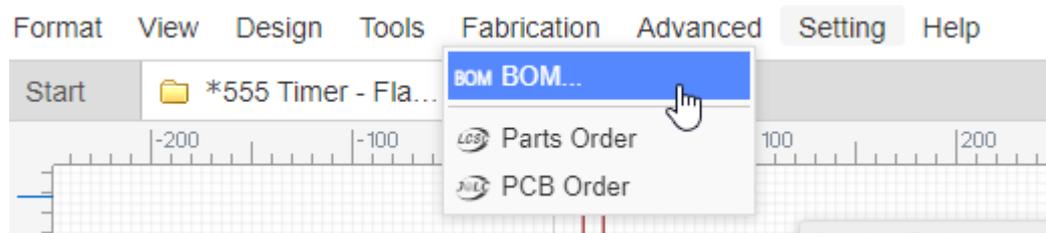
[jlpcb.com](http://jlpcb.com)

[tracespace.io/view](http://tracespace.io/view)

[gerber.ucamco.com](http://gerber.ucamco.com)

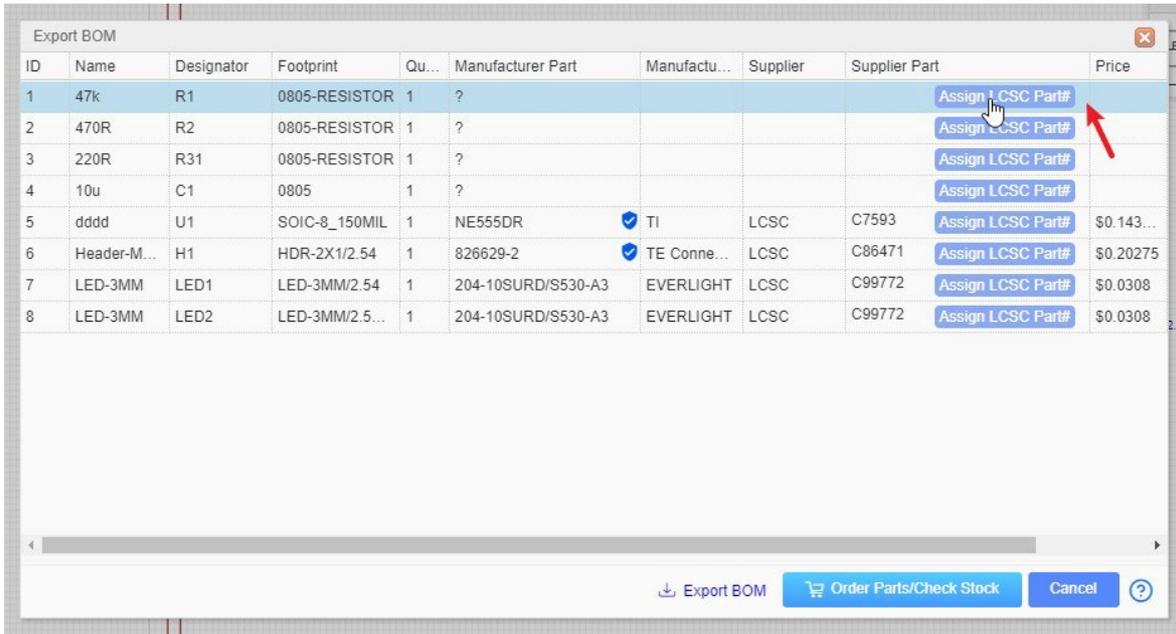
## Export BOM

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".

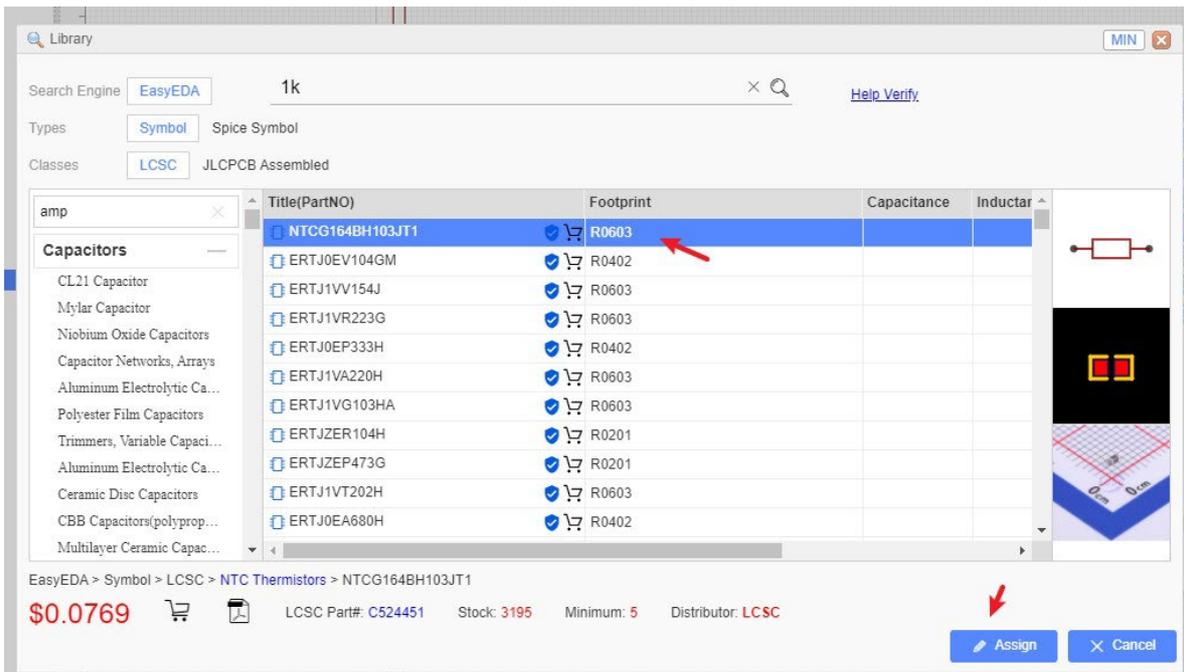


After clicking the BOM export option, the dialog below will open.

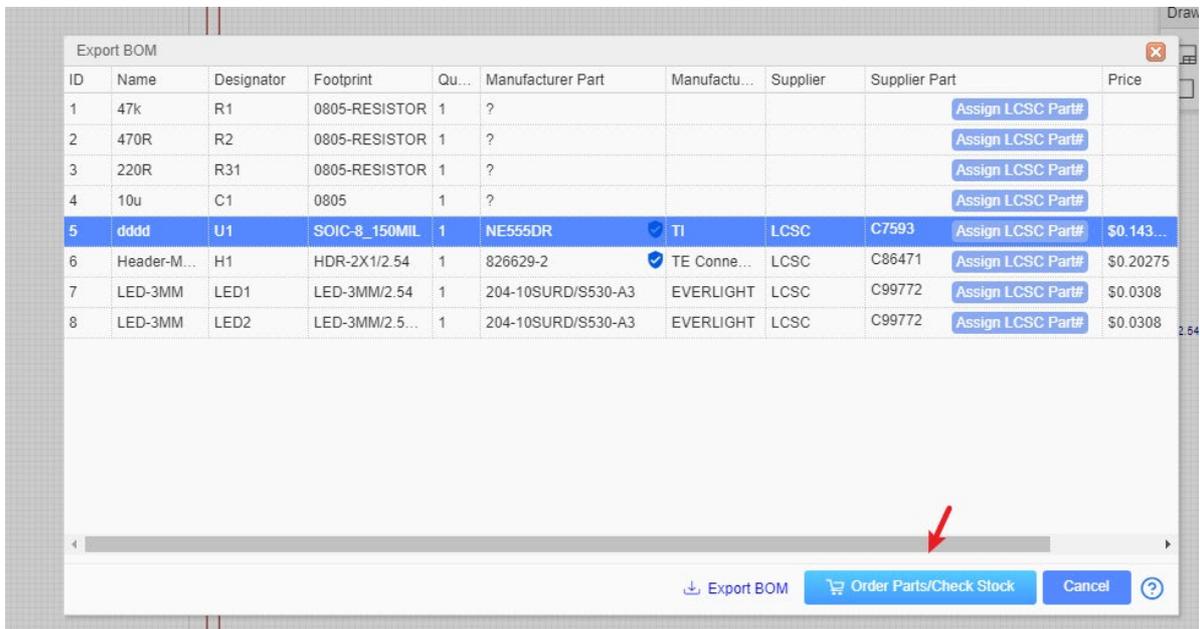
In this dialog, you can click the button to assign LCSC part's order code for your components.



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 form 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.

ID	Name	Designator	Footprint	Quantity	Manufactur	Manufactur	Supplier	Supplier Pa	LCSC Assembly
1	HDR-M-2.54	KJ1,AJ1,BJ1	HDR-M-2.54	8			LCSC	C66690	
2	NE555P~NA	U1	DIP-8	1	NE555P	TI	LCSC	C46749	
3	MC306(6pF	C1	CAP-D3.0X	1	HV010M05C	CapXon	LCSC	C59954	
4	0.1u	C63,C73	C1210K	2					
5	MC306(6pF	C8	C1210	1					
6	19-217/GHC	LED1,LED2	LED0603-R-	2	19-217/GHC	EVERLIGHT	LCSC	C72043	Yes
7	1N4148W	KD1,AD1,BI	SOD-123FL	8	1N4148W	Tak Cheong	LCSC	C129216	
8	CAP-1uF	C2	C0805	1	RVT2A1R0M	HONOR	LCSC	C87863	
9	CAP-1uF	C4	RAD-0.1	1	?				
10	CAP-1uF	C5	R0805	1	?				
11	HDR-IDC-2.	P1	IDC-TH_6P-	1	2X3 2.54mn	BOOMELE	LCSC	C11214	
12	0.1u	KC1,AC1,BC	C1210	8					
13	1KOHM	R2	R0805	1	?				
14	1KΩ	R1	AXIAL-0.3	1	?				
15	2N3906(TO	KQ1,AQ1,BI	TO-92-3_L4	8	2N3906	CJ	LCSC	C9809	
16	1m	KL1,AL1,BL1	L0402	8					

Export BOM supports to export LCSC part price, it is the same as LCSC website.

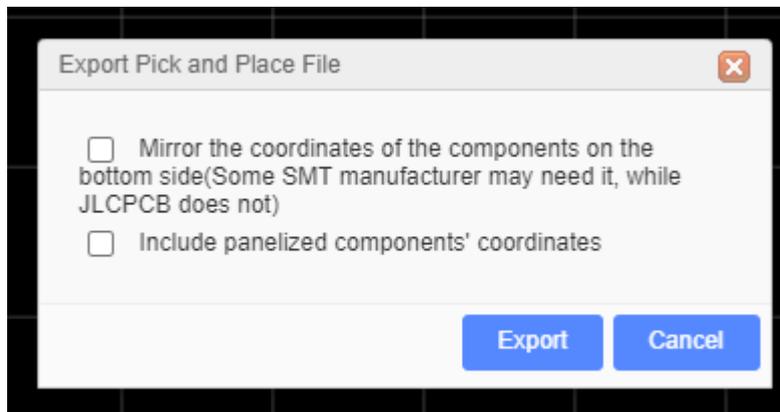
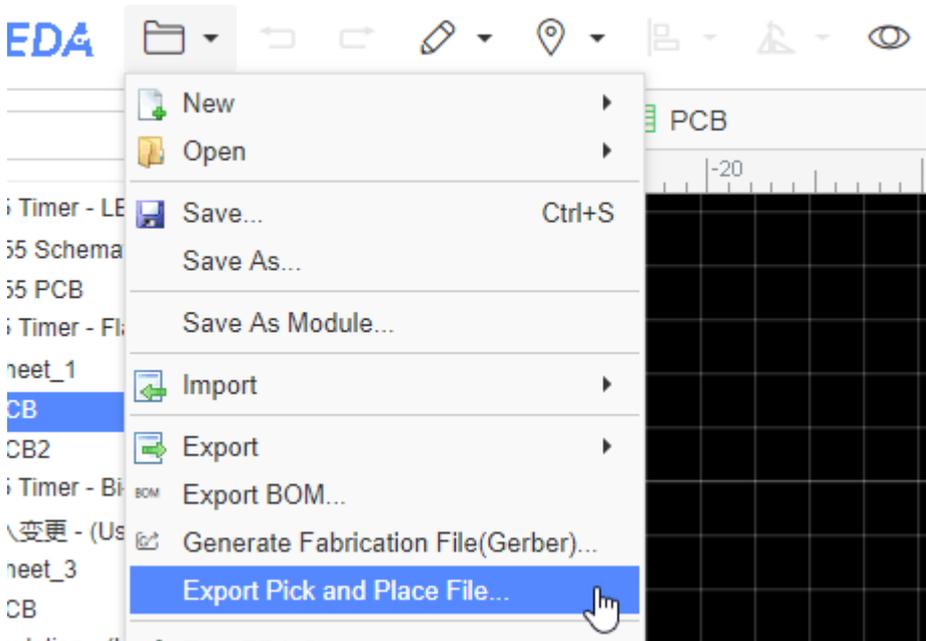
**Notice:**

- Before v6.4.17, 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.
- Since v6.4.17, the schematic BOM and PCB BOM are separated. If you assign the LCSC part at the PCB, it will not modify the schematic.
- 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.**



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	N
1	Designator	Footprint	Mid X	Mid Y	Ref X	Ref Y	Pad X	Pad Y	Layer	Rotation	Comment		
2	LED2	LED-3MM/2.5	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_150N	13.31mm	7.49mm	10.92mm	9.4mm	10.29mm	9.4mm	T	0	NE555DR		
5	LED1	LED-3MM/2.5	4.16mm	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.

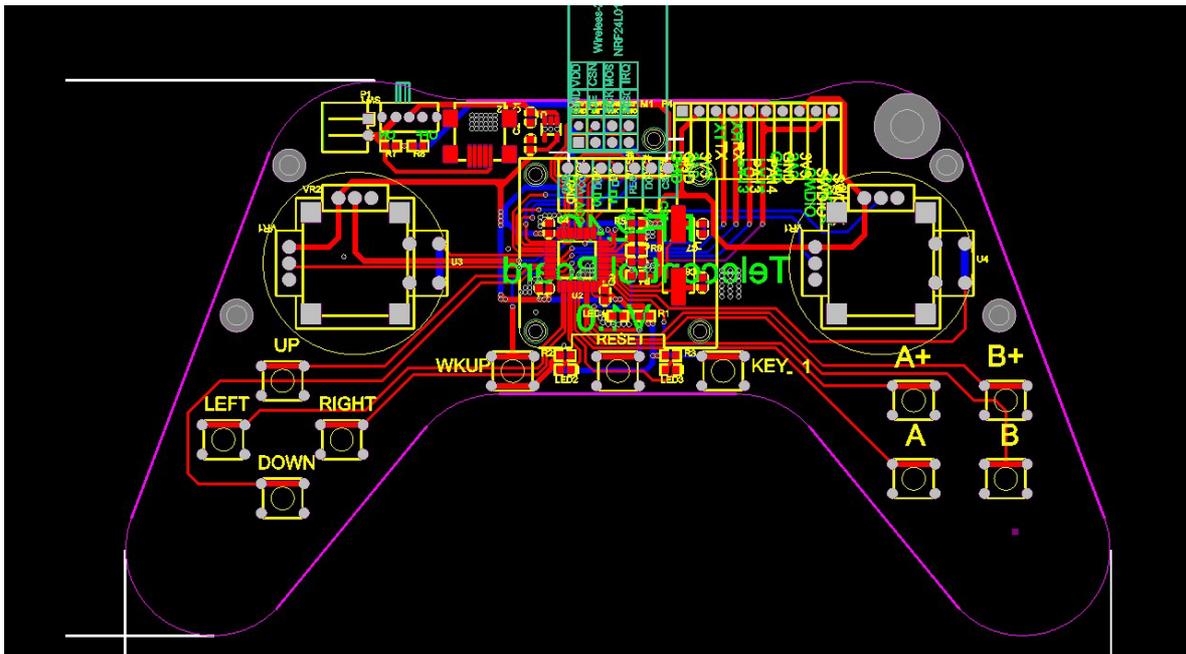
---

## Export DXF

---

EasyEDA support to export PCB to DXF.

At present EasyEDA supports to export a full layers and objects DXF file:



You can edit it at CAD tools very easy, toggle the layers as you want.

---

## 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](https://www.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 JLCPCB <https://jlcpcb.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.
  - [LCSC Assembled Libraries](#)
  - [Export BOM](#)
  - [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](#)

# Experiment no: 17

---

## Experiment Title: Create The Footprint in EasyEDA

---

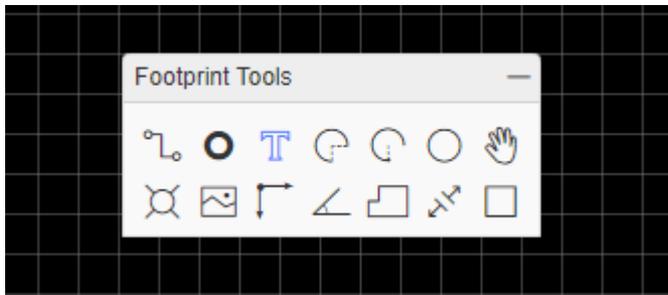
### Create The Footprint

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.

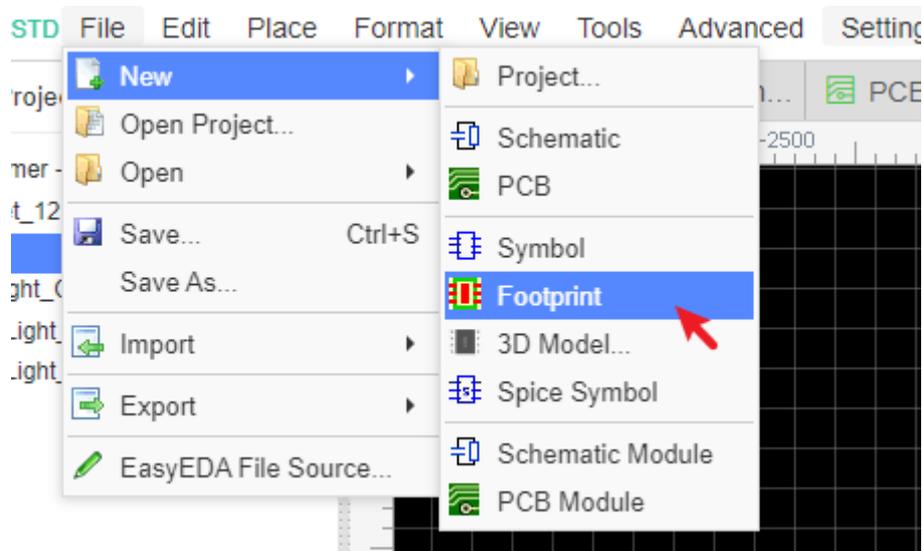
Footprint Tools almost are the same as PCB tools, just lacking some of the functions.



### Create Footprint

Start a new Footprint as shown below or by doing:

**File > New > Footprint**

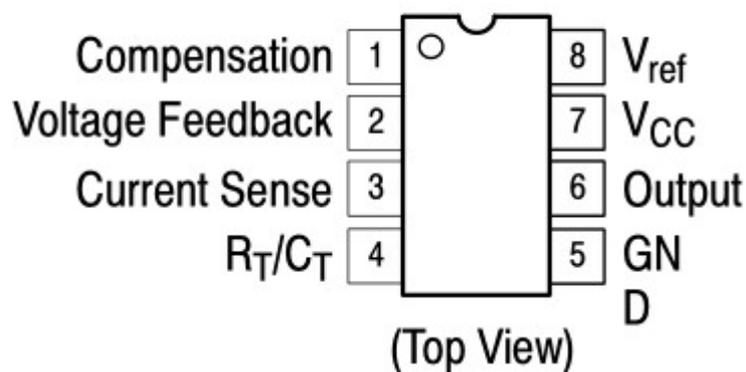


This opens the New Footprint editor.

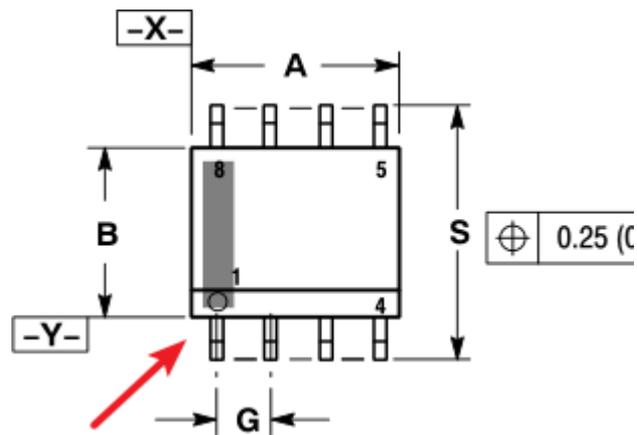
## Drawing Steps

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 be 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 based on your real usage situation.
  - Component's pin direction, page 1.

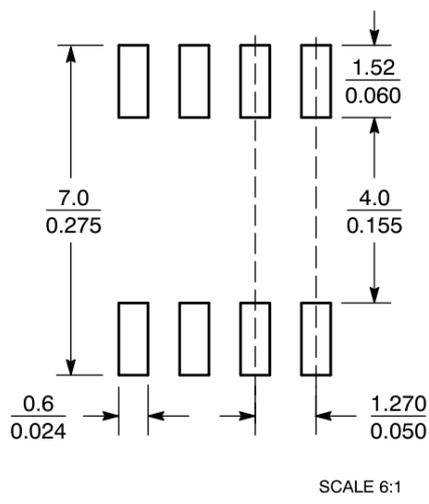
## PIN CONNECTIONS



- footprint polarity, page 1 and 18.

