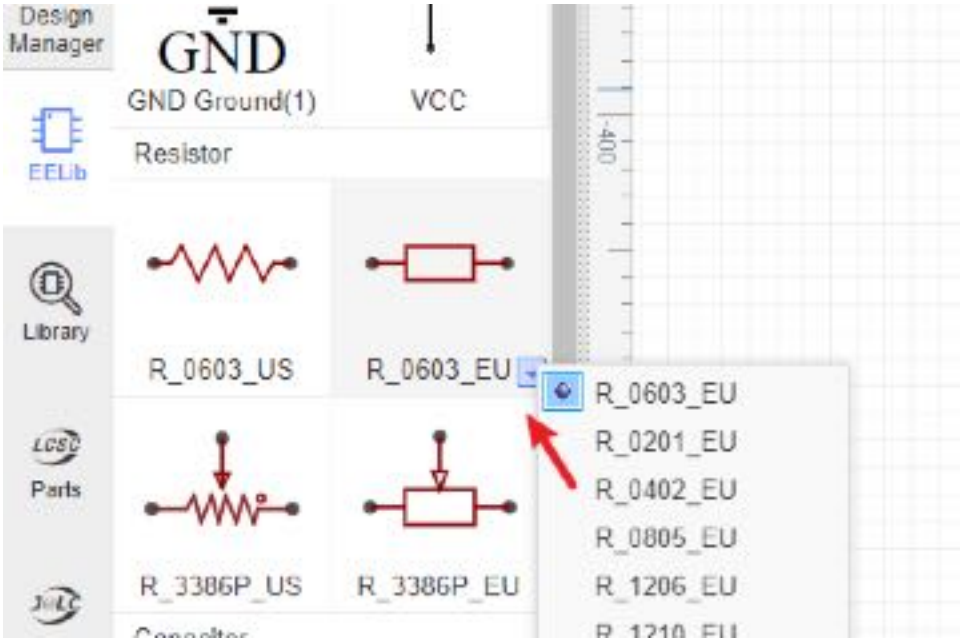
Libraries

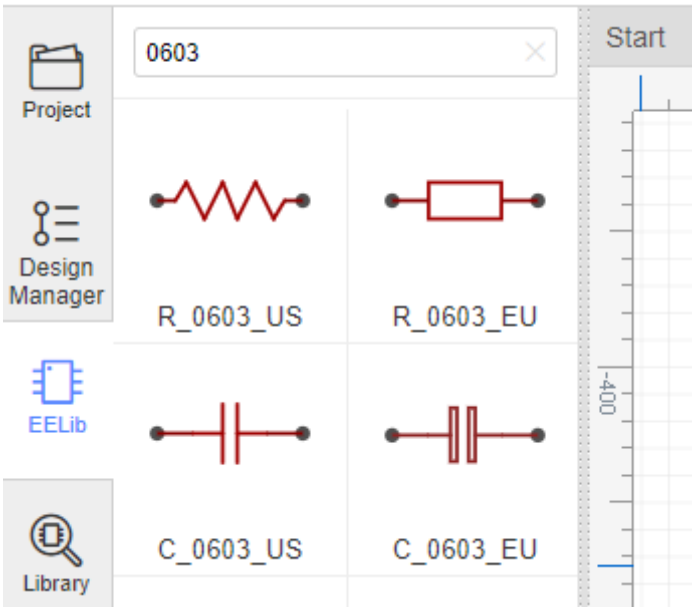
EELib

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



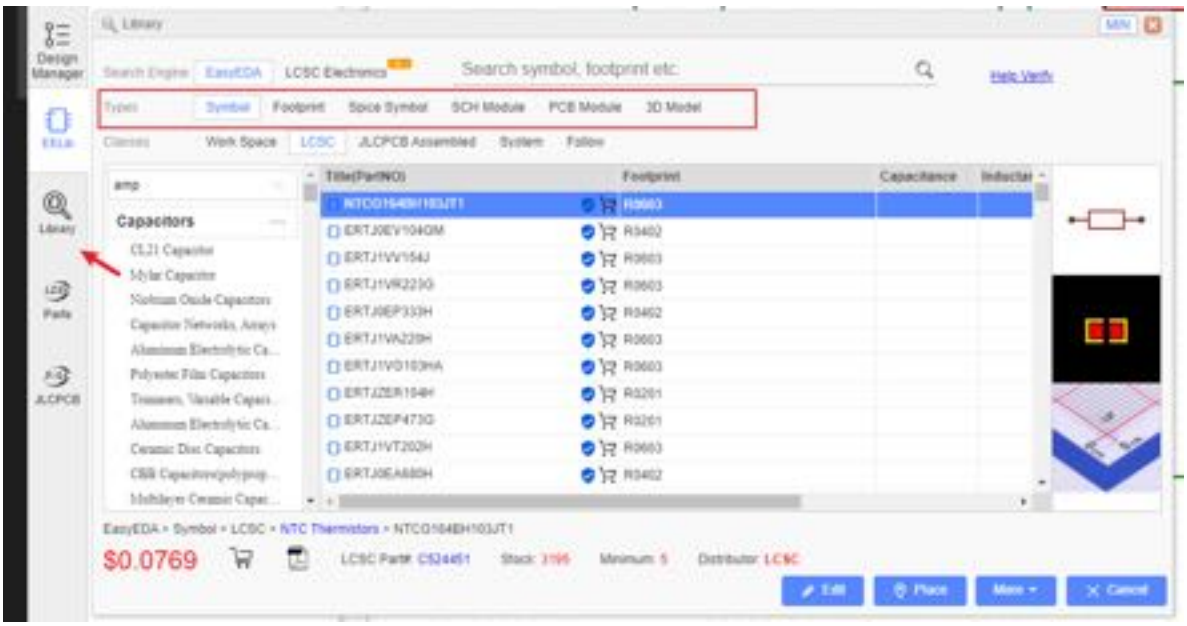
Many of these components have optional US and EU style symbols, we split them, so you can select those you like. Click on the drop down list or right click to popup the context menu, it contains many footprints or parameters. EasyEDA will remember your choices for the next time.

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



Library

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



Type

- Symbol: Schematic symbols
- Spice Symbol: Symbols for spice simlation
- 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 parts(Officail 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

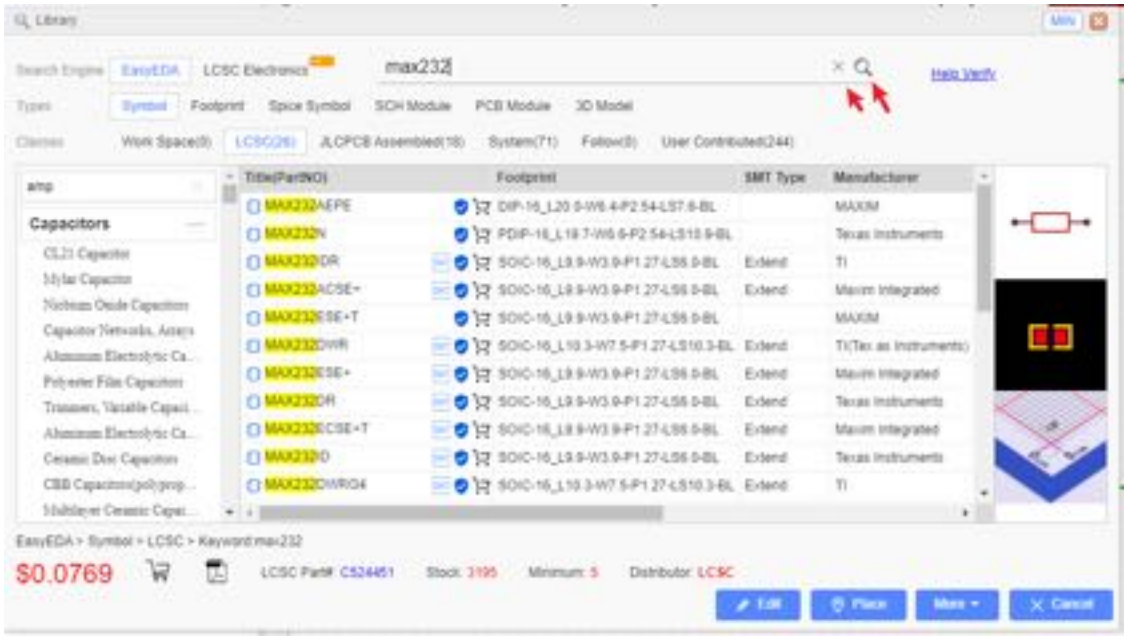
Search Engine - EasyEDA

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

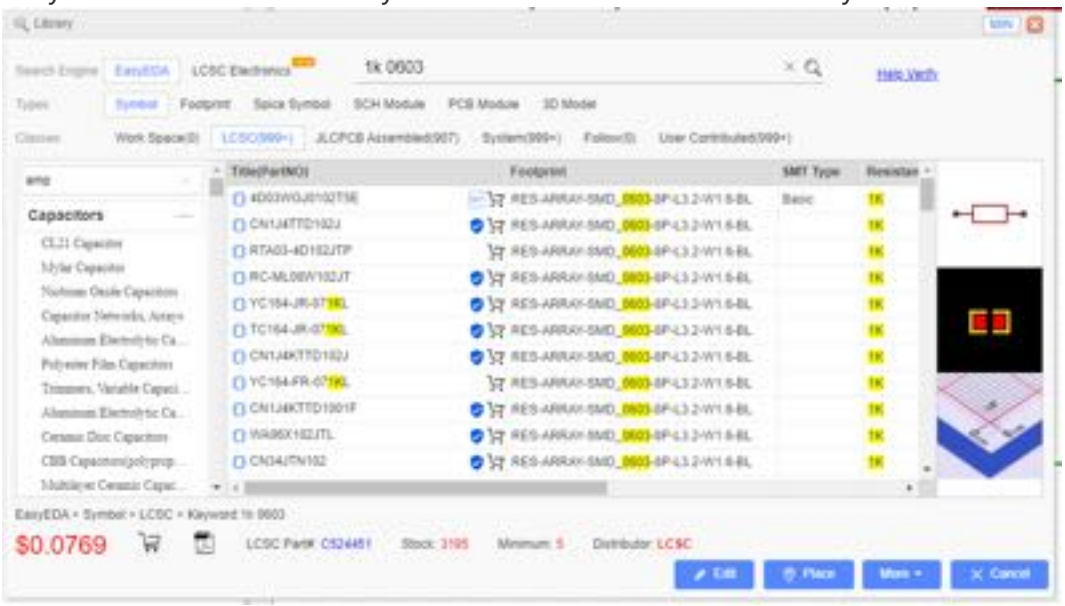
and then click the "Table of contents" to open the categories list to choose your components.