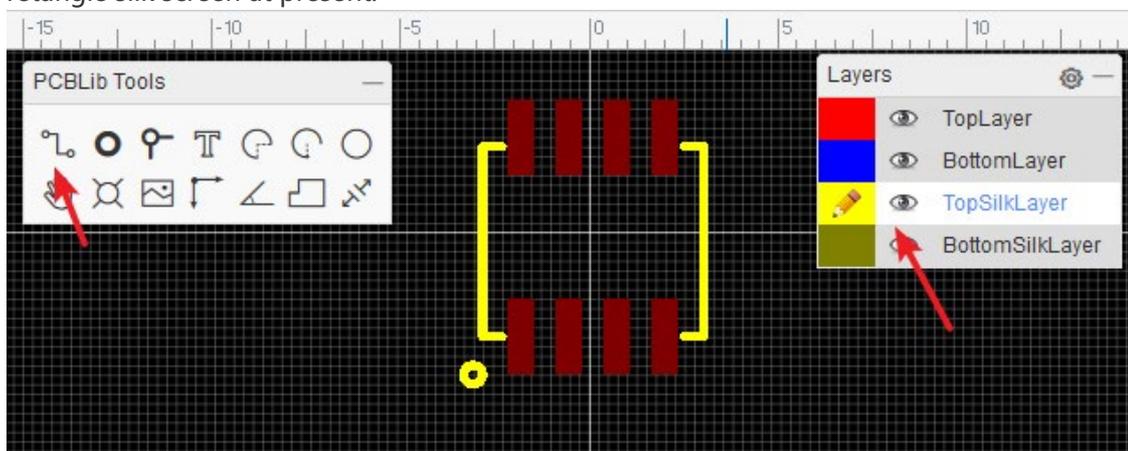


- Depends on page 18, placing one pad on the canvas on the top layer, and then change the pad number, size, shape type etc. And then set the coordinate for it, and place the less pads, you can use the "Top Menu - Align" tools to align the pads to fit the location. If you want to move the pad by mouse or direction key by small steps, you can set a new snap size at the right-hand panel.

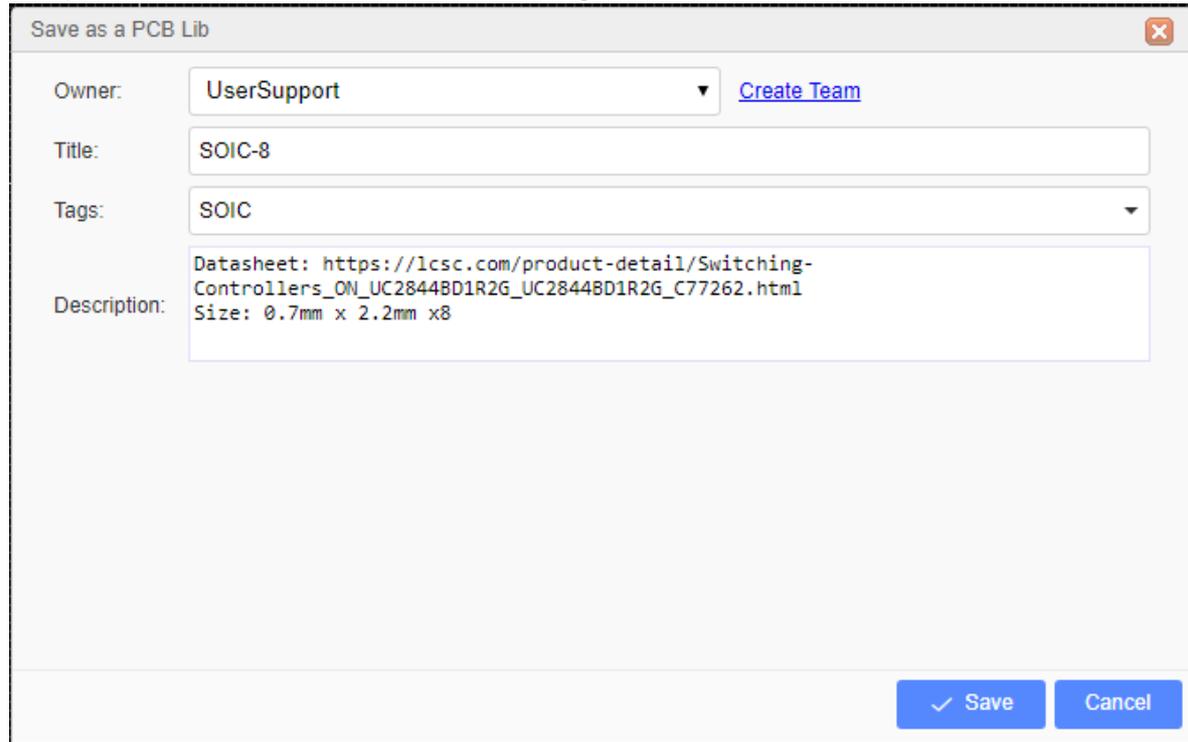


4. Drawing the Footprint silk screen. And sometimes you can add some marking and text on the mechanical or document layer.

- Switching layer to TopSilkLayer
- Using the Track and Arc to draw the silk screen. The editor doesn't support draw the rectangle 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.



Save as a PCB Lib

Owner:  [Create Team](#)

Title:

Tags:

Description:

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 .

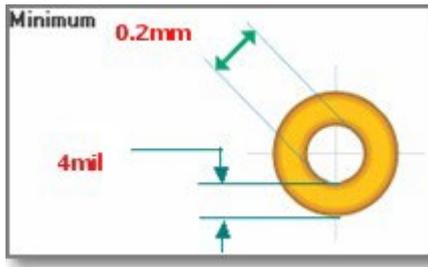
#### Notice :

- 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 rotation 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 not 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  $\geq 4\text{mil}$ . 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.

The screenshot shows a circular pad with the number '1' in the center on a black workspace. To the right, the 'Pad Properties' panel is open, displaying the following settings:

Pad Properties	
Layer	Multi-Layer
Number	1
Shape	Round
Width	1.524mm
Height	1.524mm
Rotation	0
Hole Shape	Round
Hole(D)	0.914mm
Plated	Yes
Center X	70.739mm
Center Y	36.068mm
Solder Mask ...	0.102mm
ID	gge5
Locked	No

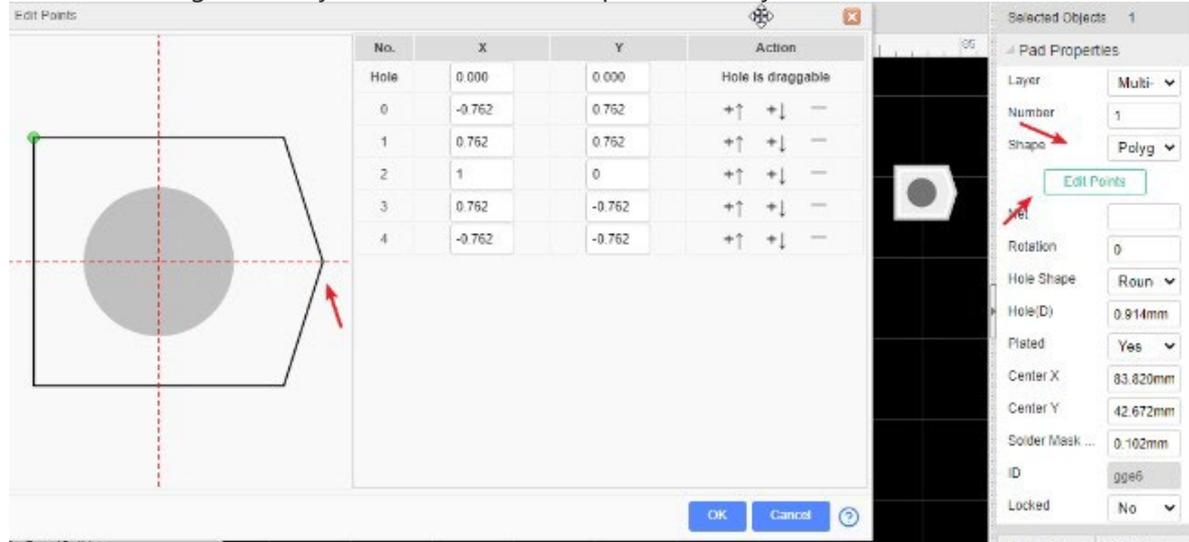
**Number:** Remembering the pin numbers you set in the schematic symbol in your Schematic symbol: to connect those schematic symbol pins to the pads in your PCB footprint, the pad numbers you set here in the Footprint footprint must be the same.

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

EasyEDA supports four shapes: **Round** , **Rectangular** , **OVAl** and **POLYGON**.

- **OVAl** PAD will give your 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.

---

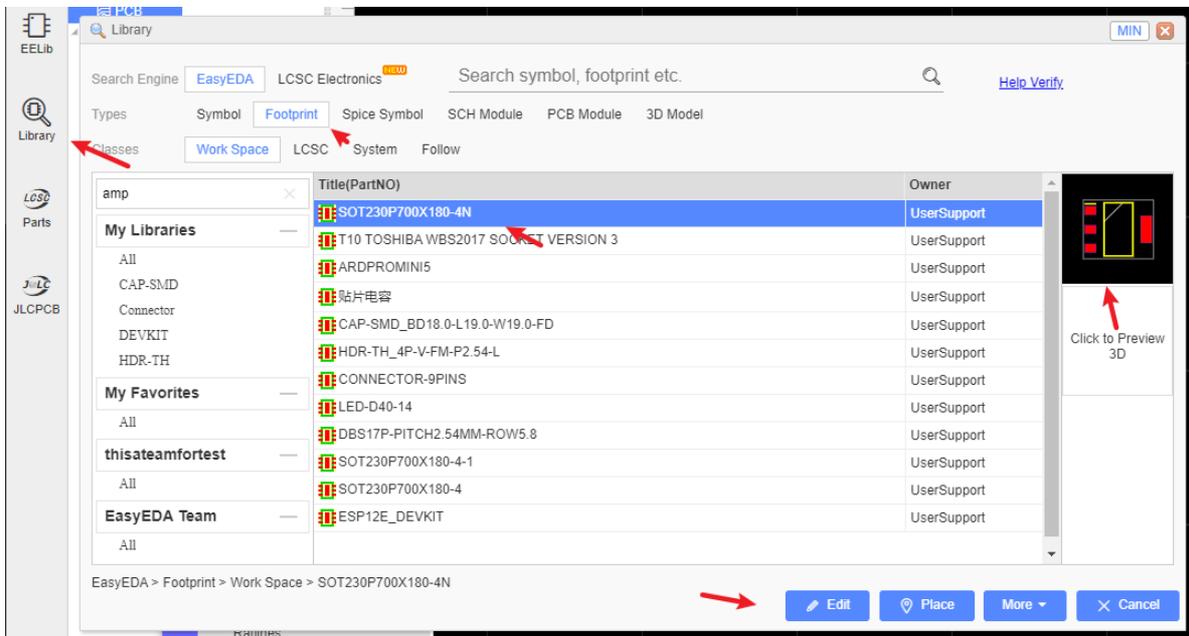
## Edit Footprints

---

### Edit Footprint in Library

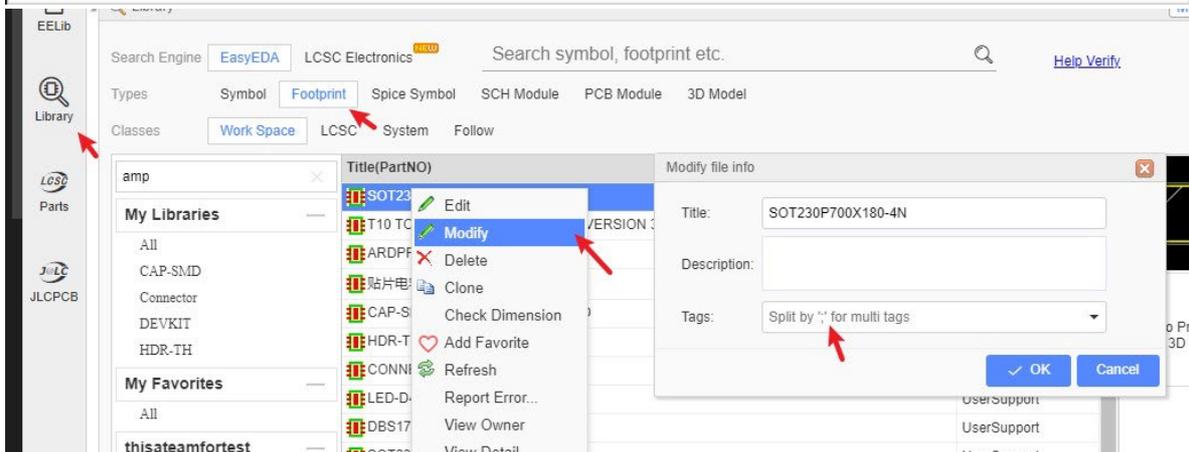
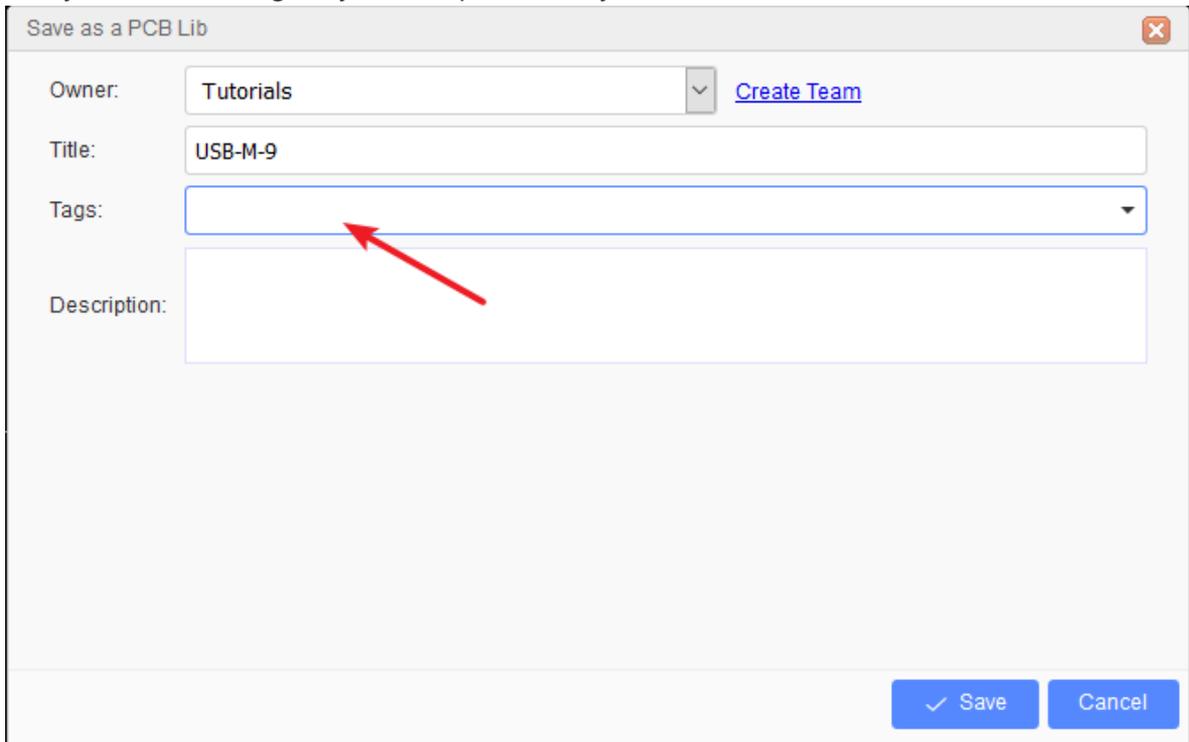
When you found a Footprints(footprint) but it can not be satisfied for your design, you can edit it to be your personal PCB footprint.

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.

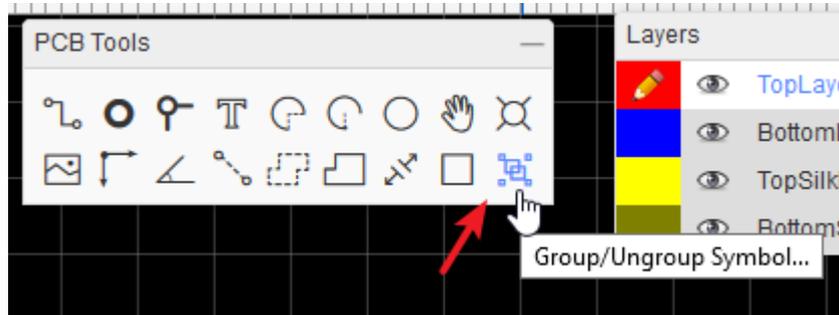
And you can add a tag for your Footprint when you save it:



## 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.

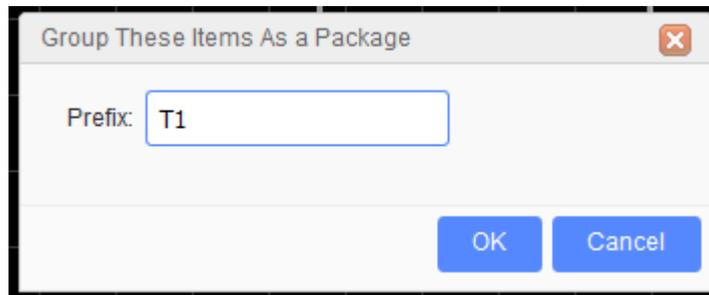
On the **PCB Tools** palette there is the **Group/Ungroup Symbol...** button.



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.

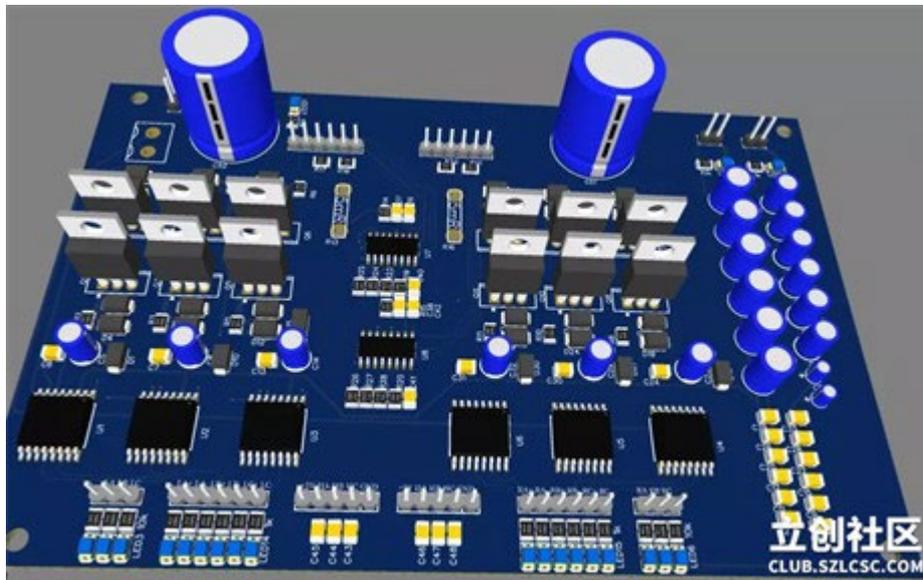
---

## 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 "mtl" file and then imported, and the "mtl" 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> ("mtl" files are automatically downloaded when obj files are downloaded.)

<https://github.com/KiCad/kicad-packages3D>

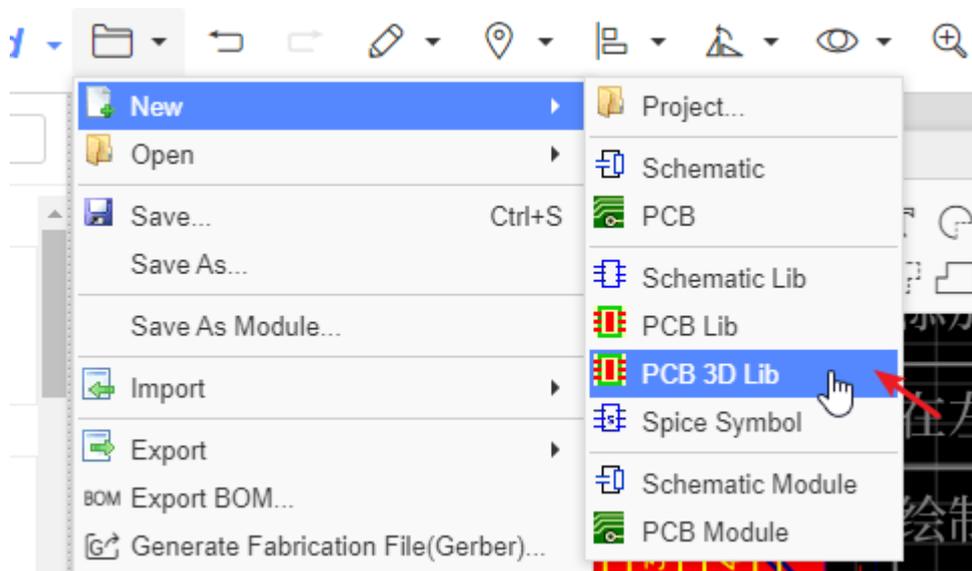
<https://www.traceparts.com/zh>

<https://www.3dcontentcentral.com/>

<https://grabcad.com/>

## 2. Create a new 3D library

in "Top Menu - New - PCB 3D Lib".



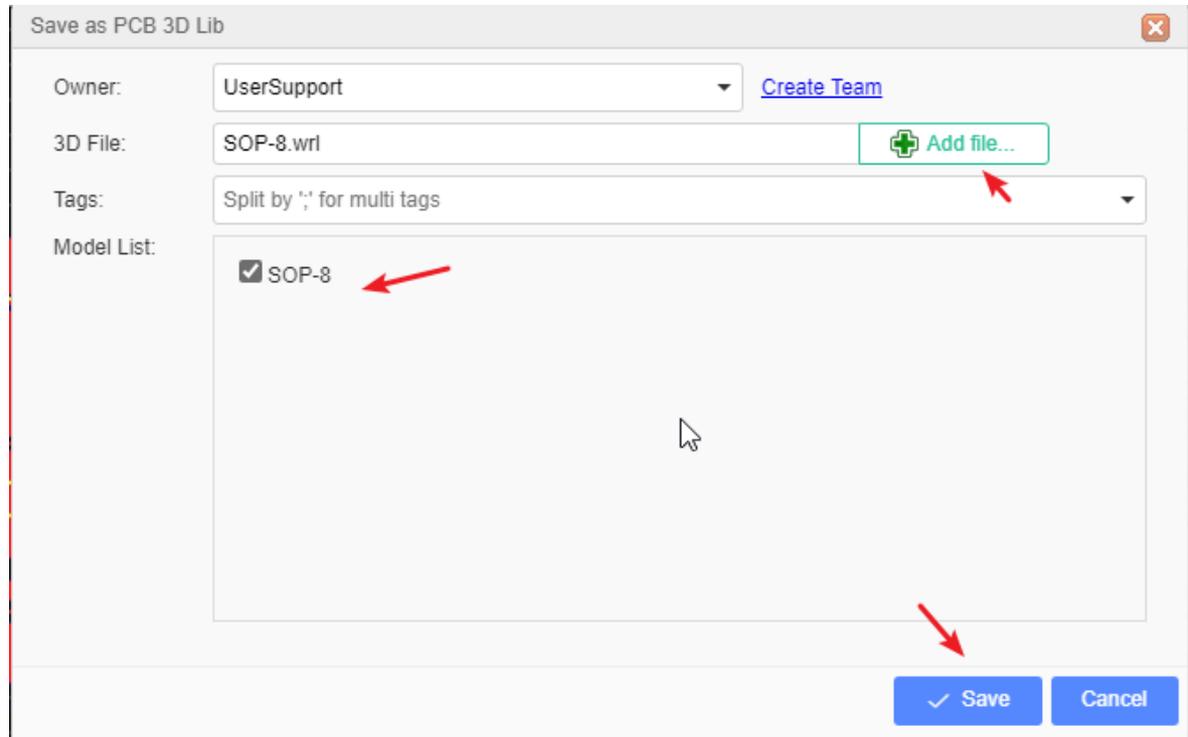
If you have many 3D libraries, you can zip them to a zip file to import, no more than 10 WRL files for one zip file, otherwise it will fail to import.

OBJ format contains many 3D models in one file, you don't need to zip them.

If your WRL file cannot import directly, please use FreeCAD to export a new one and try again.

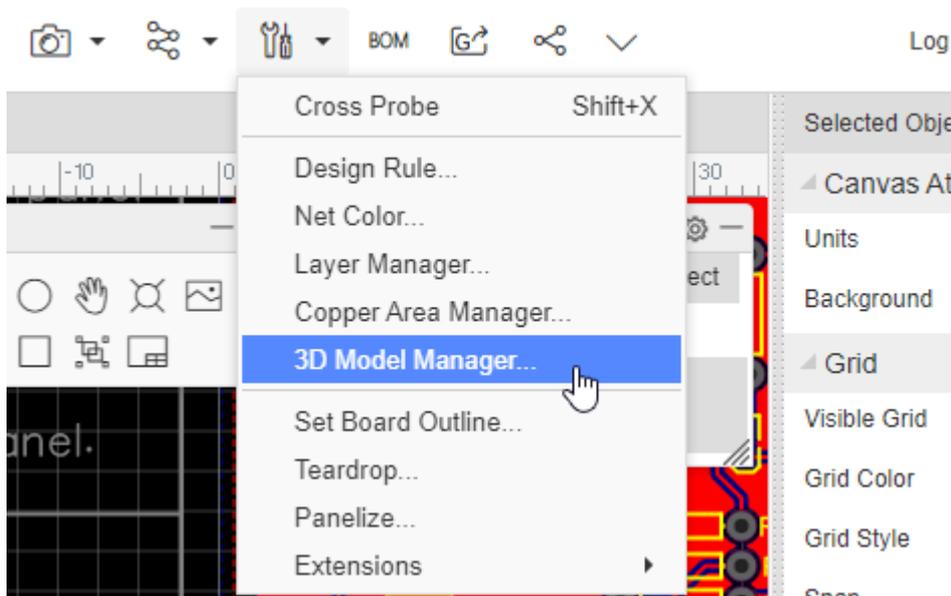
### 3. Import 3D model.

You can check which 3D model you want to import.



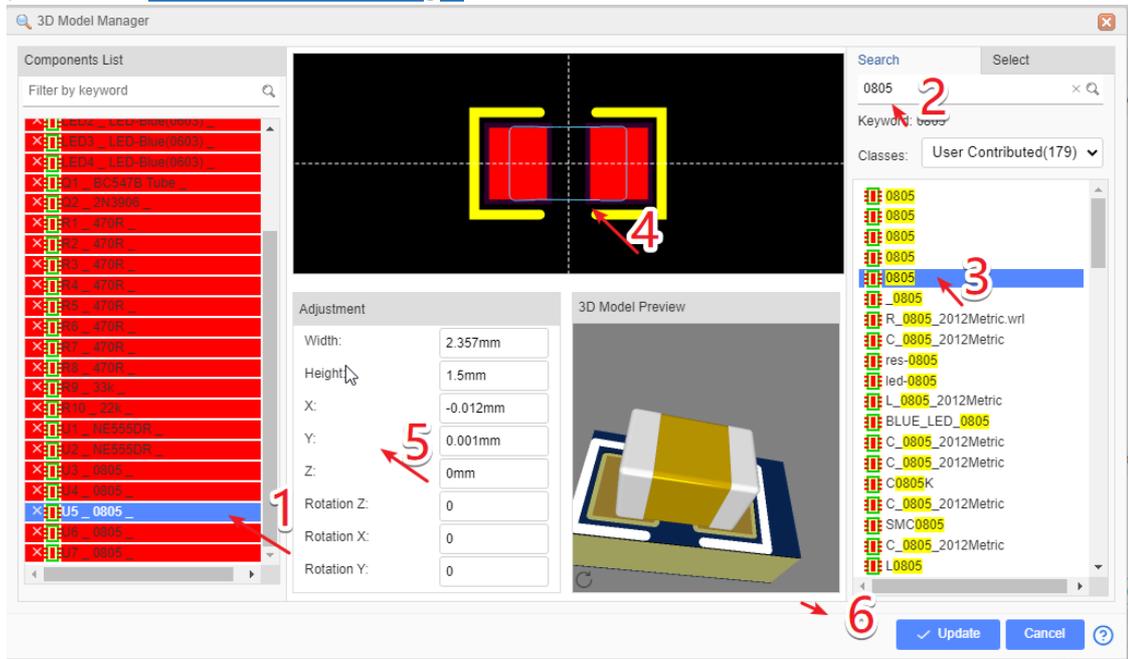
### 4. Specify the 3D model

- 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,

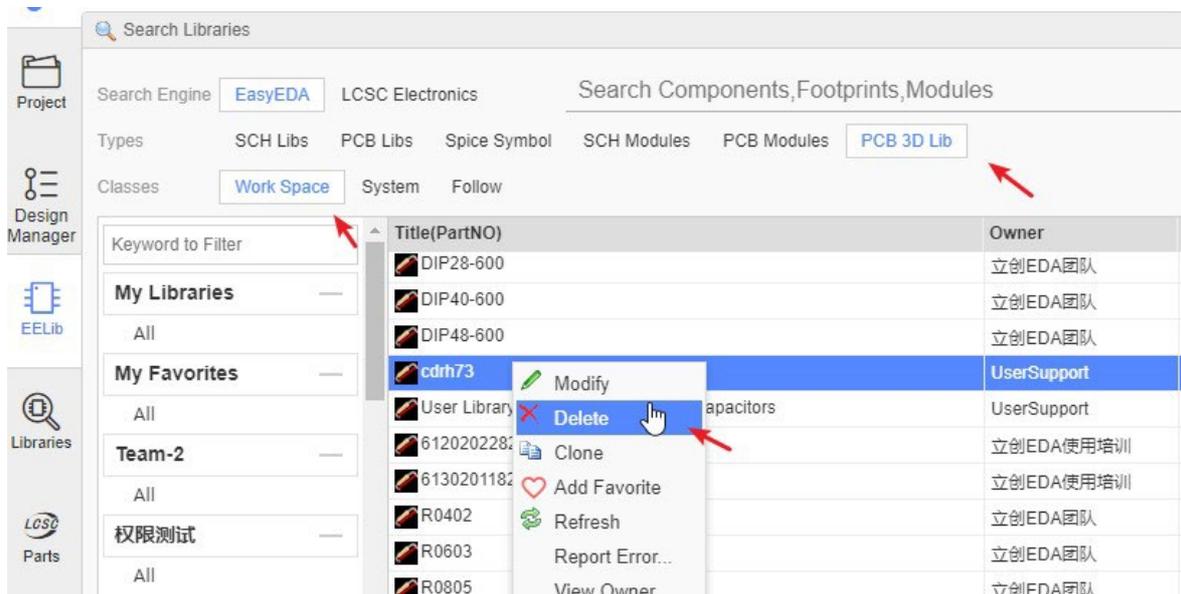
please see: [PCB - 3D Model Manager](#)



- 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



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".

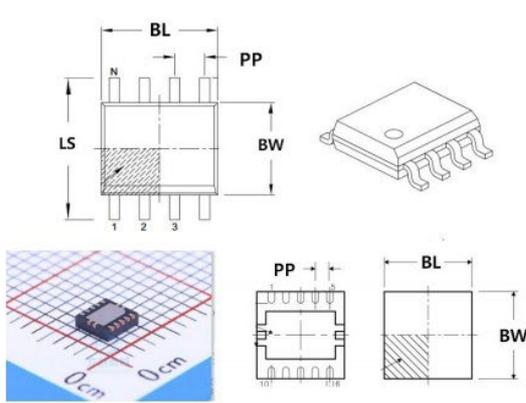
Each company should have its own footprint naming specification, EasyEDA is no exception, EasyEDA has more than 180,000 of official library (LCSC library), multiple engineers in the construction of footprint, more need unified library rules and footprint naming rules to ensure library consistency and 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.

书签

- 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
    - 3.2.2 Integrated Circuit



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. VSOP, 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:

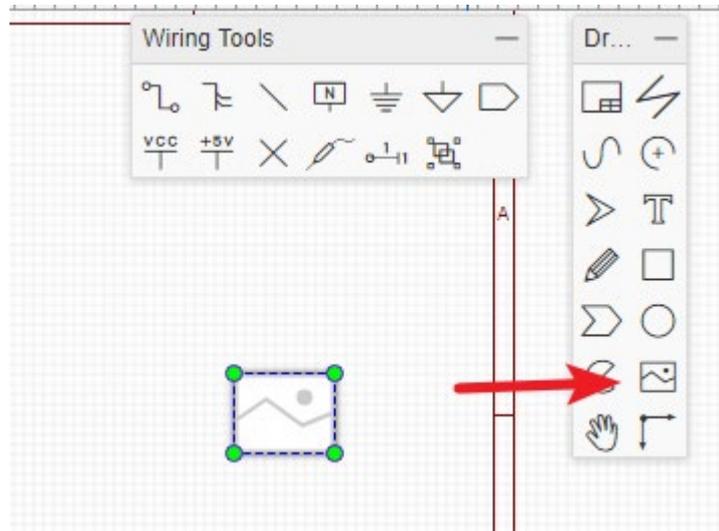
Download: [EasyEDA Footprint Naming Rule Reference.pdf](#)

# Experiment no: 18

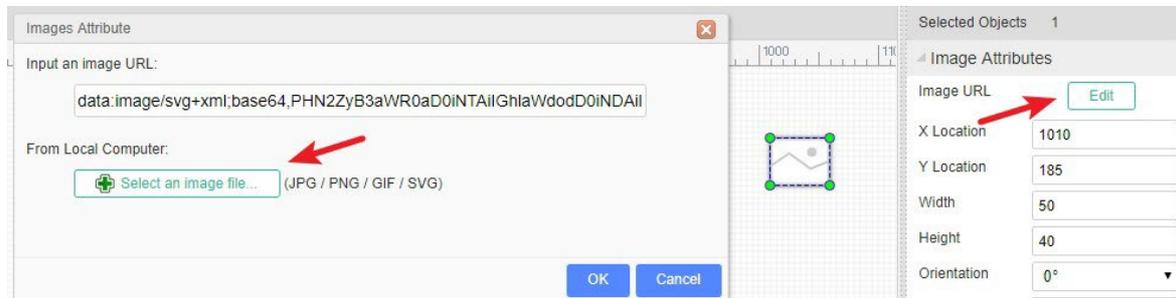
## Import Image

### Import Image to Schematic

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



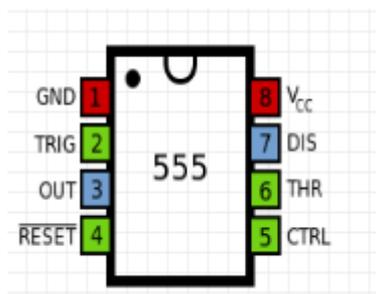
Select the place holder, so you can see the image's attributes in the right hand Properties panel:



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)

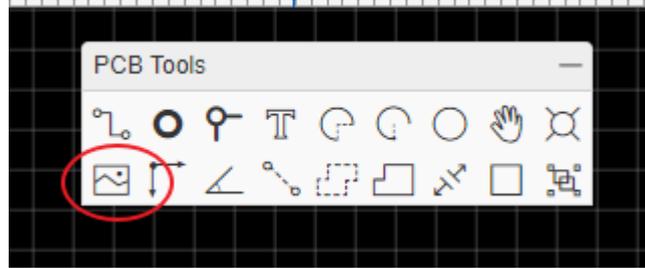
will make your image look like this:



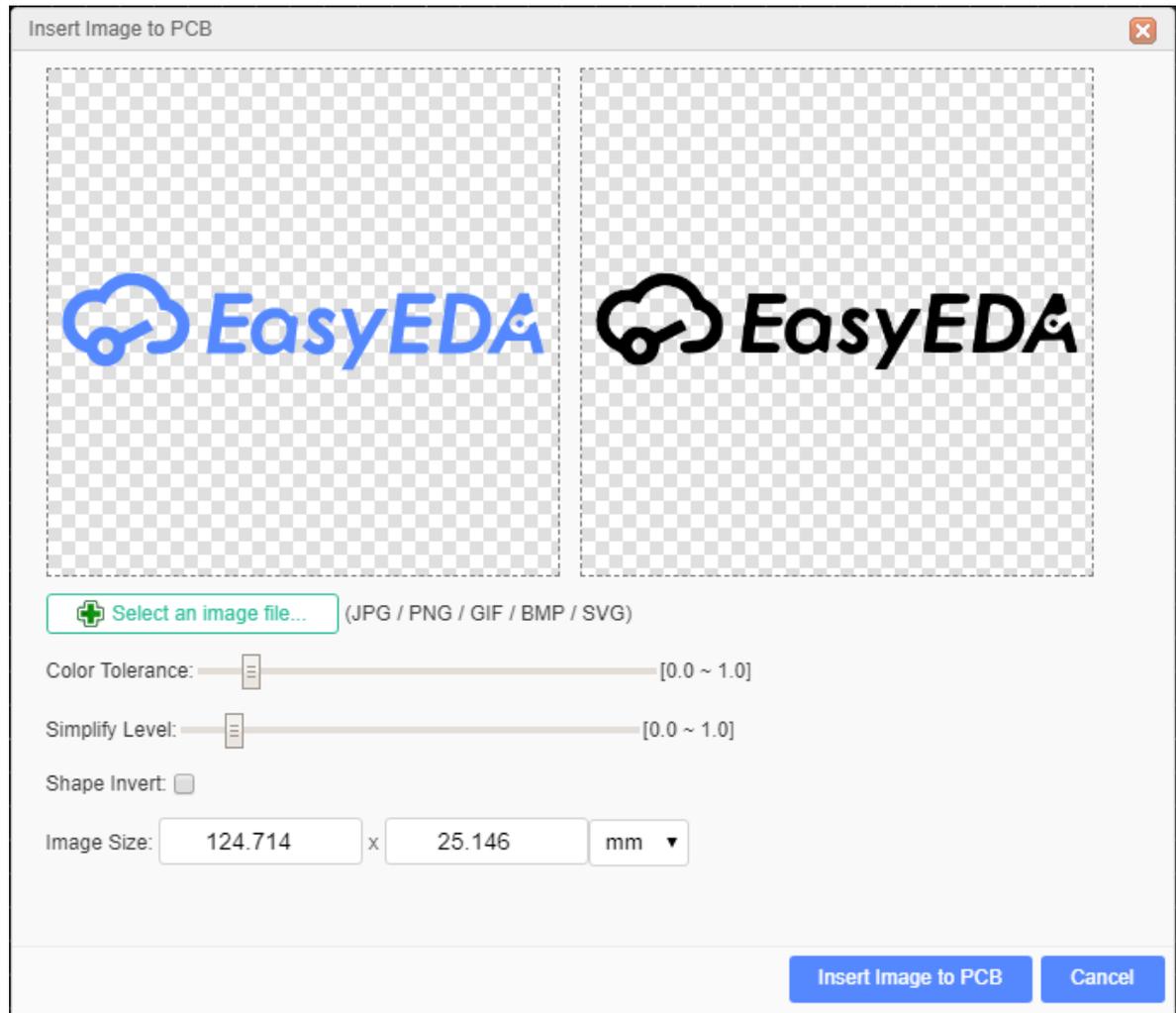
Please note: at present, EasyEDA does not host images, so you need to upload your images to an image sharing site such as <http://www.imgur.com>.

## Import Image to PCB

In the 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 shown below.

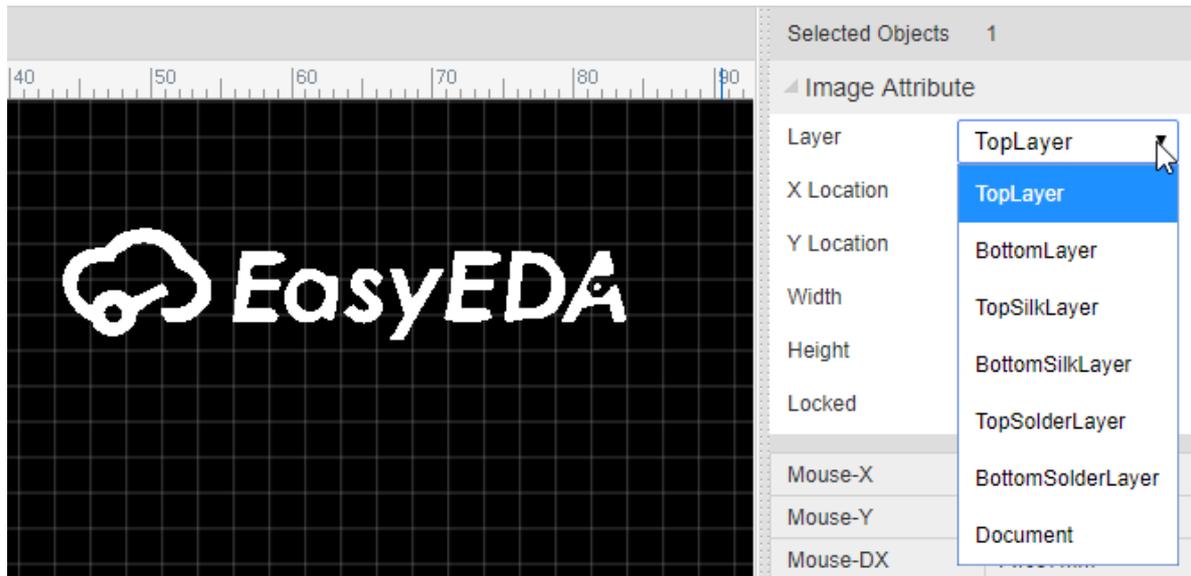


SVG. Unlike some other EDA tools which only support a monochrome bitmap image, EasyEDA supports full color, but Monochrome Bitmap is welcome.

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

You can also select shape invert.