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:

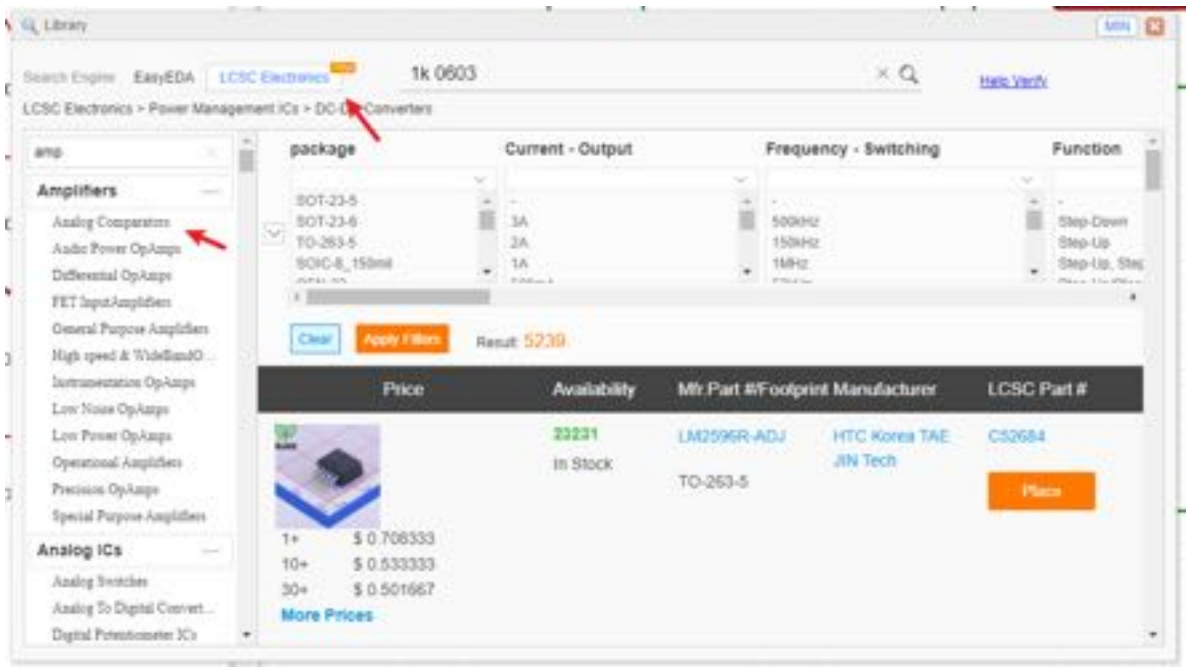


- If you don't know the component's name
  For example, you want to find a resistor which value is 1kohm, footprint is 0603, at Libraries you can follow below steps:
  - 1.Choose the library type
  - 2.Typing the keyword such as `1k 0603`
  - 3.Click the search button
  - 4.Select the class you which is wanted of the result
  - 5.If you don't need the search you need to remove all the search keywords

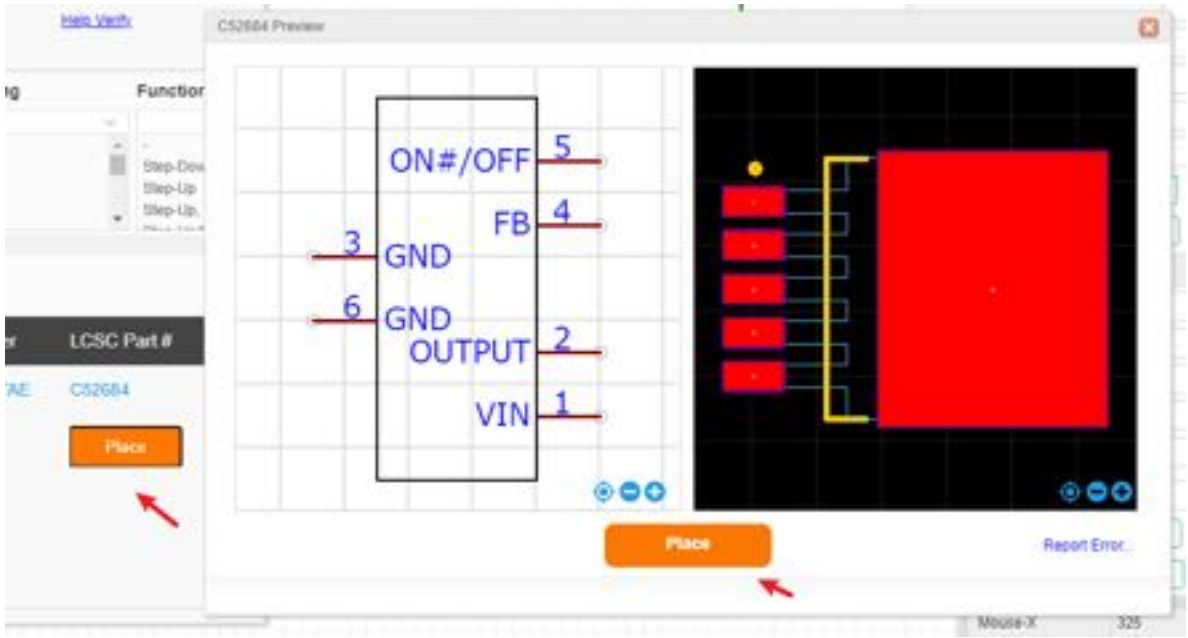


Search Engine - LCSC Electronics

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

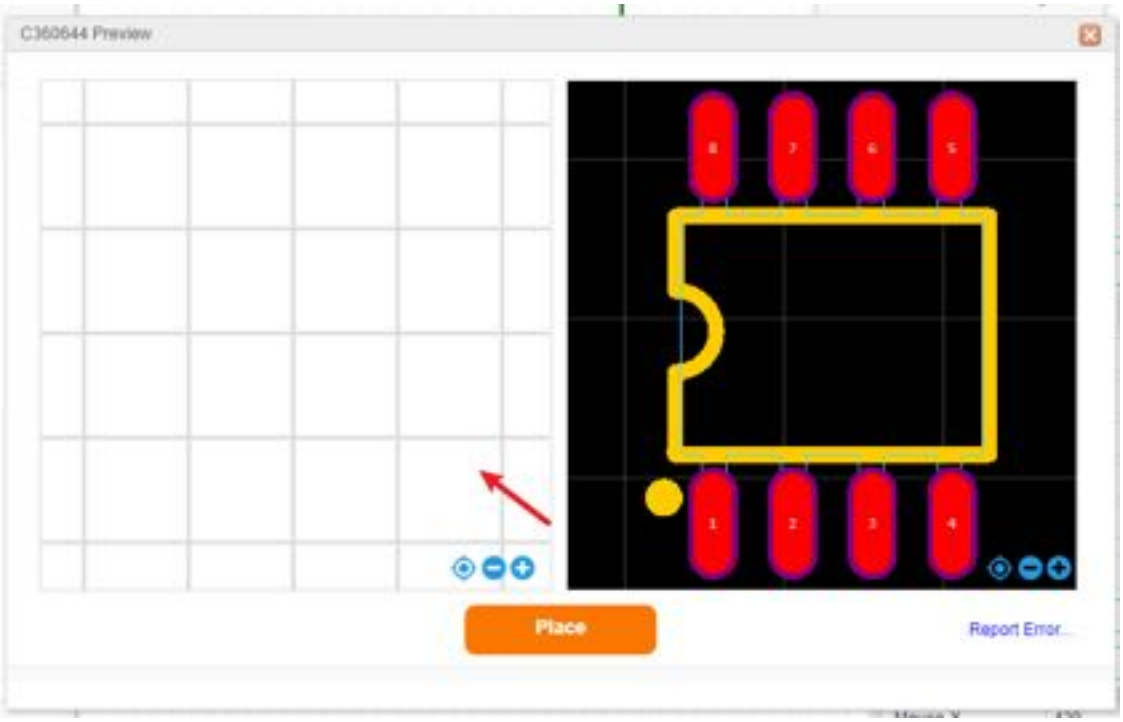


When you find out part, and you can place into the schematic:



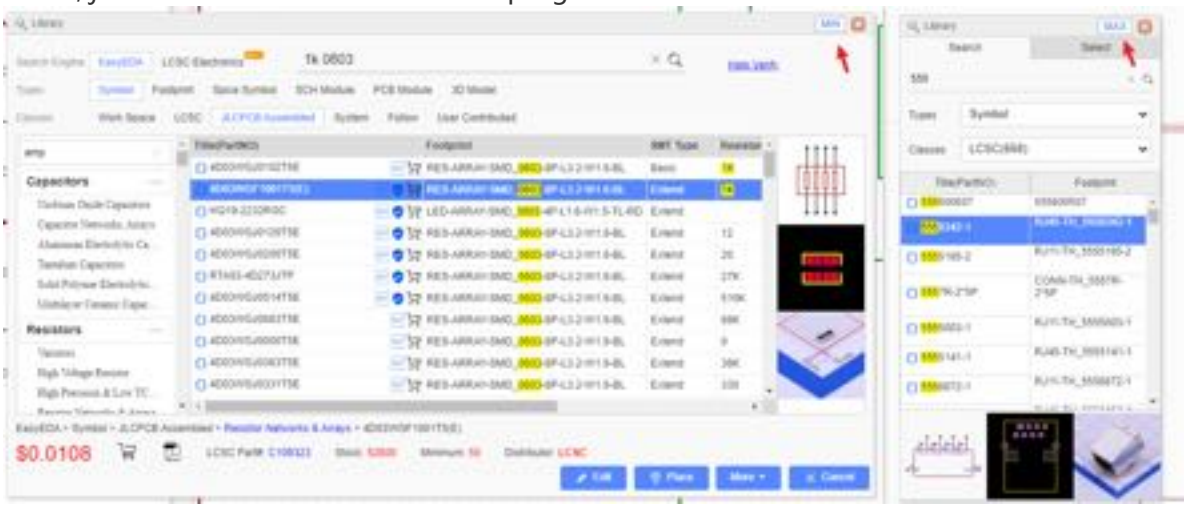
Notice:

- *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 known. suggested order a sample first before*

ordering a big order.

Edit:

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

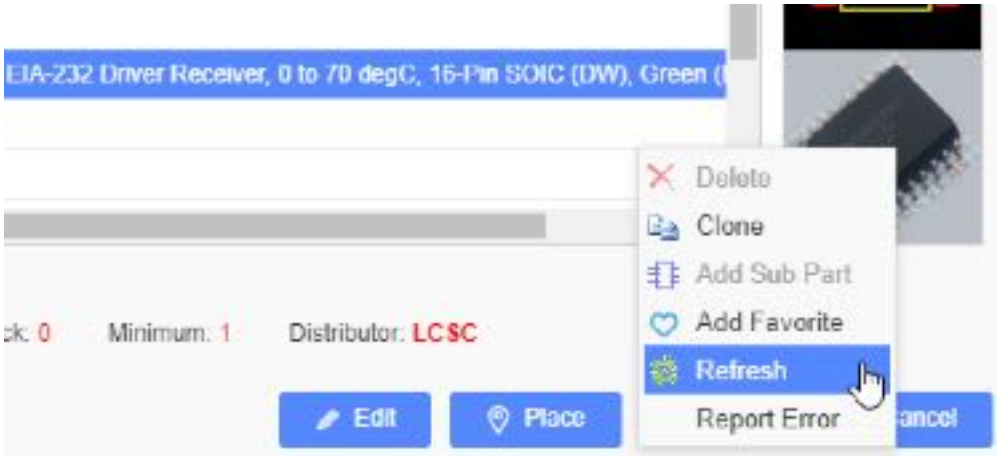
More:

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

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

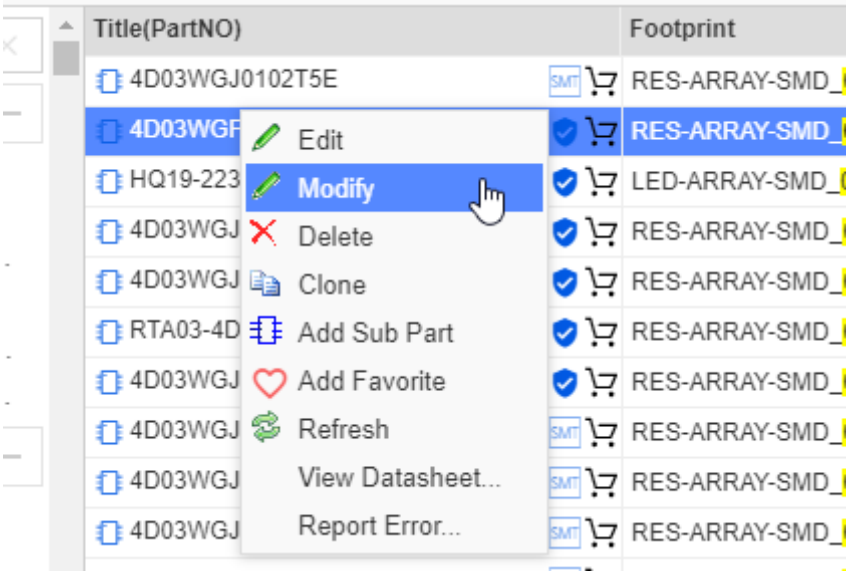
Components with sub parts (multi-device footprints).