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

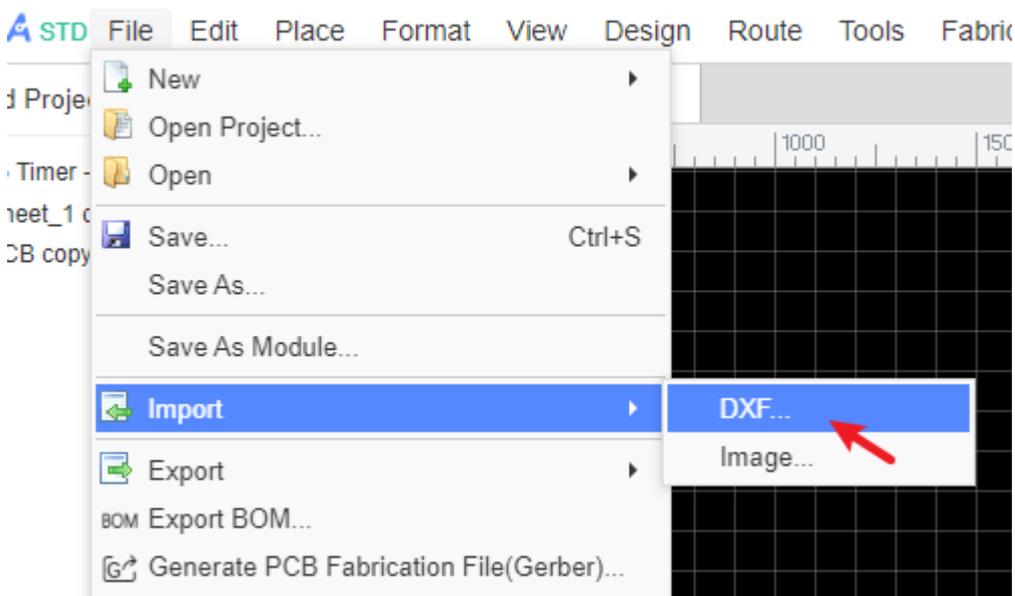


## Import DXF File

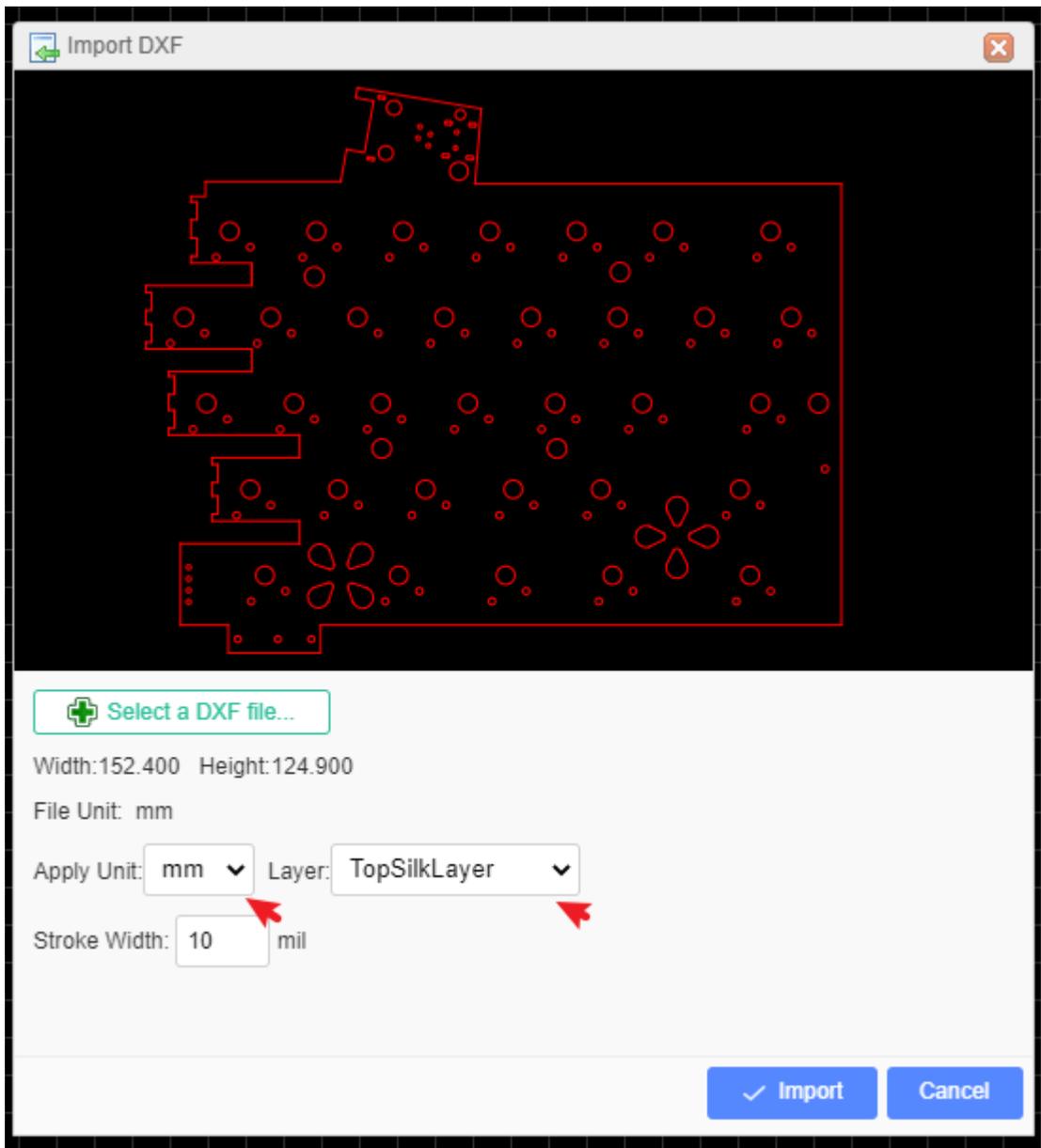
How can you create irregular board outlines or complex board outline in EasyEDA? This is sometimes needed when you are designing a PCB for an enclosure that may have a curved profile, or other unavoidable mechanical features that must be accounted for in the design.

EasyEDA supports importing DXF into PCB.

Find the import DXF menu under the file menu. Use: File - Import - DXF



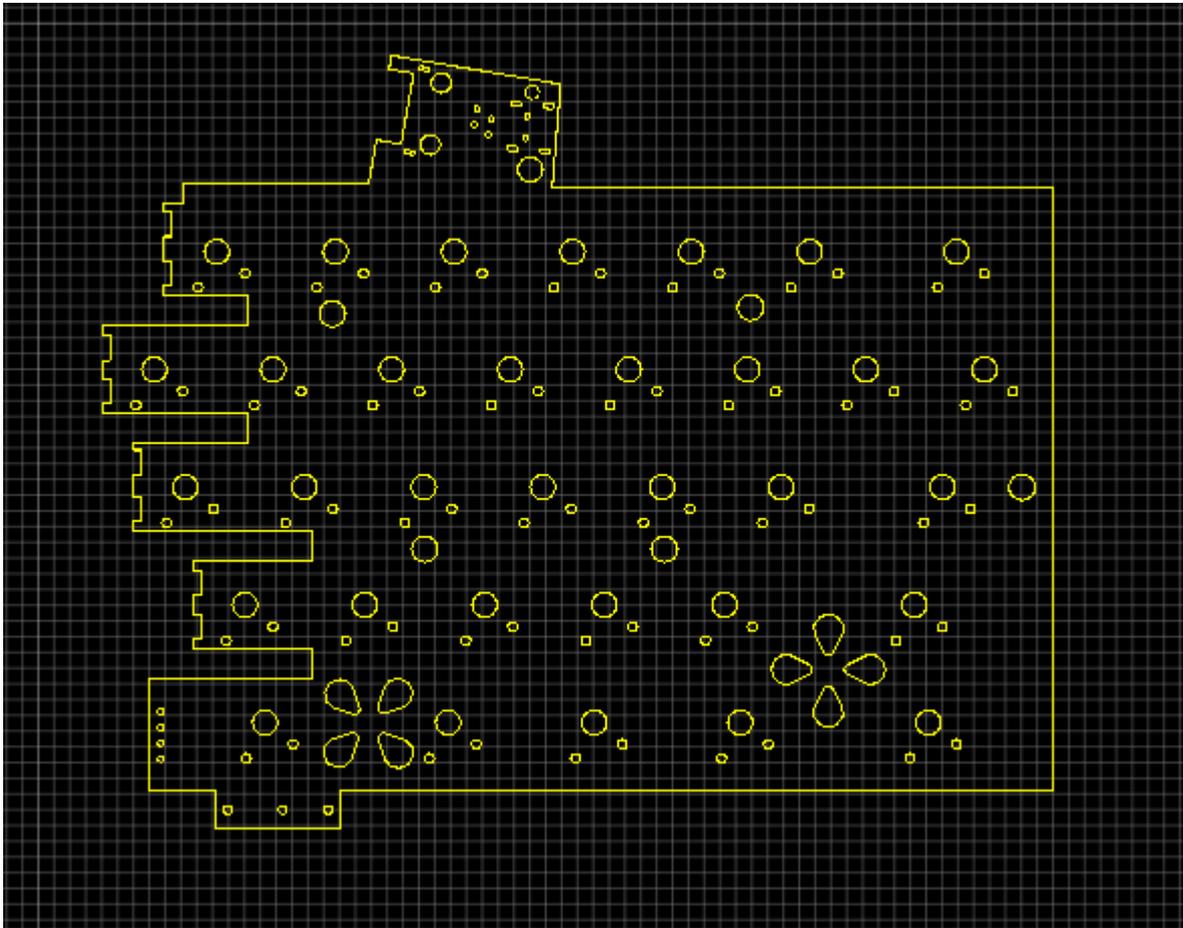
After selecting the \*.DXF file, you will find a dialog as shown below



EasyEDA provides some options, units(mm, cm, mil, inch), and selecting the layer to which the shapes will be applied.

When importing DXF into schematic or symbols, the units are pixels.

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 a large DXF to copper layer directly, it will be very slow and appear to be hung in the editor.

---

## Import Altium Designer

---

**The import function is in beta now, please check your design carefully after importing.**

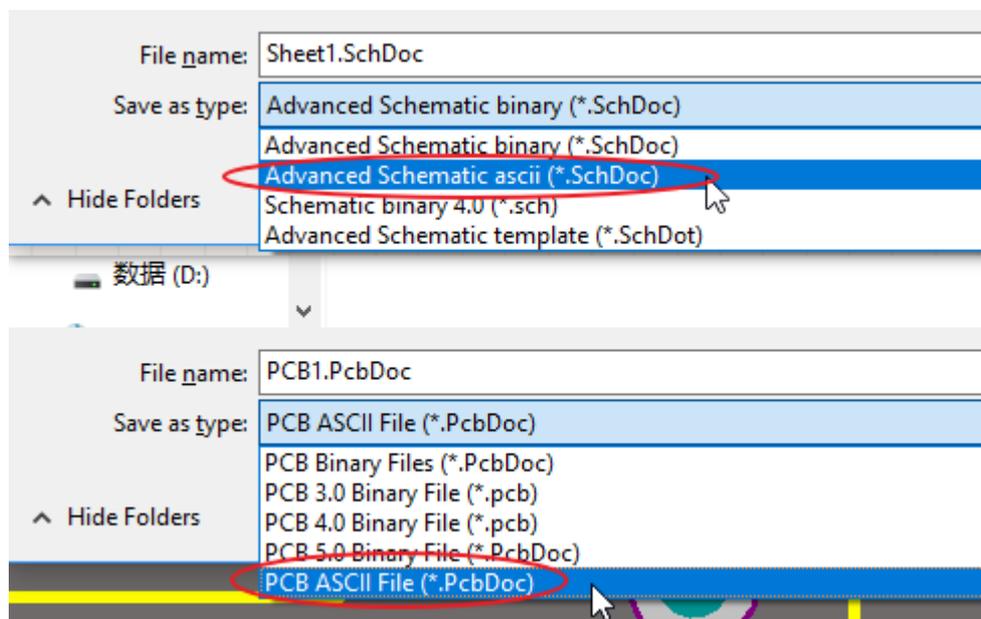
Some design rules are not yet supported by EasyEDA.

### Import Schematic and PCB

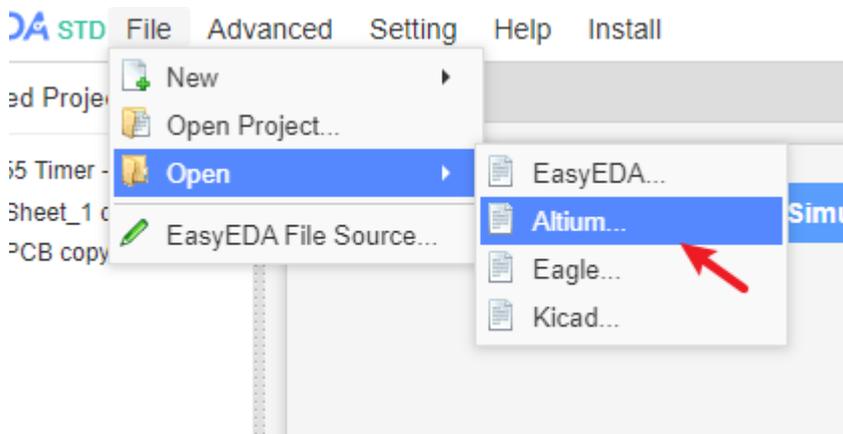
1. You can import Altium Designer's Schematic and PCB files into EasyEDA but the format must be **ASCII** files, so you need to save the designs as Ascii files like this.

Schematic saved as "Advanced Schematic ascii (\*.SchDoc)"

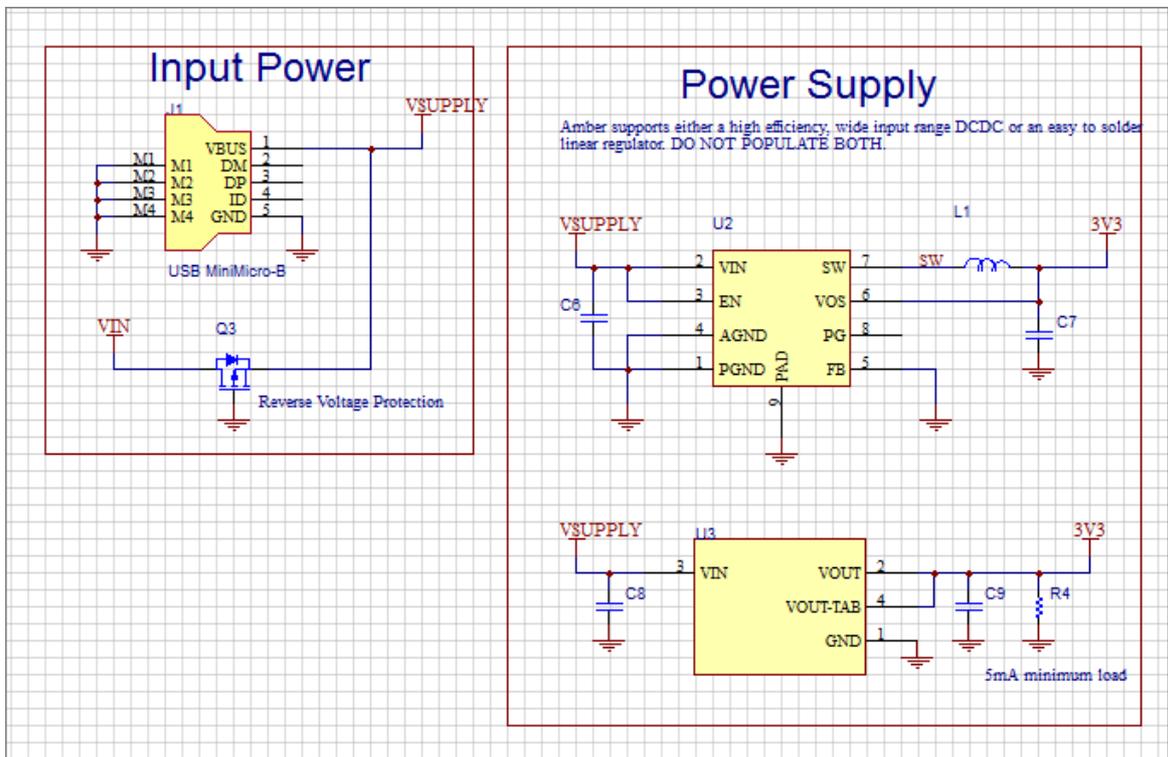
PCB save as "PCB ASCII File (\*.PcbDoc)"



Then import it using: 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:

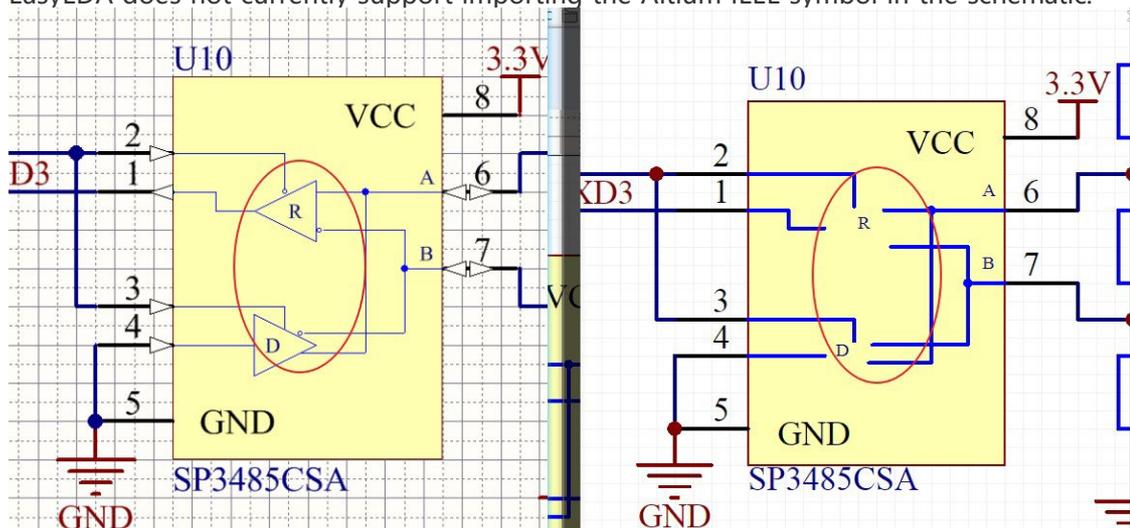


If your schematic and PCB are Protel 99se format files, please open in Altium Designer and save as ASCII format, and then import them. EasyEDA does not support Protel 99se file format directly.

2. If you import Altium schematic and some text becomes unreadable, please encode your ASCII file with UTF-8.

**Notice:**

- EasyEDA does not currently support importing Altium PCB rules.
- EasyEDA does not currently support importing Altium PCB inner layers, please modify manually after imported.
- EasyEDA does not currently support importing the Altium IEEE symbol in the schematic.



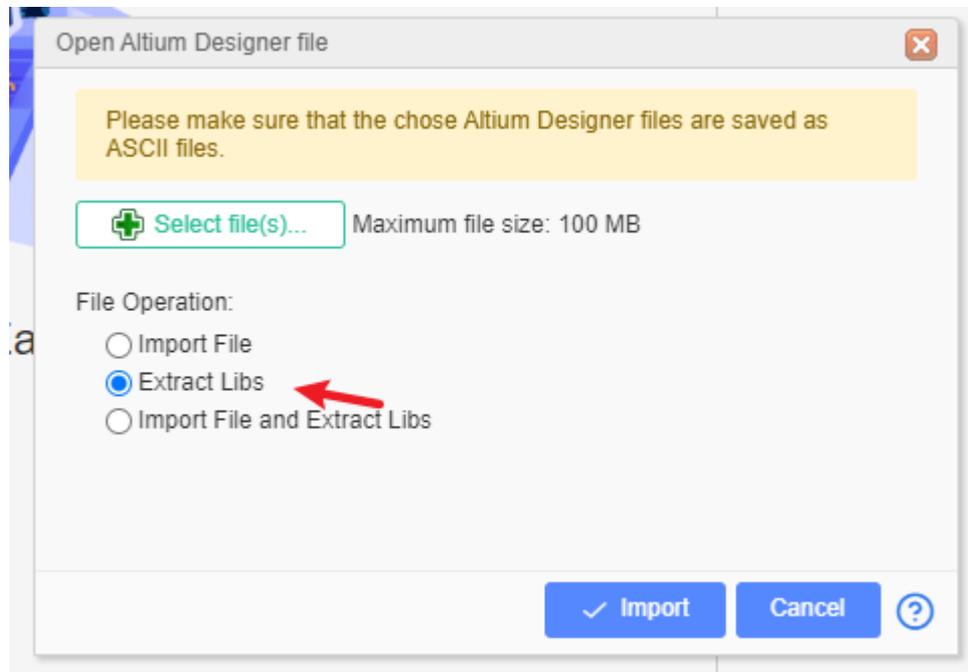
- some details to be lost!!!
- If the Altium file is large it may take a long time to import, removing the copper layer before importing can improve the speed.
- Importing the border processing logic of AD file: The Board Shape turns into a border layer, keep-out turns into a document layer, and Mechanical becomes a mechanical layer.

## 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 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

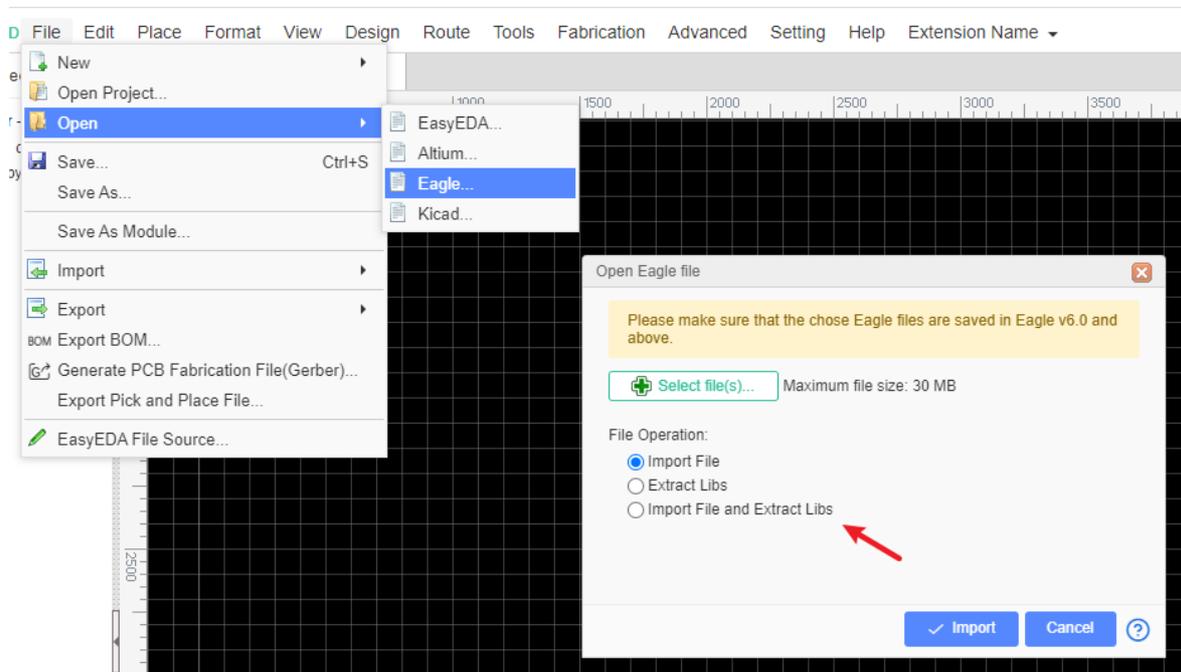
Please refer next section

---

## Import Eagle

---

Eagle Schematic/PCB/libs can be imported, but EasyEDA only supports 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 opened in Eagle, but fails import to EasyEDA, save a copy with the latest Eagle, and then import that one.