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



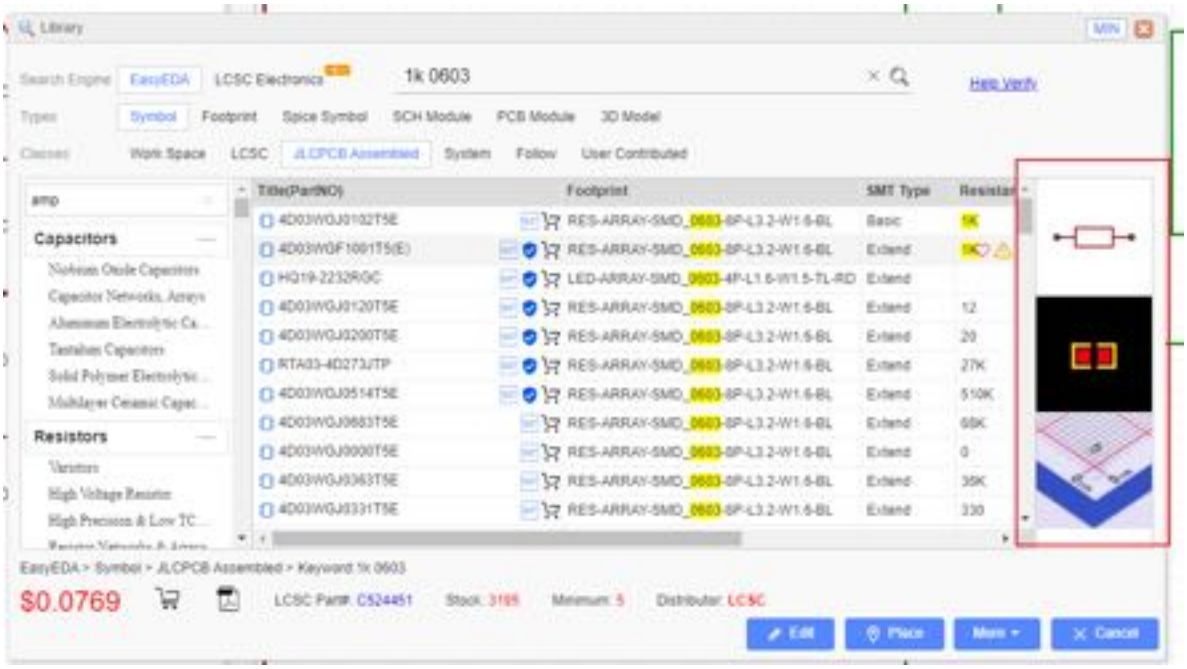
Right-Click

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



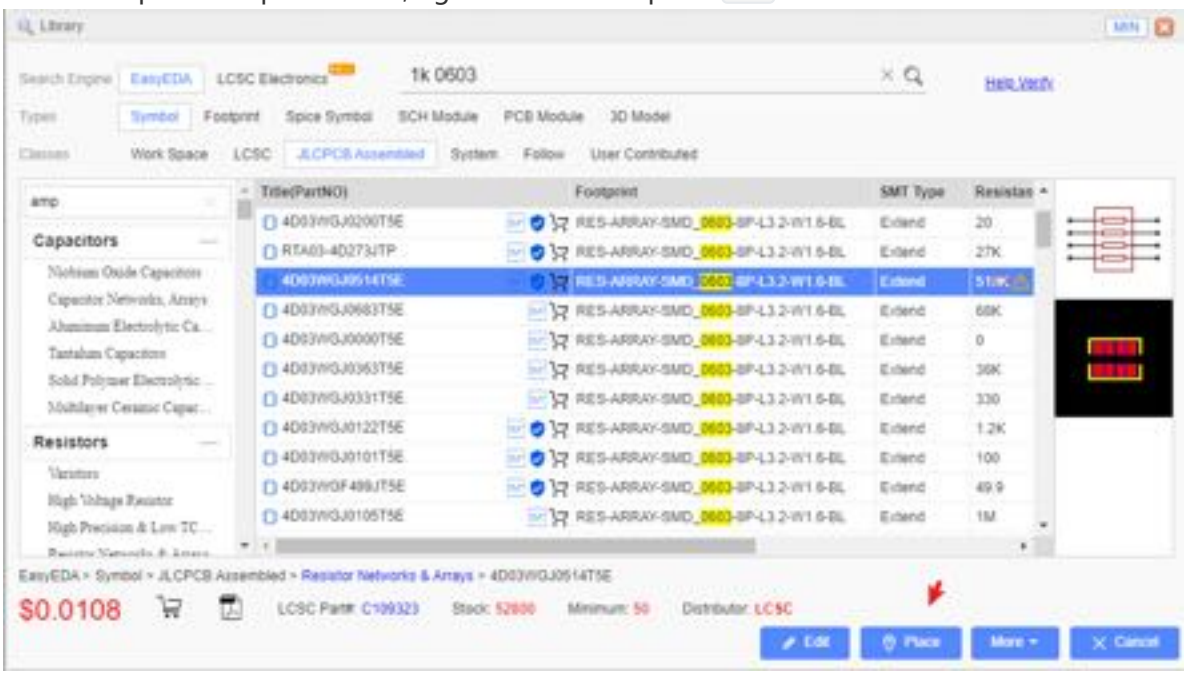
Preview Image

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



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.

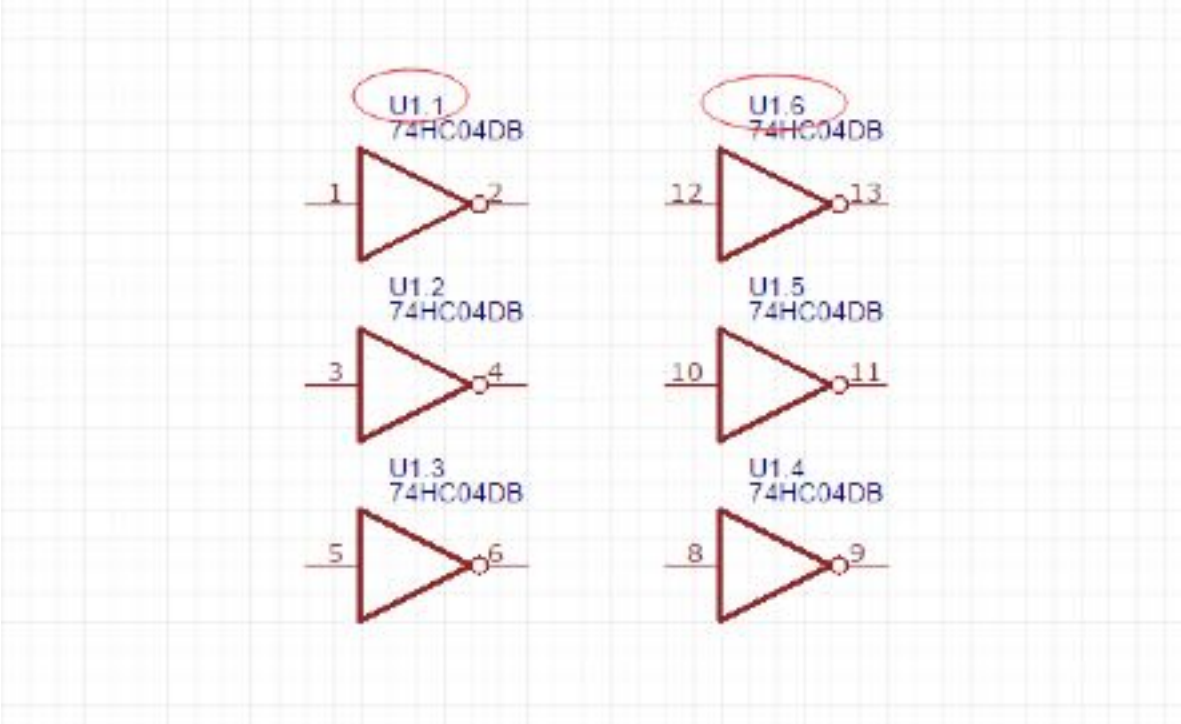
Multi-part Components

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.

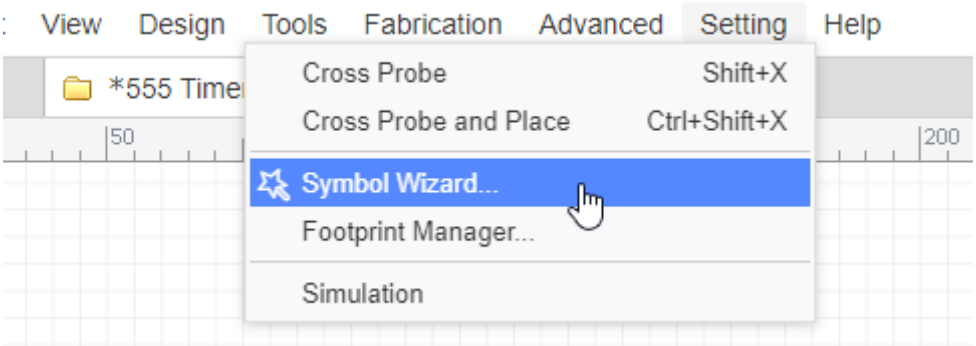
How to create multi-part(subpart) please refer Create Symbol

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.