If your file has been saved as a copy from v6.0 or greater, but importing still fails, then edit the Eagle file with a Text Editor. Search the file for garbled characters (something that looks strange), remove them, and try again.

If your schematic needs to update the PCB, please use the "Import File and Extract Libs" option, make sure that all libraries are imported first.

Some rules or primitives are not supported, please check carefully after importing to make sure the design is correct.

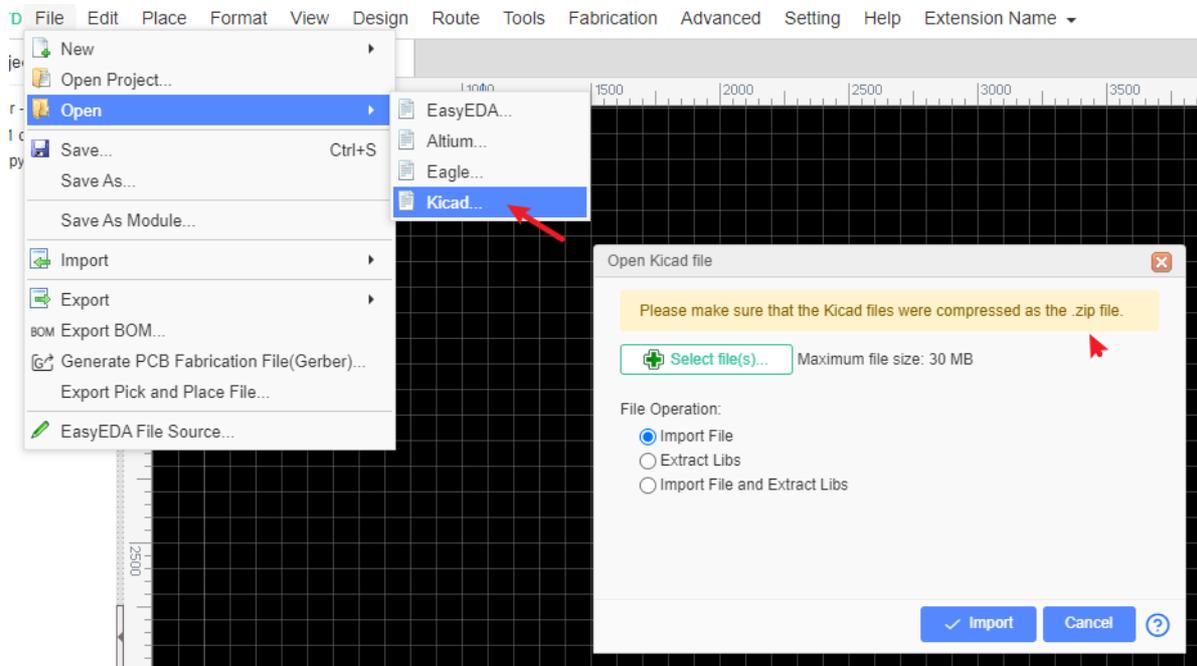
---

## Import KiCAD

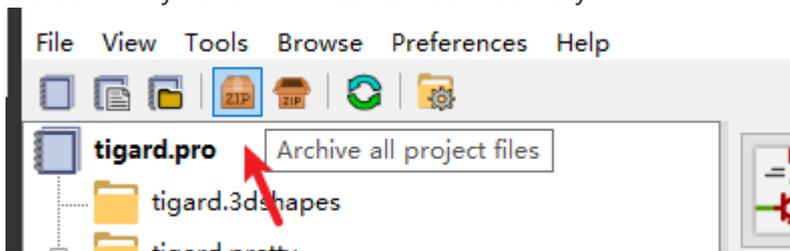
---

EasyEDA supports importing KiCAD v4.06 and greater version KiCAD files, if the KiCAD files version is less than v4.06, please open them with 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, we suggest using the KiCAD archive tool when opening the project in KiCAD, it will include the symbols in the ZIP file automatically.



### Notice

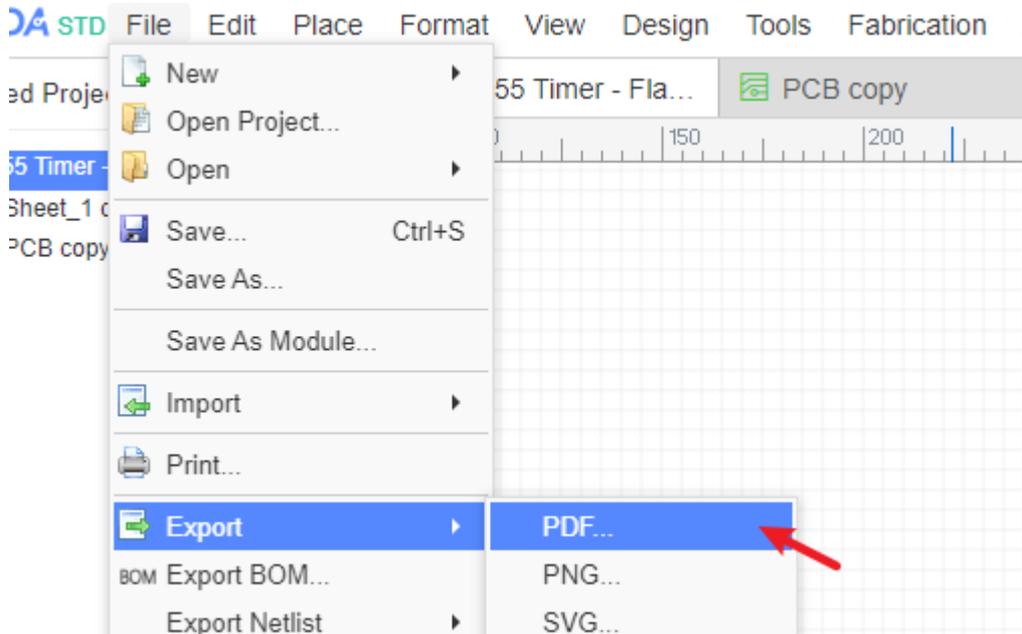
- For the KiCAD special symbols such as Power symbol (Power Flag(PWR\_FLAG)), EasyEDA will convert them as the symbol not Netflag, you can delete them if you don't need them.
- The PCB design rule is not supported yet.
- KiCAD has updated the document format since KiCad v5.1.3, if the import fails, please try a previous version. A fix for this is being developed.

# Experiment no: 19

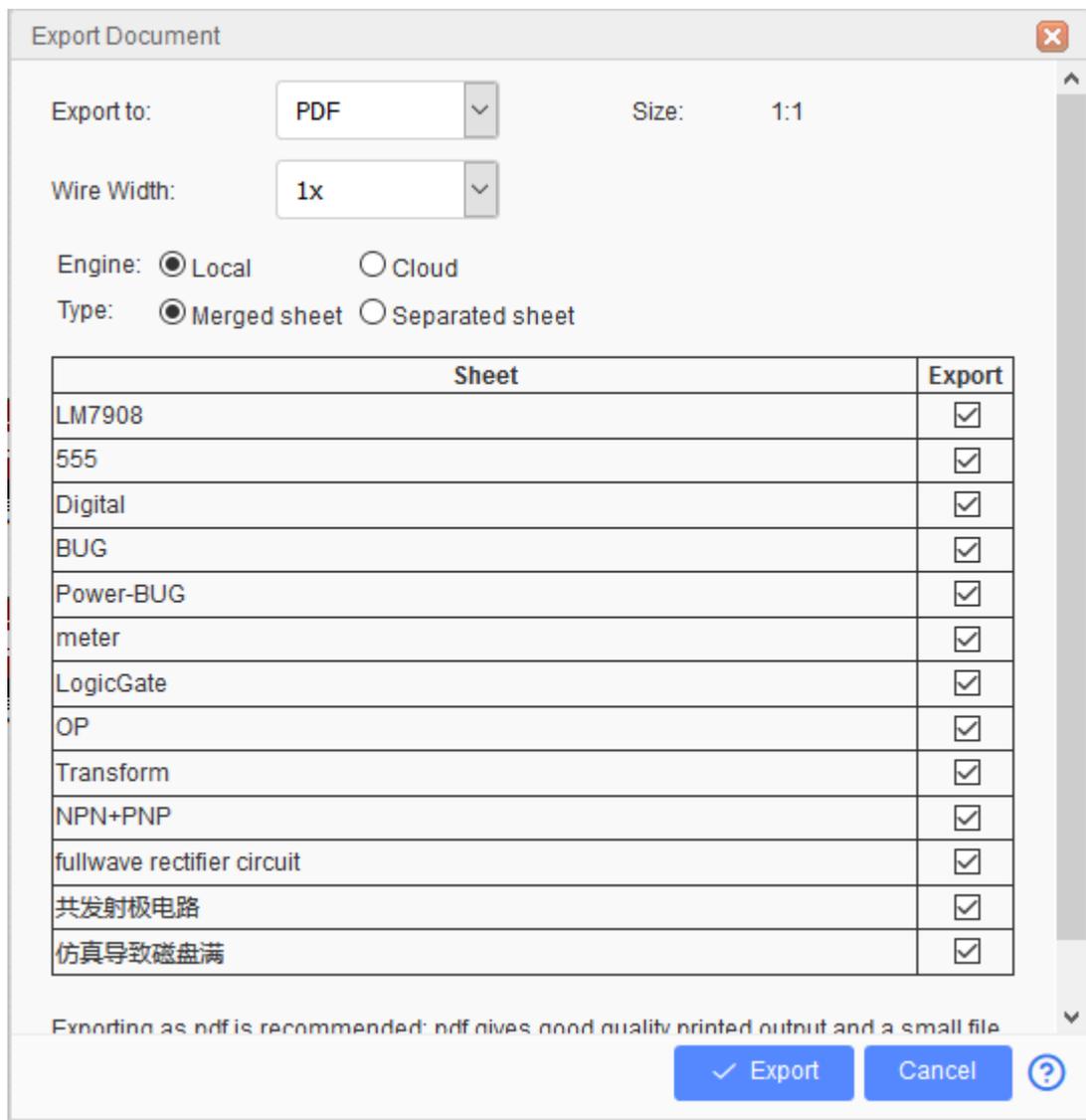
## Experiment Title: Export Schematics

### Export Schematics in PDF/PNG/SVG

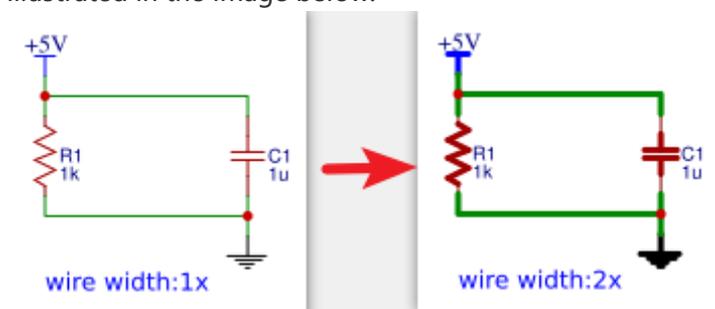
Use: \*\* > File > Export > PDF/PNG/SVG...\*\*



will open this dialog:



- **Export to:** This allows you to export your design to SVG, PNG and PDF file format.
- **Size:** Only for PNG and SVG. This is the width of the image, for example, when you set size as 1x, the exported PNG's width is 600 pixels. If you set the size as 2x the width will be 1200 pixels in the exported PNG.
- **Wire Width:** If you set this to 2x, the line width will be enlarged two times. This is illustrated in the image below.



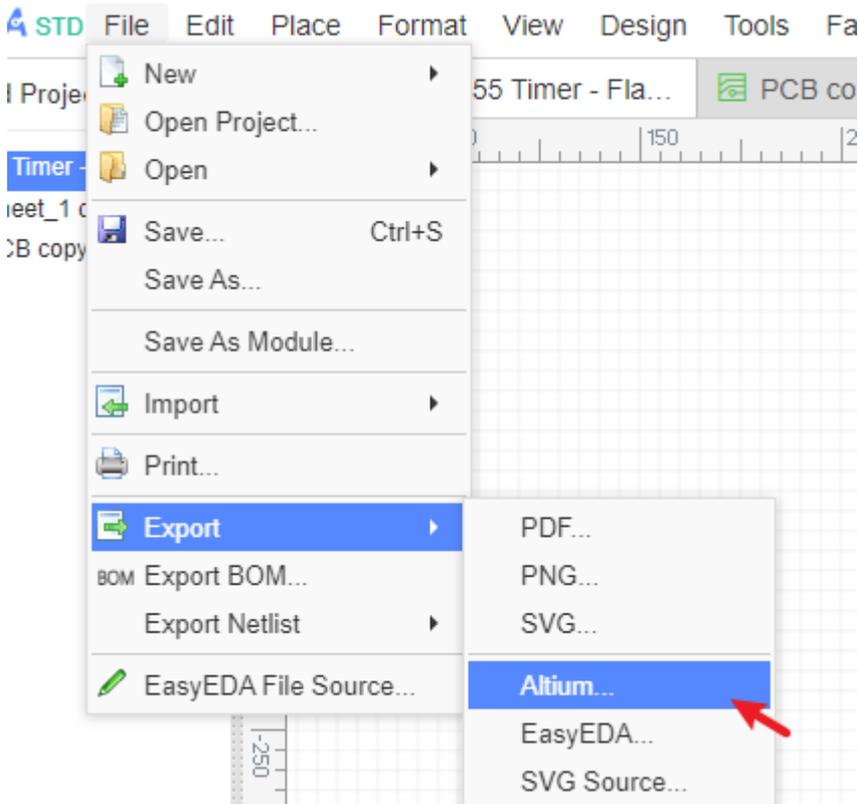
- **Engine:**
  - **Local:** PDF generated by Editor
  - **Cloud:** PDF generated by the Cloud Server, this feature will likely soon be removed from EasyEDA.
- **Type:**
  - **Merged sheet:** If your schematic has multiple sheets it will be merged into one file.
  - **Separated sheet:** If your schematic has multiple sheets each sheet will be created in a separate file.

# Export Schematics in Altium Designer Format

file will be

EasyEDA supports exporting the schematics in Altium Designer format.

Use "**File > Export > Altium...**", and click the "**Download now**". A `.schdoc` file will be created.



The more information please refer to [Export Altium](#)

## Download Schematics

Please refer to [Export EasyEDA Source](#)

## Print Schematic

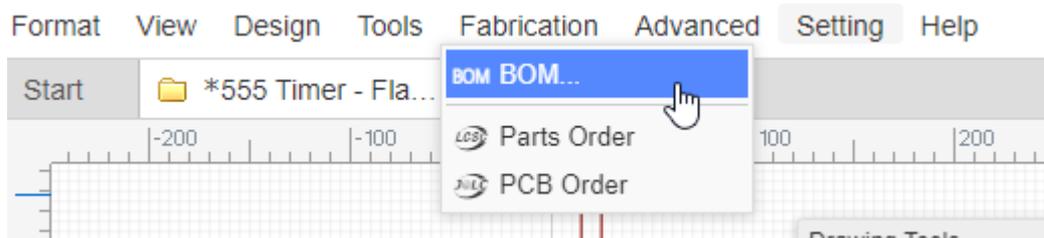
The command **File > Print**, will open a web page with the print dialog which is used to print the schematic.

If you are using EasyEDA Client you need to take note of the edge of the sheet, if it is past the page edge it is recommended to export to PDF and print.

If you are using Chrome, you will need to adjust the print options to fit to the page size and disable the option for "Headers and footers". Without this the A4 size will print an extra page.

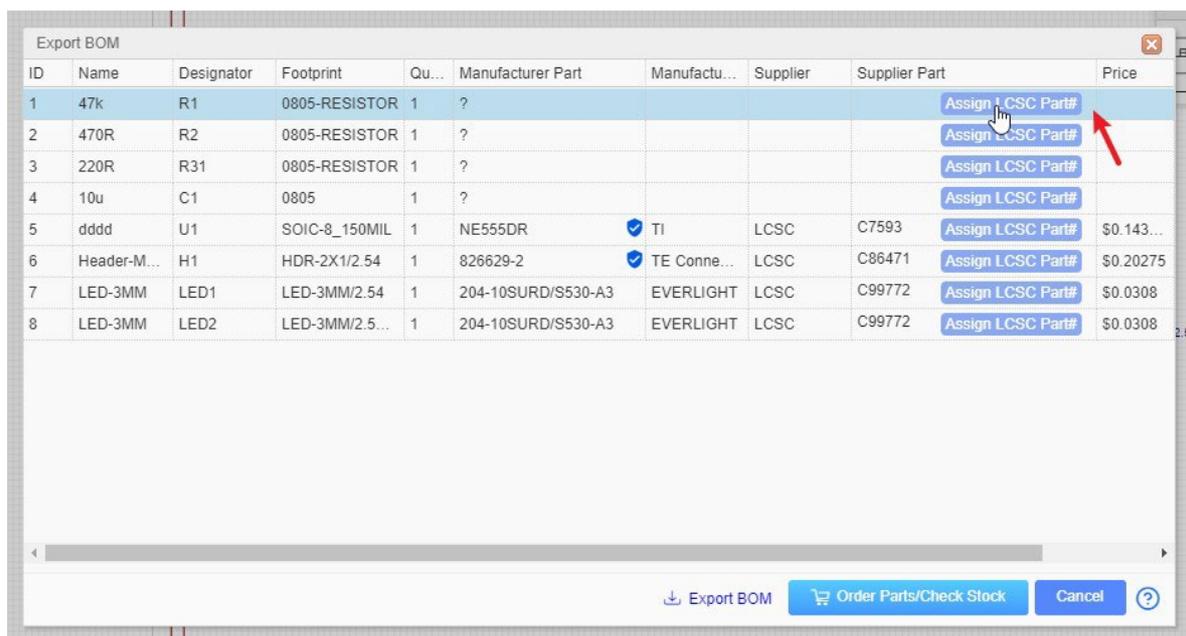
## Export BOM

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".

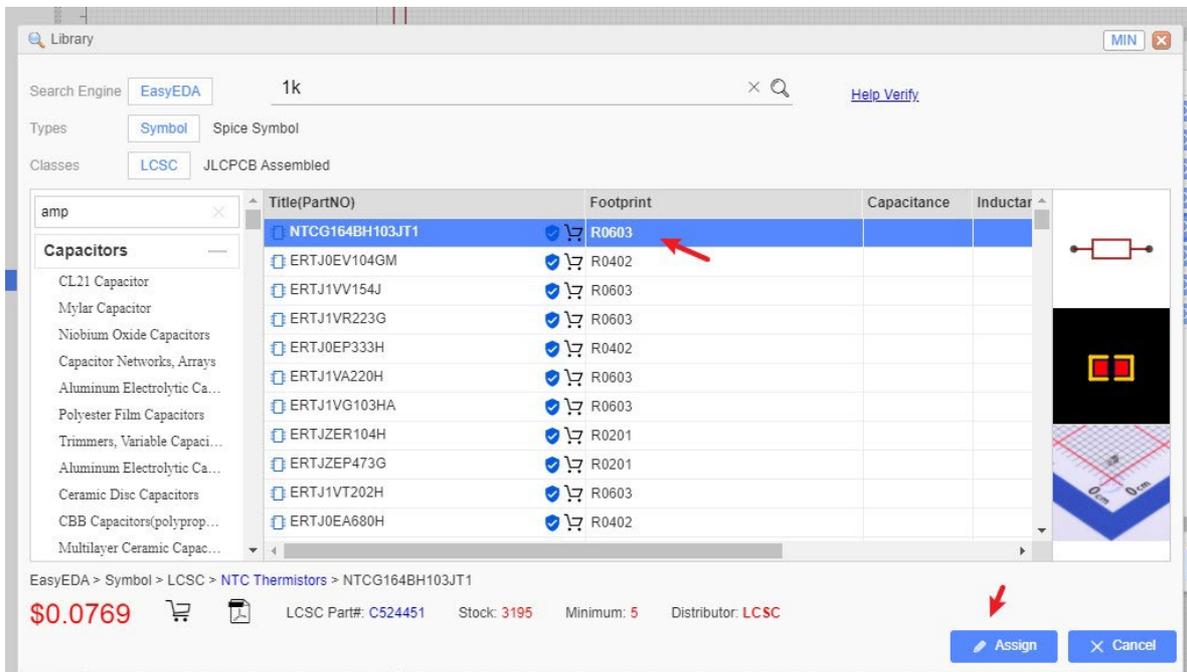


After clicking the BOM export option, the dialog below will open.

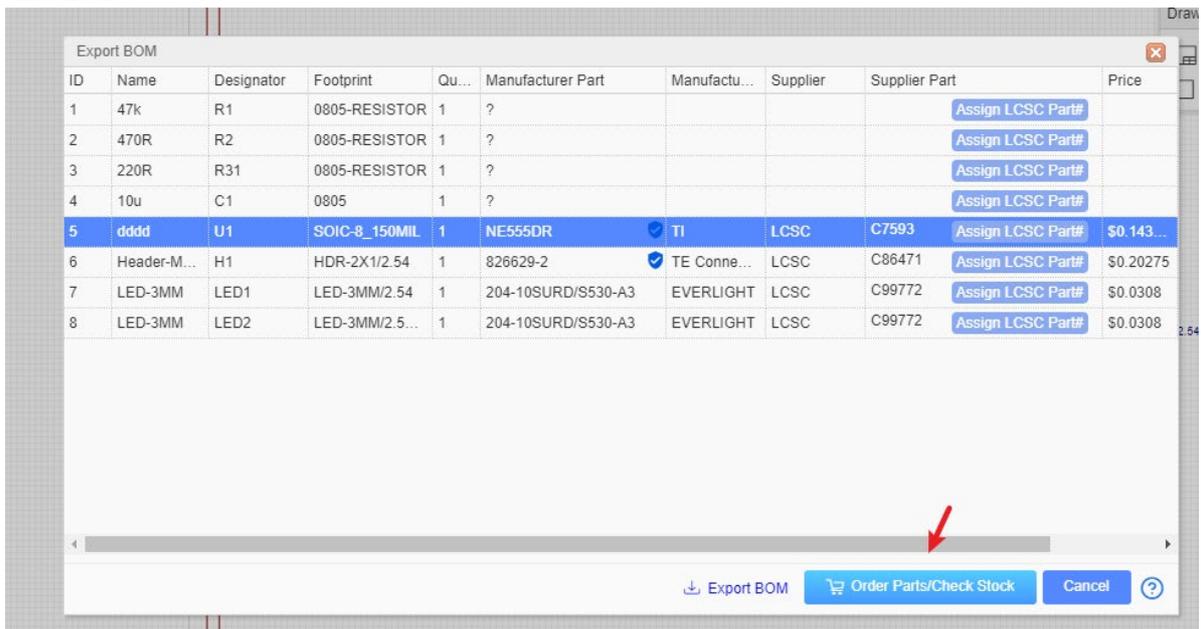
In this dialog, you can click the button to assign LCSC part's order code for your components.



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 form 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	K
	ID	Name	Designator	Footprint	Quantity	Manufacturer	Manufacturer	Supplier	Supplier Pa	LCSC Assembly	
	1	HDR-M-2.54	KJ1,AJ1,BJ1	HDR-M-2.54	8			LCSC	C66690		
	2	NE555P~NA	U1	DIP-8	1	NE555P	TI	LCSC	C46749		
	3	MC306(6pF)	C1	CAP-D3.0X	1	HV010M050	CapXon	LCSC	C59954		
	4	0.1u	C63,C73	C1210K	2						
	5	MC306(6pF)	C8	C1210	1						
	6	19-217/GHC	LED1,LED2	LED0603-R-	2	19-217/GHC	EVERLIGHT	LCSC	C72043	Yes	
	7	1N4148W	KD1,AD1,BI	SOD-123FL	8	1N4148W	Tak Cheong	LCSC	C129216		
	8	CAP-1uF	C2	C0805	1	RVT2A1R0M	HONOR	LCSC	C87863		
0	9	CAP-1uF	C4	RAD-0.1	1	?					
1	10	CAP-1uF	C5	R0805	1	?					
2	11	HDR-IDC-2.	P1	IDC-TH_6P-	1	2X3 2.54mm	BOOMELE	LCSC	C11214		
3	12	0.1u	KC1,AC1,BC	C1210	8						
4	13	1KOHM	R2	R0805	1	?					
5	14	1KΩ	R1	AXIAL-0.3	1	?					
5	15	2N3906(TO-18)	KQ1,AQ1,BI	TO-92-3_L4	8	2N3906	CJ	LCSC	C9809		
7	16	1m	KL1,AL1,BL1	L0402	8						
3											

Export BOM supports to export LCSC part price, it is the same as LCSC website.

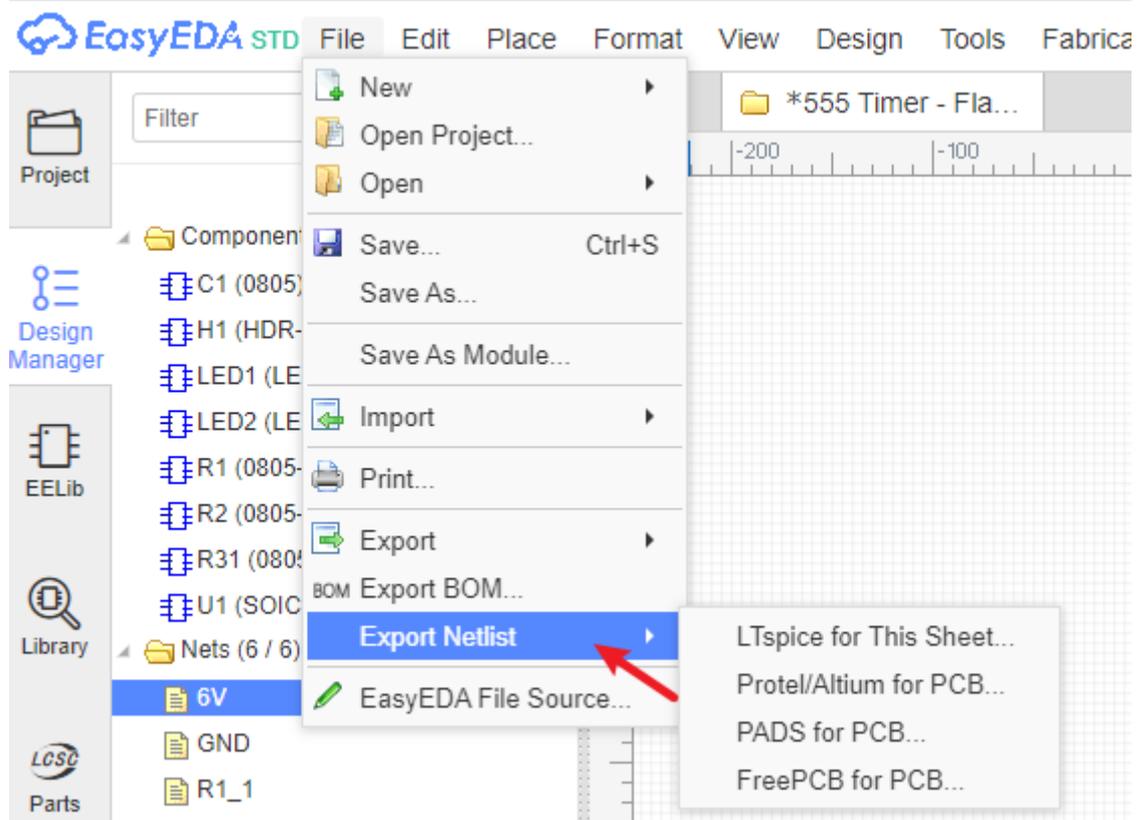
#### Notice:

- Before v6.4.17, 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.
- Since v6.4.17, the schematic BOM and PCB BOM are separated. If you assign the LCSC part at the PCB, it will not modify the schematic.
- 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.

---

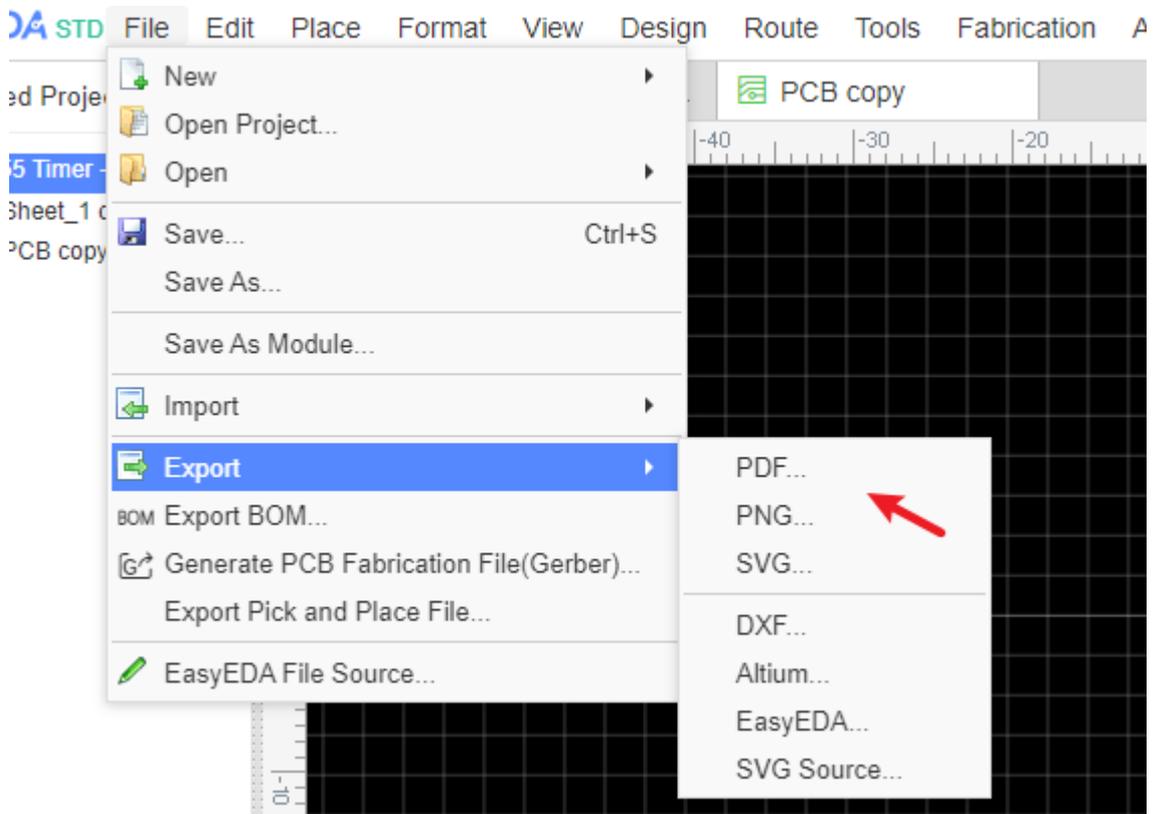
## Export PCB

---

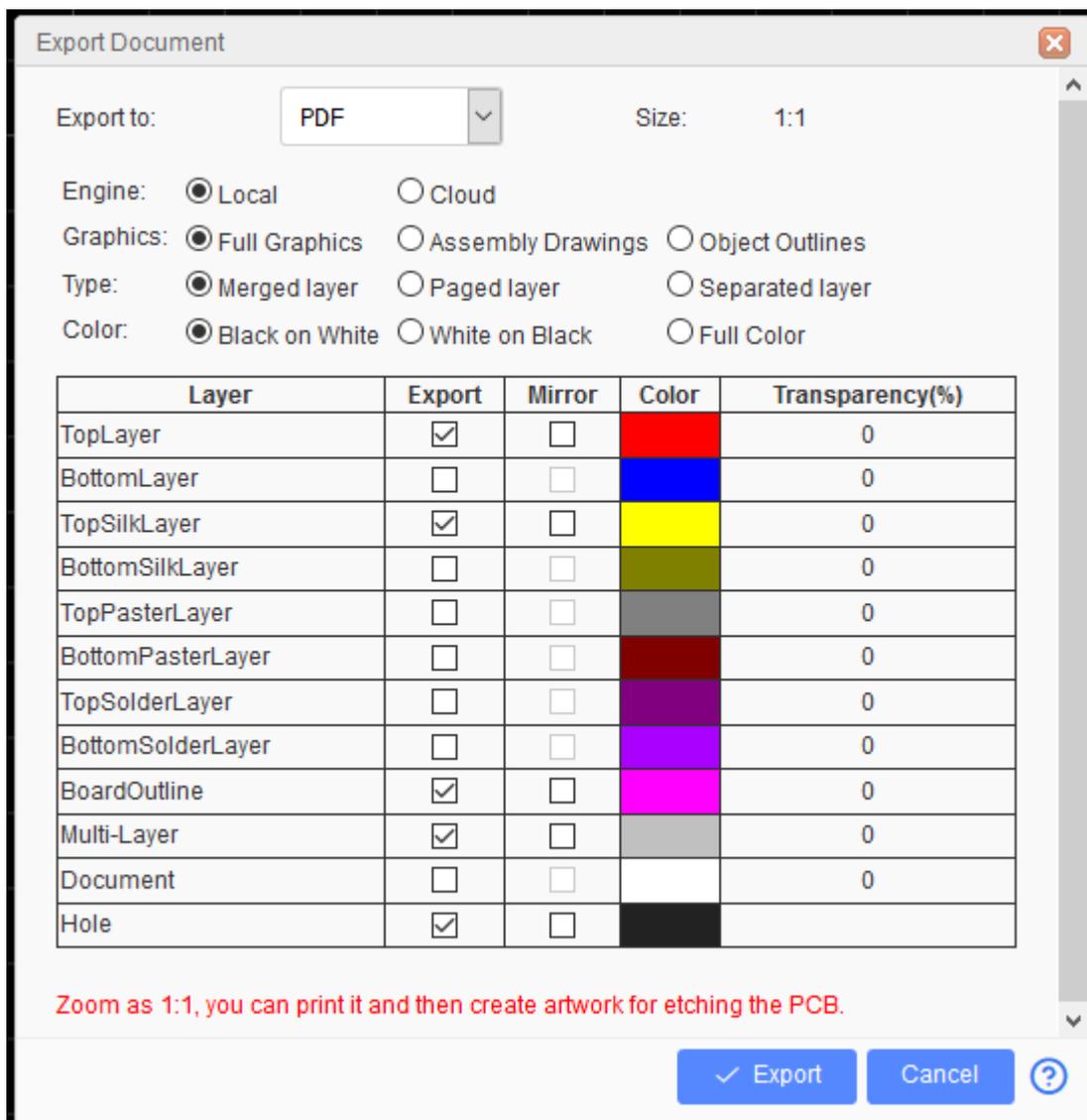
### Export PCB in PDF/PNG/SVG

Exporting a PCB design or footprint from EasyEDA is very similar to exporting a Schematic or a Symbol.

Use: **File > Export > PDF/PNG/SVG...**



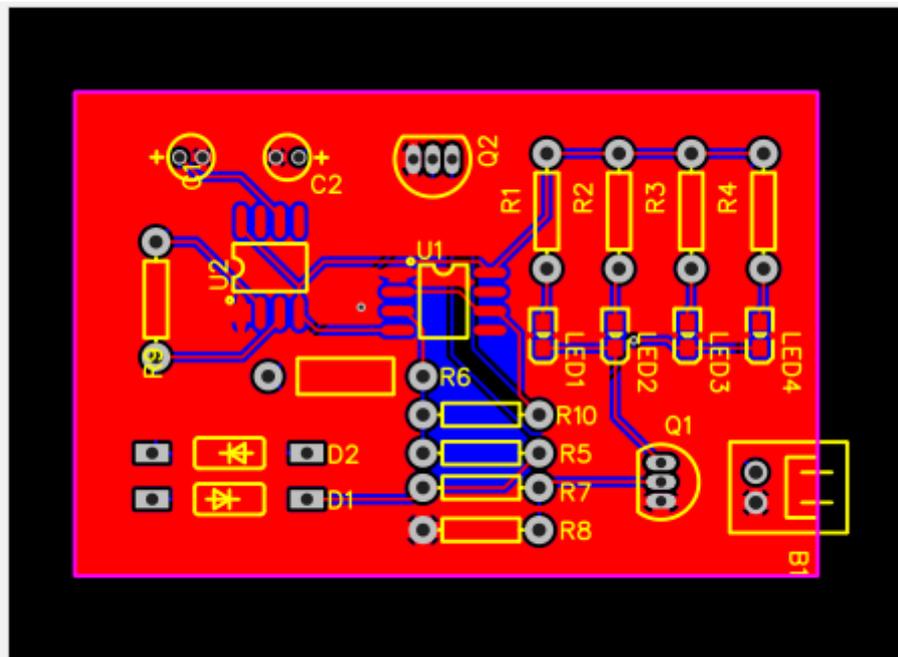
This dialog will open:



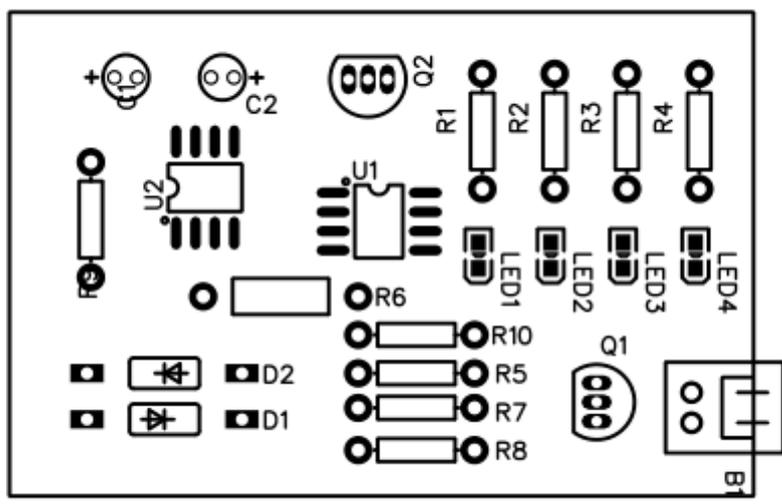
You can choose to export in PDF, PNG or SVG format.

**Note:** \*The PDF size is zoom as 1:1 with PCB. \*

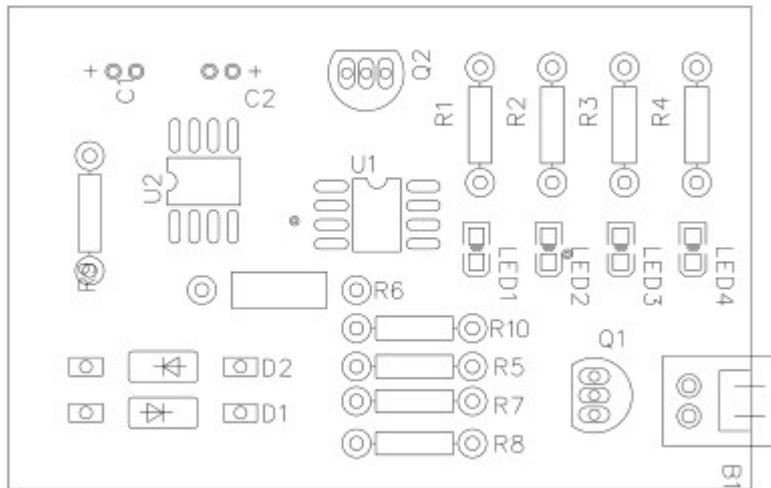
- **Export to:** Support export to PDF, PNG, SVG. To print the PCB as 1:1 you should choose PDF.
- **Engine:**
  - **Local:** PDF generated by Editor
  - **Cloud:** PDF generated by Cloud Server, this feature is planned to be removed in the future.
- **Graphics:**
  - **Full Graphics:** All graphics and objects will be exported.



- This is used for parts assembly.



- **Object Outline:** This only exports the objects outlines, such as Pad and silkscreen outlines.



- **Type:**
  - **Merged layer:** All selected layers will be merged into one page.
  - **Paged layer:** All selected layers will be separate pages in a single file.
  - **Separated layer:** All selected layers will be exported into separate files contained in 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 so that the bottom layers are shown in easily readable orientation. This is recommended when all your selected layers are bottom ones.

If EasyEDA PDF does not satisfy your requirements, please let us know.

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

If you generated the Gerber file, you can use the Gerbv application to easily export the PDF file.

Use: [Gerbv](#)

## Export PCB in Altium Designer Format

The more information please refer at [Export Altium](#)

## Download PCB

Please refer to [Export EasyEDA Source](#)

## Print PCB and Etching

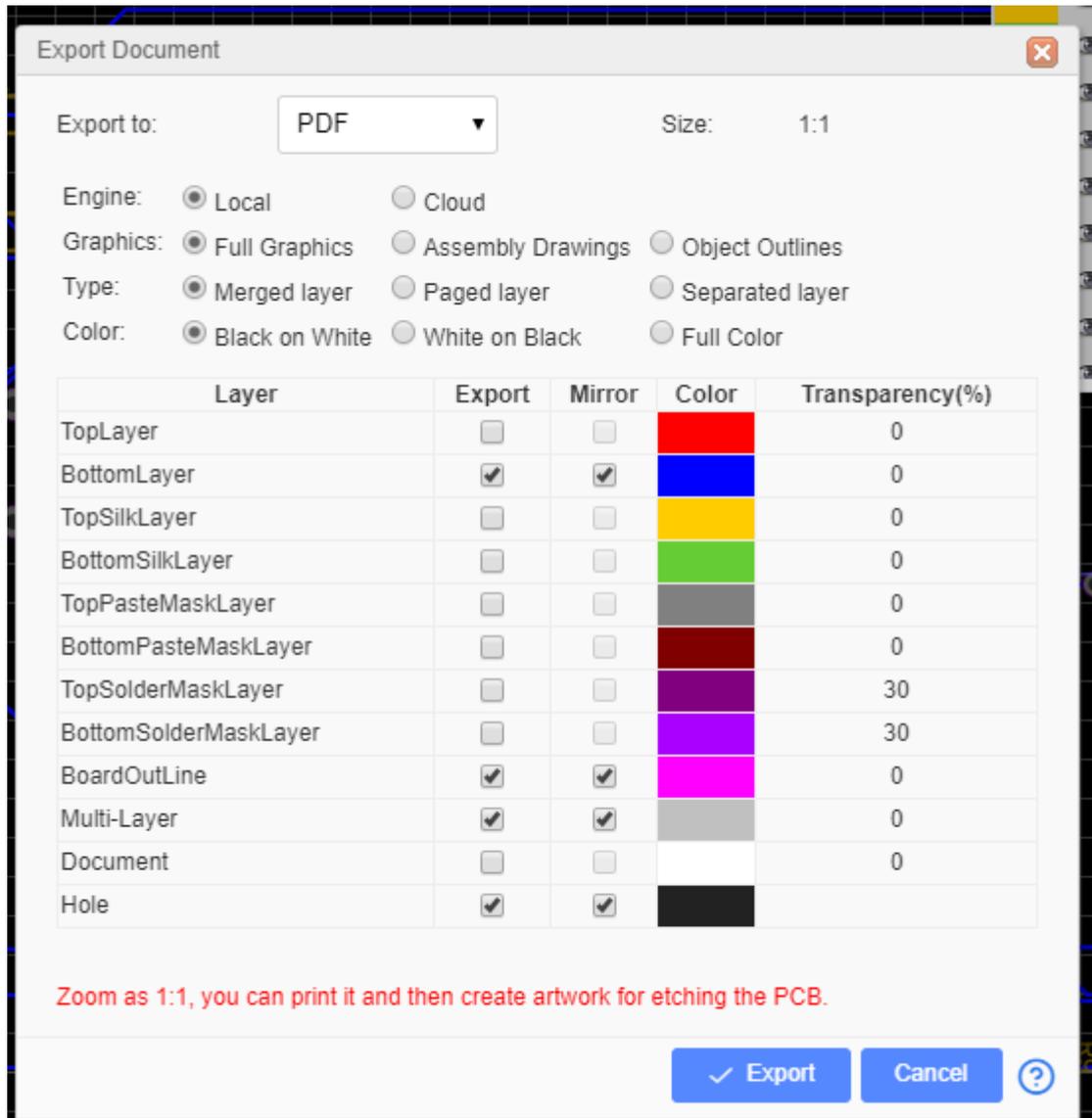
EasyEDA doesn't support directly printing the PCB, instead export to PDF and then print.

If you don't want to order your PCBs from EasyEDA then maybe - for single and double sided PCB designs - you can try using some homemade PCB tech:

<http://hackaday.com/2012/12/10/10-ways-to-etch-pcbs-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...**

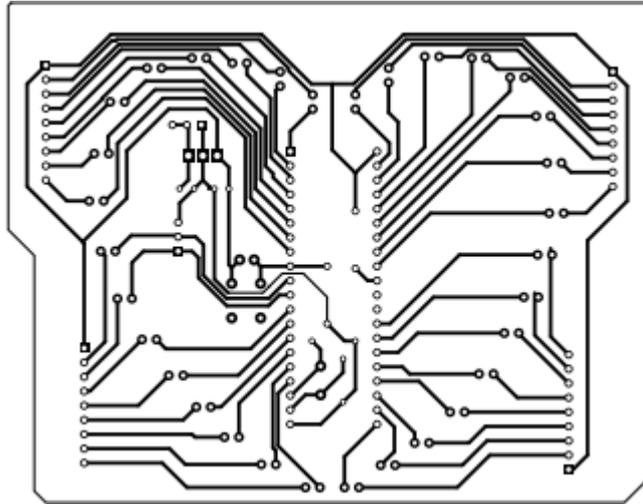


**Note:** Make sure the Colour is Black on White Background.

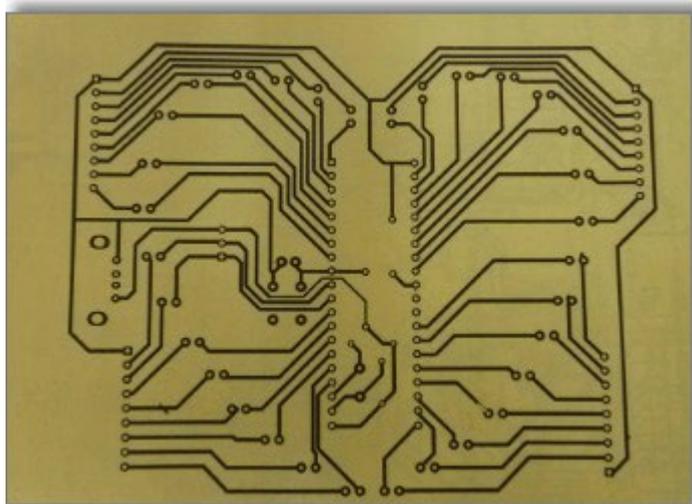
For the bottom layer, select the mirror option on export if needed.

If you have routed PCB tracks on the top layer you will also need to export that layer. Etched PCBs generally need to use the mirror export printing option.

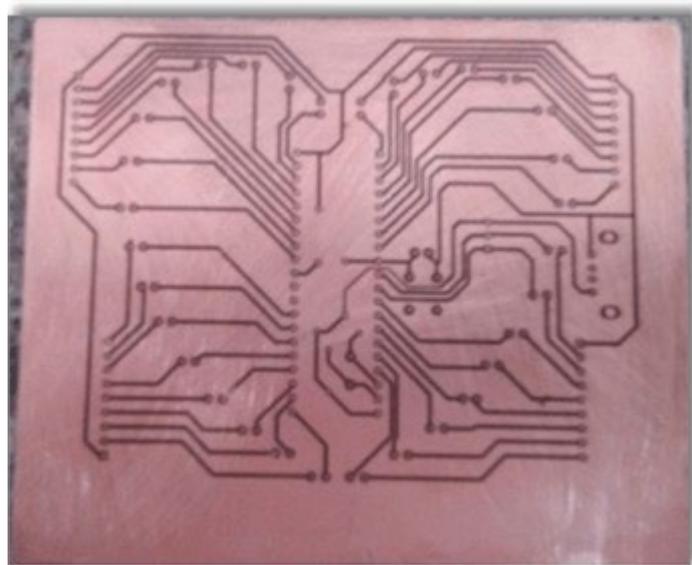
Step 2) Open the pdf file in a viewer



Step 3) Print it to paper

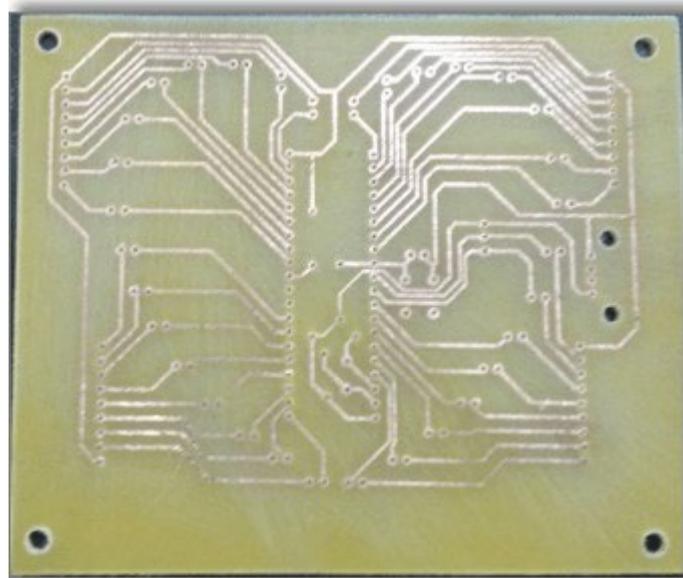


Step 4) Copy it to the copper

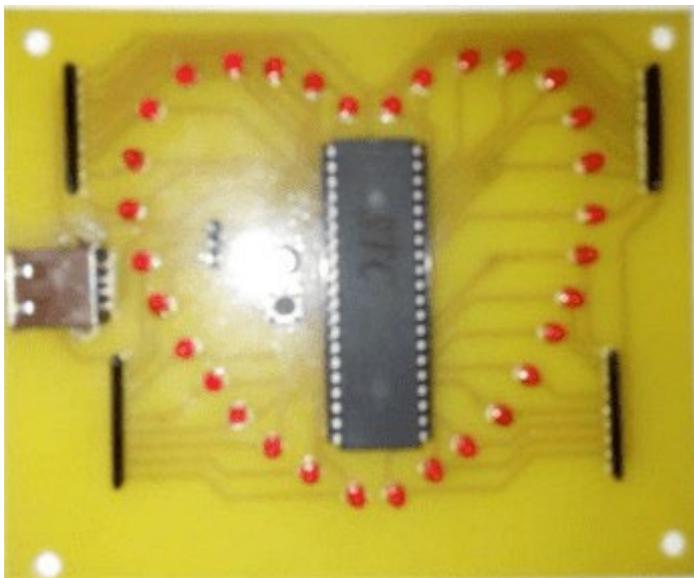


Step 5) Etch it

Step 6) Drill it



Step 7) Get your soldering iron out!



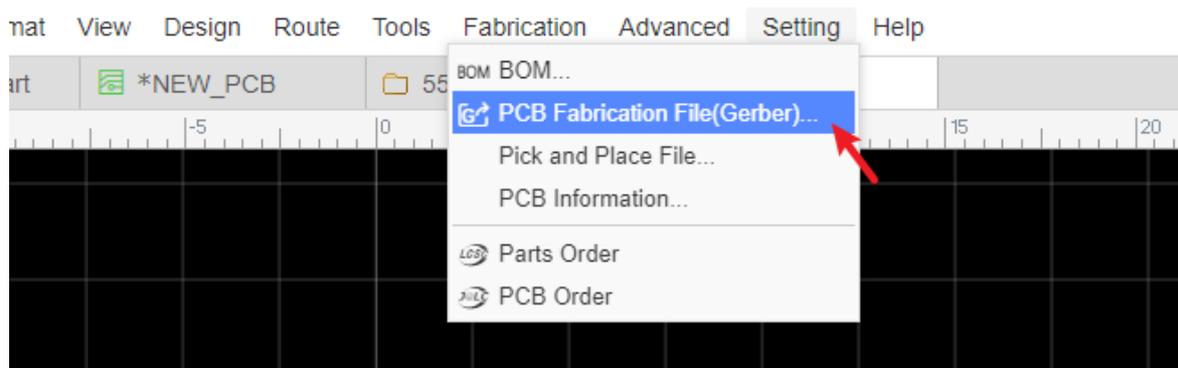
---

## Generate Fabrication File(Gerber)

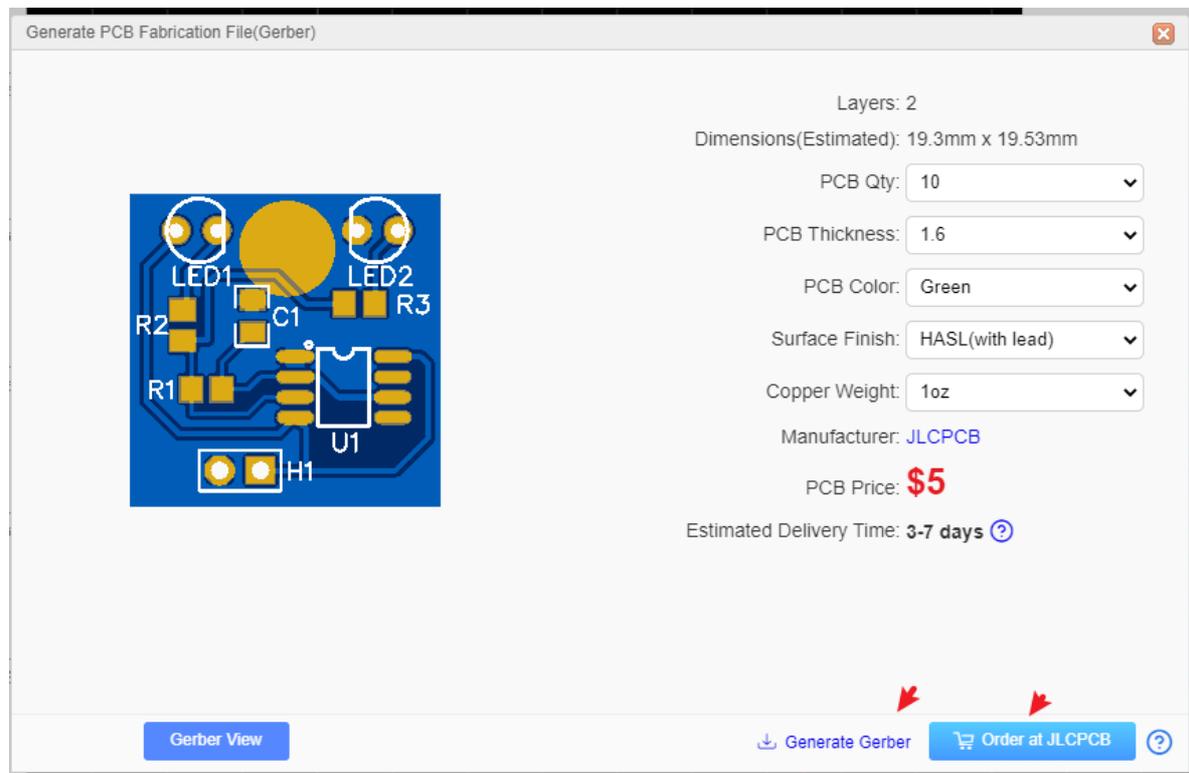
---

### 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\_BoardOutlineLayer.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:** Inner copper layer, signal type.
- **Gerber\_Inner2.GP2:** Inner copper layer, plane type
- **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.
- **Drill\_PTH\_Through.DRL:** Plated drill through hole layer. This document shows the location of the hole where the inner wall needs to be metallized. Old name: Gerber\_Drill\_PTH.DRL
- **Drill\_NPTH\_Through.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. Old name: Gerber\_Drill\_NPTH.DRL
- **Gerber\_TopPasteMaskLayer.GTP:** Top Paste Mask, for the stencil.
- **Gerber\_BottomPasteMaskLayer.GBP:** Bottom Paste Mask, for the stencil.

- **Gerber\_TopAssemblyLayer.GTA:**Top Assembly, read only, doesn't affect the PCB manufacture. Old name: ReadOnly.TopAssembly
- **Gerber\_BottomAssemblyLayer.GBA:** Bottom Assembly, read only, doesn't affect the PCB manufacture. Old name: ReadOnly.BottomAssembly
- **Gerber\_MechanicalLayer.GML:** Record the information on the mechanical layer in the PCB design, and only use it for information recording. Old name: ReadOnly.Mechanical. 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. If the mechanical layer has closed wires, JLCPCB will give priority to using the mechanical layer as the shape of the board when producing the board. If there is no outer frame of the mechanical layer, GKO will be used as the frame (historical influence of Altium file). It is necessary to pay attention to the use of the mechanical layer in the design.

**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!
- The coordinates of the Gerber file follow the canvas coordinates
- When exporting Gerber, the coordinate format accuracy defaults to 3:3. When the PCB size is out of range, it automatically uses 4:2 format. If you view the Gerber as such as CAM350, found that the Drill hole has been offset the location, you can modify the drill coordinate format to fit the location

## 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.

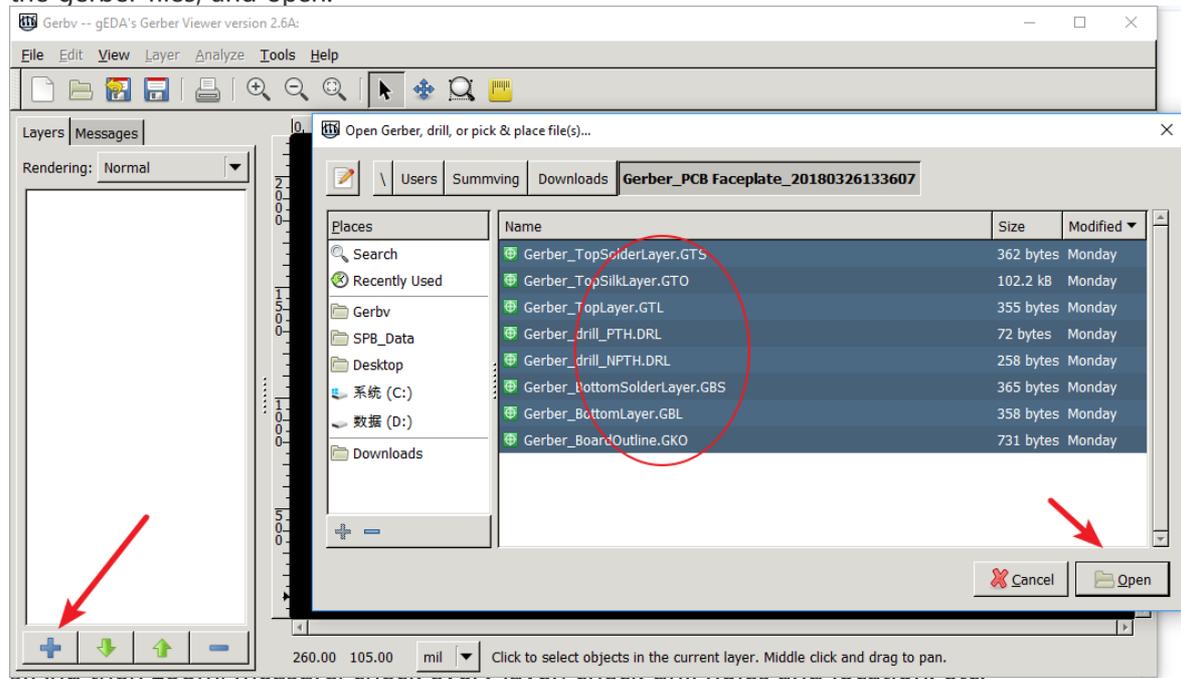
Gerber viewer recommend Gerbv:

- 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.



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)

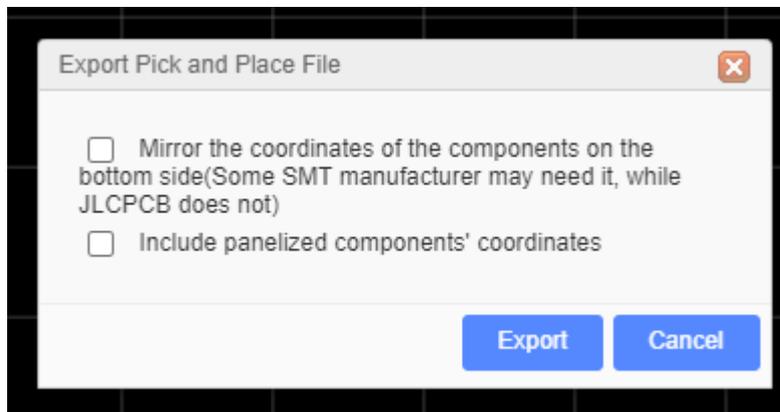
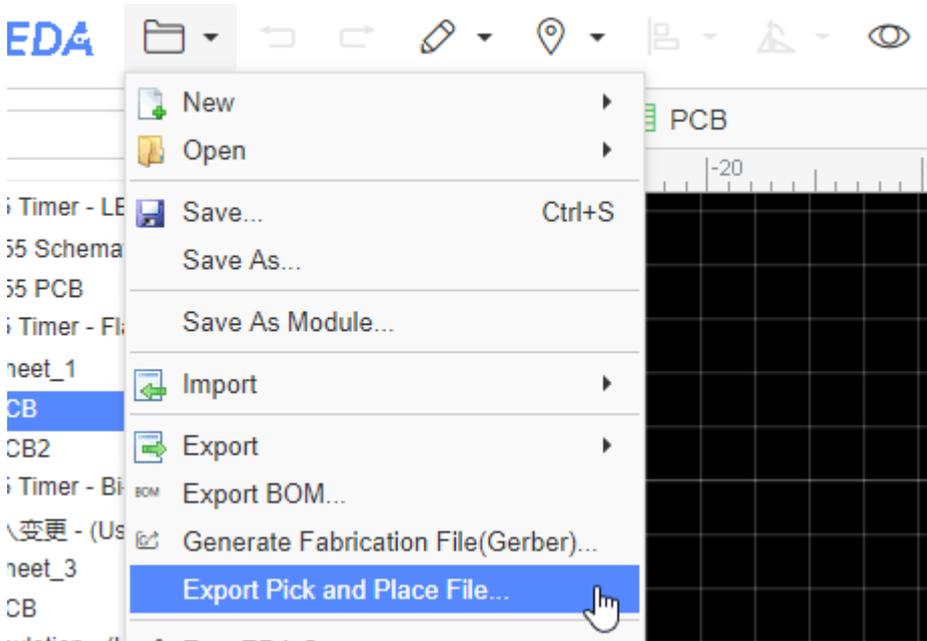
---

## 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.



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/2.5	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_150N	13.31mm	7.49mm	10.92mm	9.4mm	10.29mm	9.4mm	T	0	NE555DR		
5	LED1	LED-3MM/2.5	4.16mm	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		
10													

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.

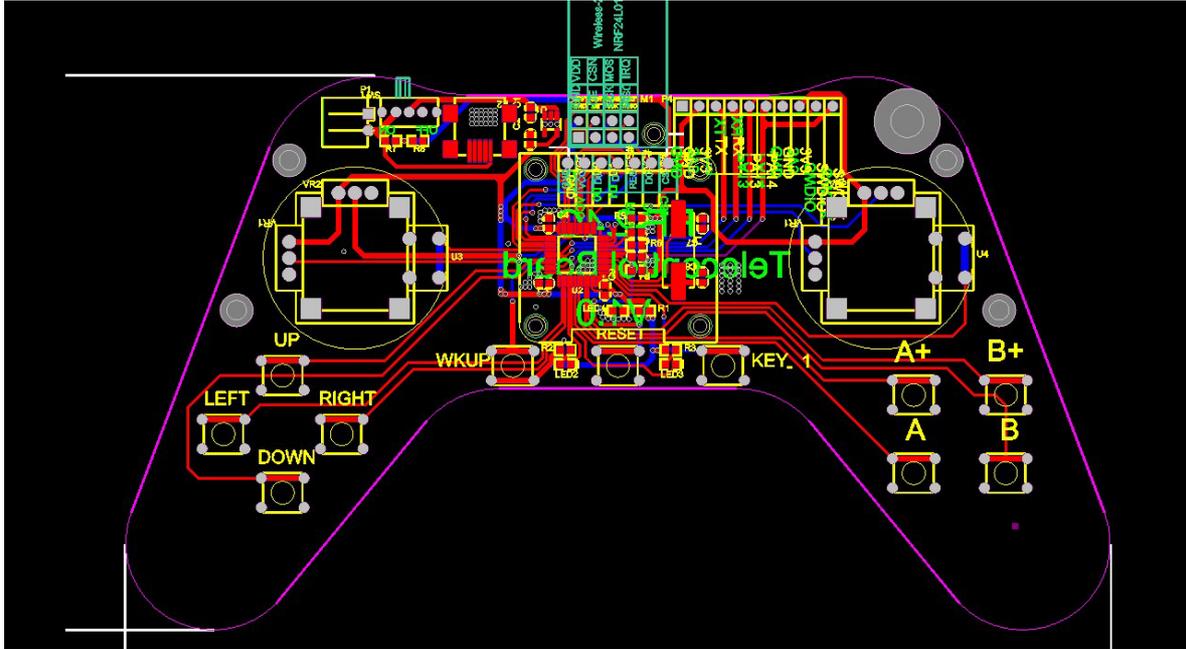
---

## Export DXF

---

EasyEDA support to export PCB to DXF.

At present EasyEDA supports to export a full layers and objects DXF file:



You can edit it at CAD tools very easy, toggle the layers as you want.

---

## Export Altium Designer Format

---

EasyEDA supports exporting the schematics and PCB in Altium Designer format.

**The "export to Altium" function is now in beta, Please check carefully after exporting the design to Altium, EasyEDA cannot guarantee that there are no errors!!! EasyEDA is not responsible for any loss due to library errors or format conversion!!! If you do not agree, please do not use Altium export!!!**

**If you want to order a PCB please generate a Gerber file instead of exporting to Altium! Please do not export your design to Altium and import it more than once as this may cause some details to be lost!!!**

**Altium version 19 is not supported yet, please open exported file with Altium version 18 or before, version 17 is recommended**

If you find any errors in the export details, please contact us so we can fix it, to help us please include the problem details and files.

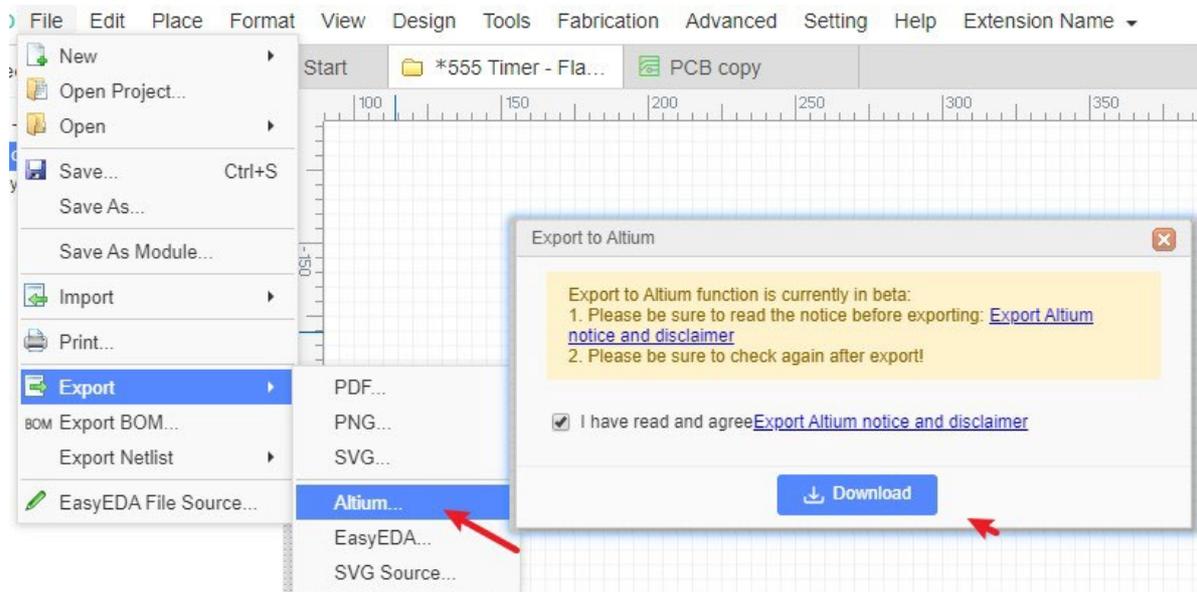
[support@easyeda.com](mailto:support@easyeda.com)

When exporting, you must log in first before exporting.

## Exporting Schematics In Altium Designer Format

EasyEDA supports exporting the schematics in Altium Designer format.

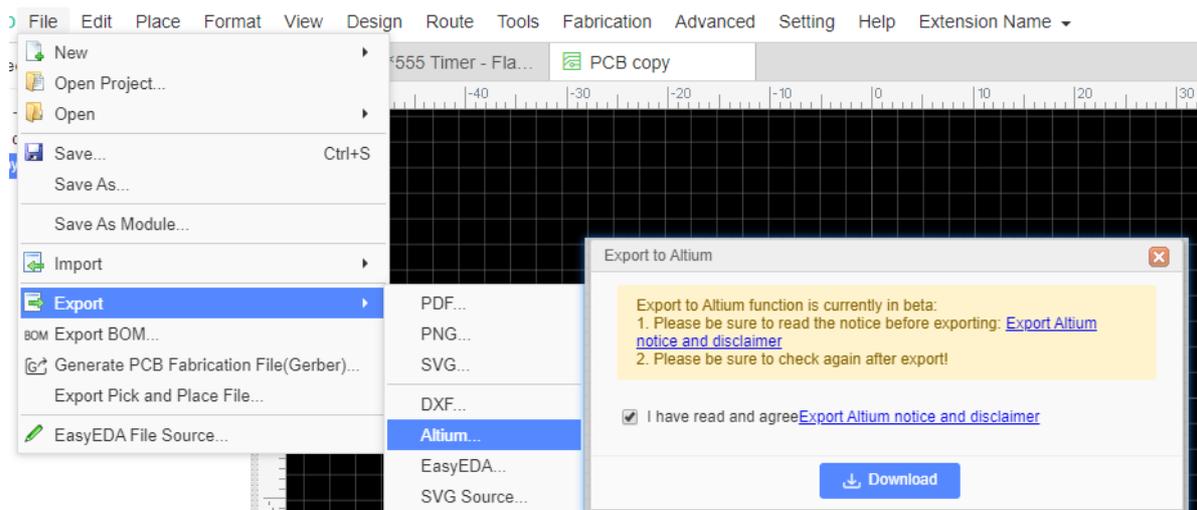
Use "File > Export > Altium...", clicking "Download" will create a `.schdoc` file.



## Exporting PCB in Altium Designer Format

EasyEDA supports exporting the PCB in Altium Designer format.

Use "File > Export > Altium..." to create a `.pcbdoc` file.



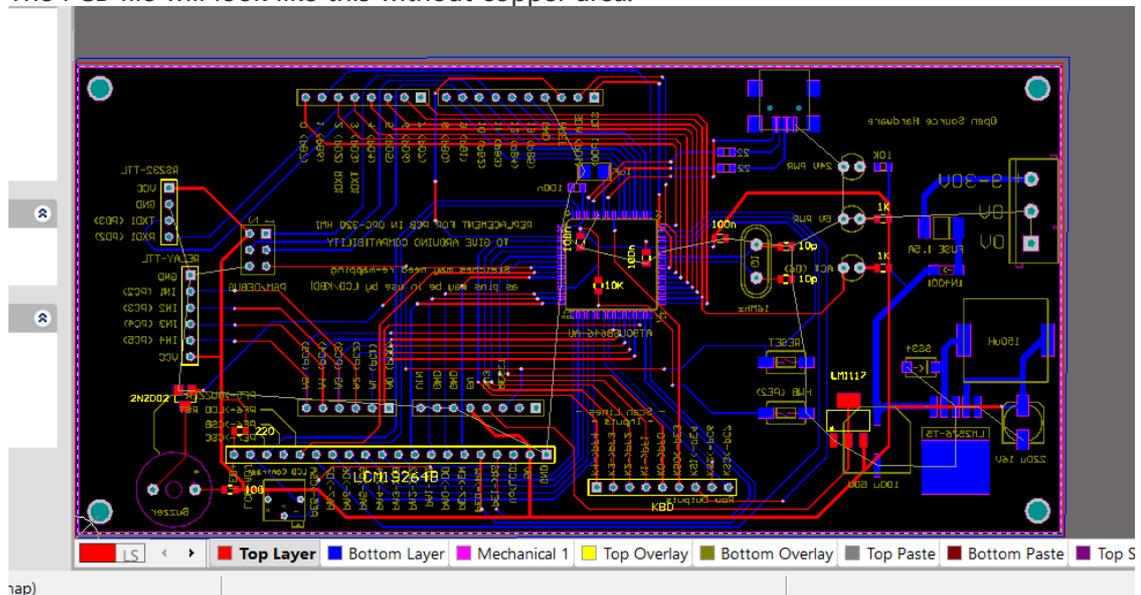
When opening the exported PCB file in Altium Designer there will be a dialog "DXP Import Wizard", don't worry, just cancel it to continue.



**Known Issues:**

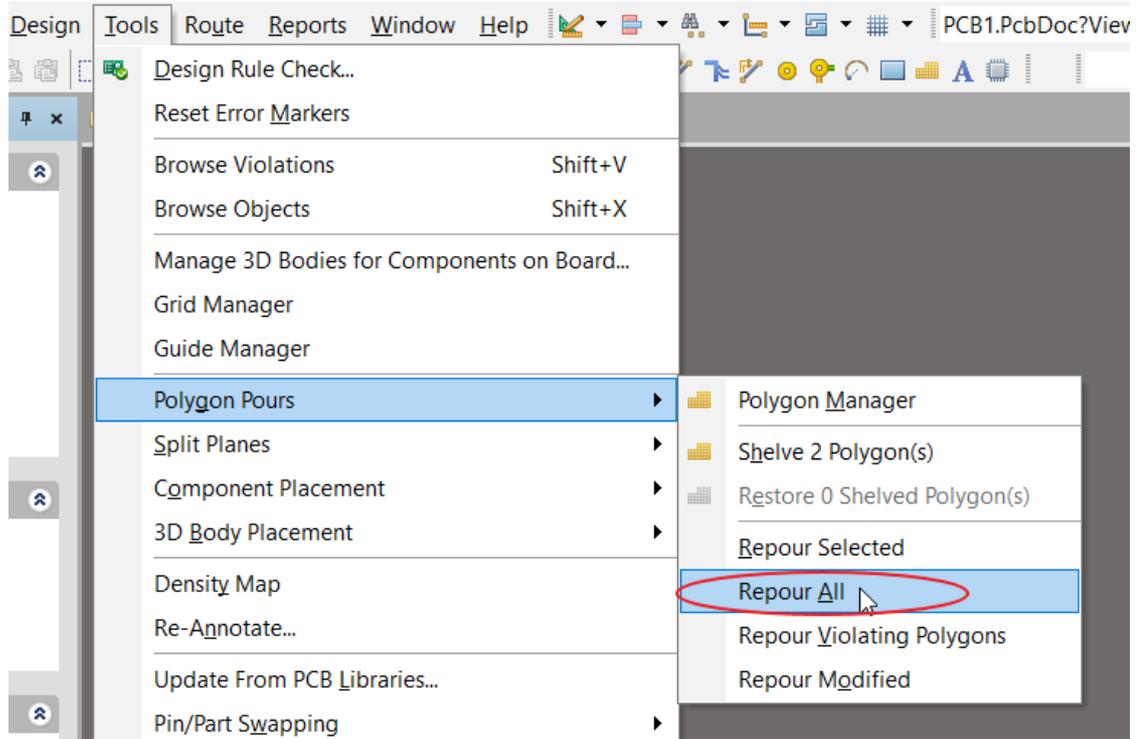
- 1. No Copper Area fill data.

The PCB file will look like this without copper area:

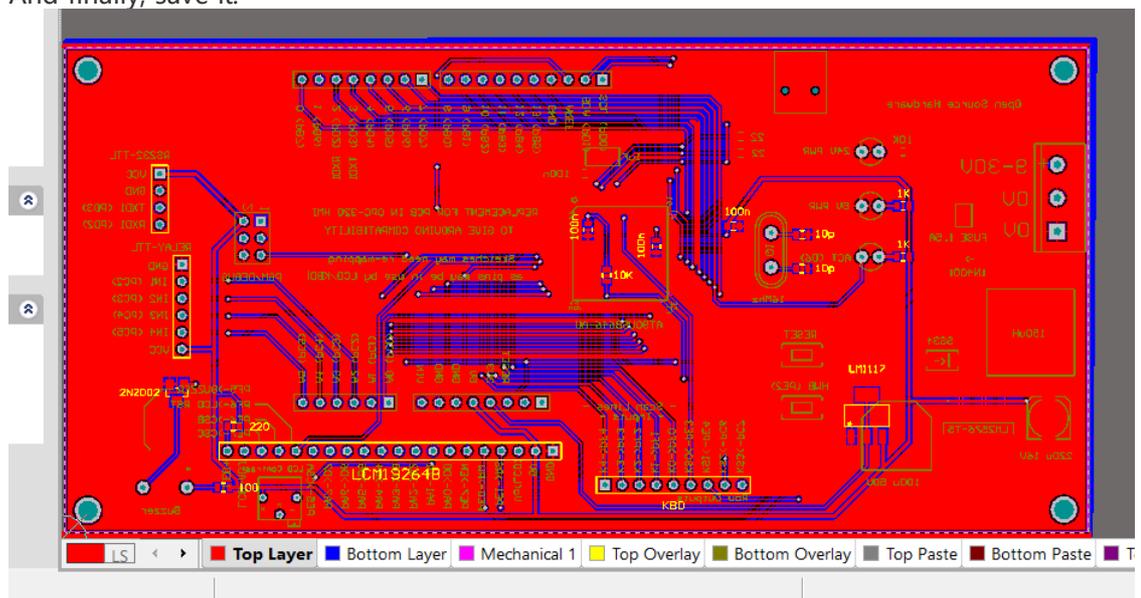


Repour All:

ee Documents. Not signed in.

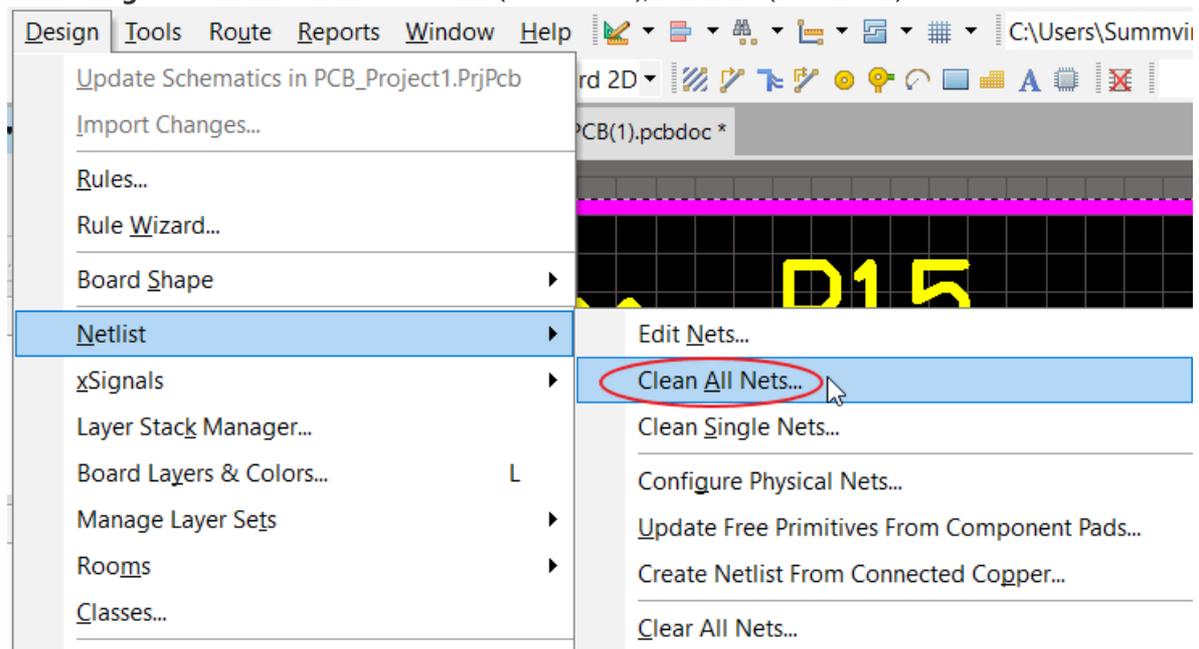


And finally, save it.

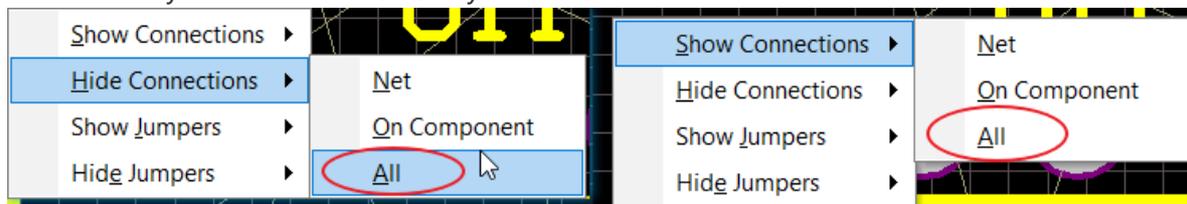


- 4. NO NAMING.

If you export the PCB without ratlines, you need to show all connections first before routing :  
Use: **Design > Netlist > Clean All Nets (D > N > A)**,and then **(V > C > S)**



Or use hotkey: **N > H > A** followed by **N > S > A**:



- **3. Inner layer Plane Zone doesn't export perfectly.**

You need to rebuild the plane zone and re-assign plane zone's net.

- **4. Doesn't support DRC rule.**

Please check the DRC manually.

- **5. The text may be changed.**

Some text may change position depending on the chosen font. If the text does not display correctly just manually adjust the position.

## Exporting Footprint and Symbol in Altium Designer Format

EasyEDA does not support exporting the Symbol or Footprint in Altium Designer library format, but you can place the libraries with the schematic or PCB and export that in Altium Designer format, and then extract the libraries from Altium Designer.

**EasyEDA is not responsible for any loss due to library errors or format conversion!!! If you do not agree to this, please do not use Altium export!!!**

---

## Export SVG Source

---

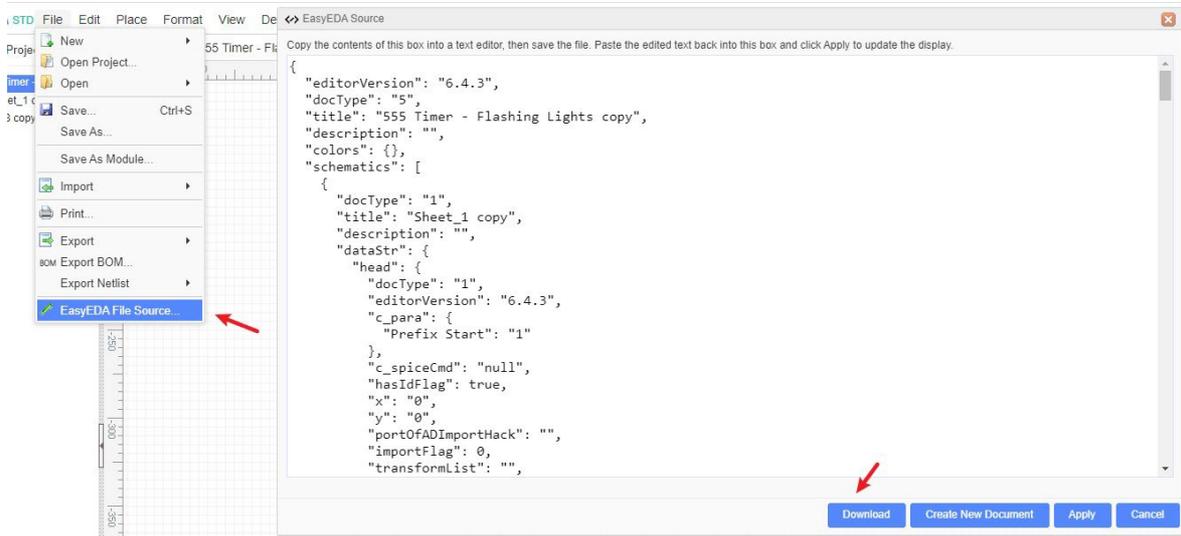
EasyEDA supports exporting or editing SVG source.

You can create an SVG source file using:

**File > Export > SVG source...**

First copy the contents of this box into a text editor and then save the file with the .svg extension. You can edit it in [Inkscape](https://inkscape.org/) or open it in your browser.

This solution doesn't need an Internet connection, you can use it off-line with EasyEDA.



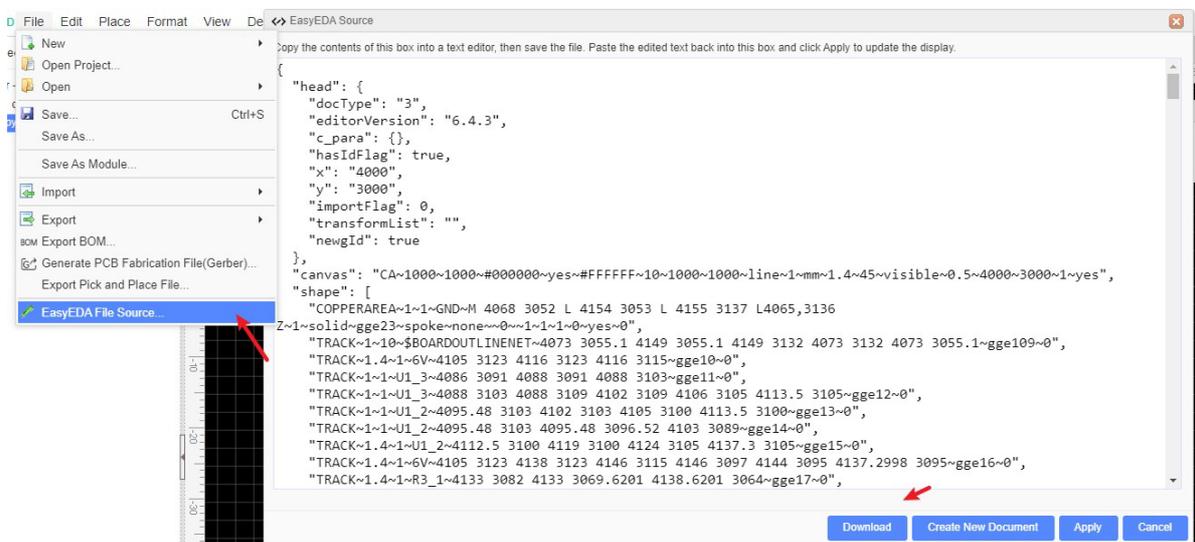
## Export EasyEDA Source

EasyEDA supports saving your file to local storage, you can download your design as an EasyEDA source file.

### 1. Export EasyEDA document directly

You can create an EasyEDA source file with:

**\*\* > File > EasyEDA File Source... \*\***



### 2. Download the project