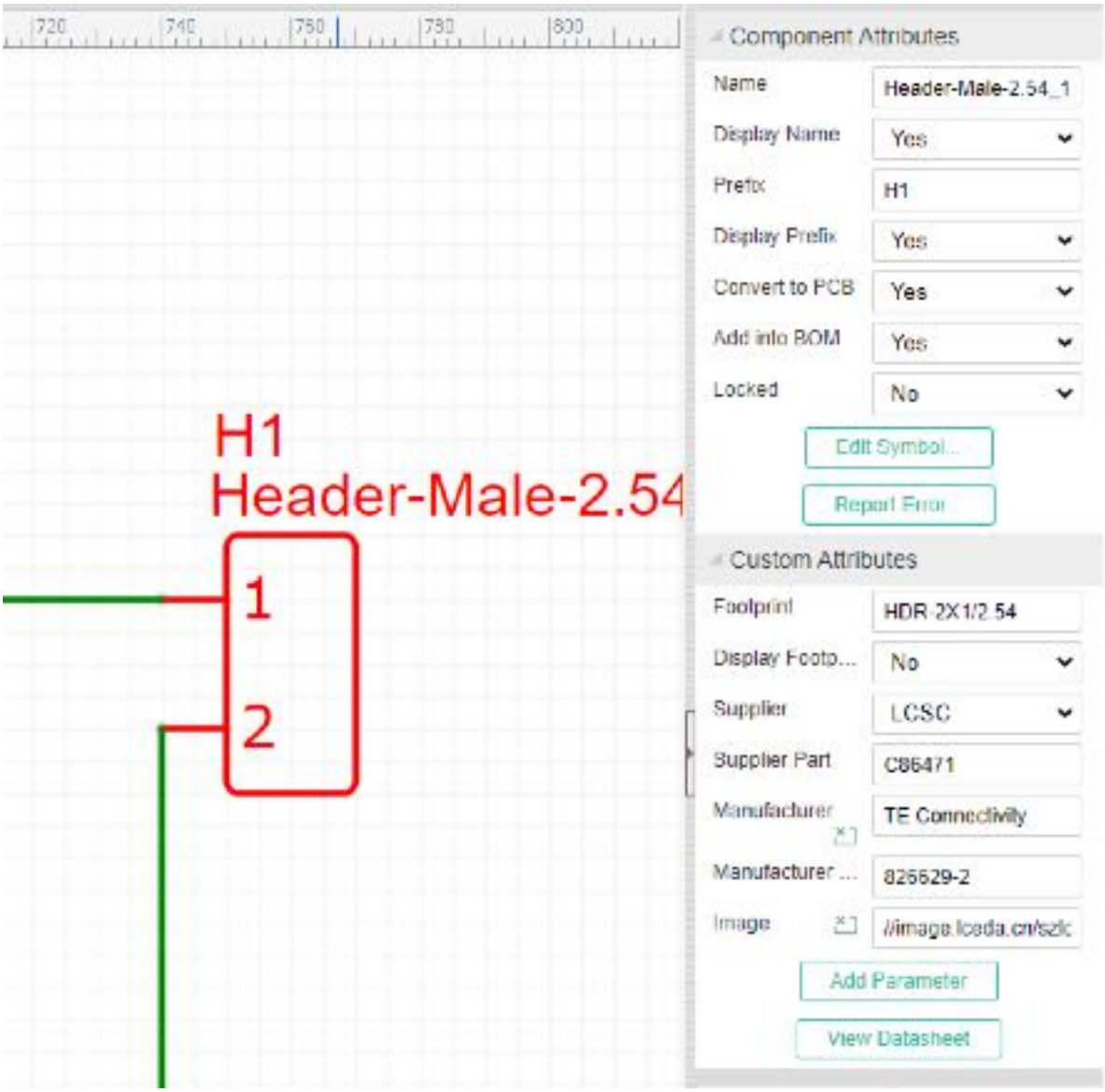


The professional function please refer at Schematic Symbol Wizard

Component Attributes

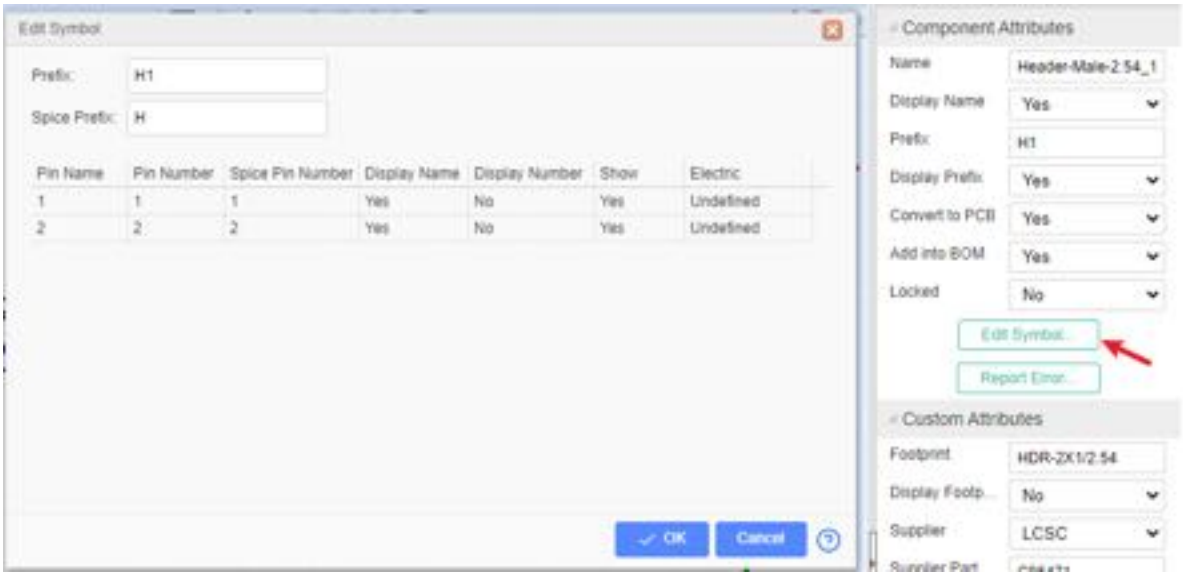
Component Attributes

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



1.Component Attributes:

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



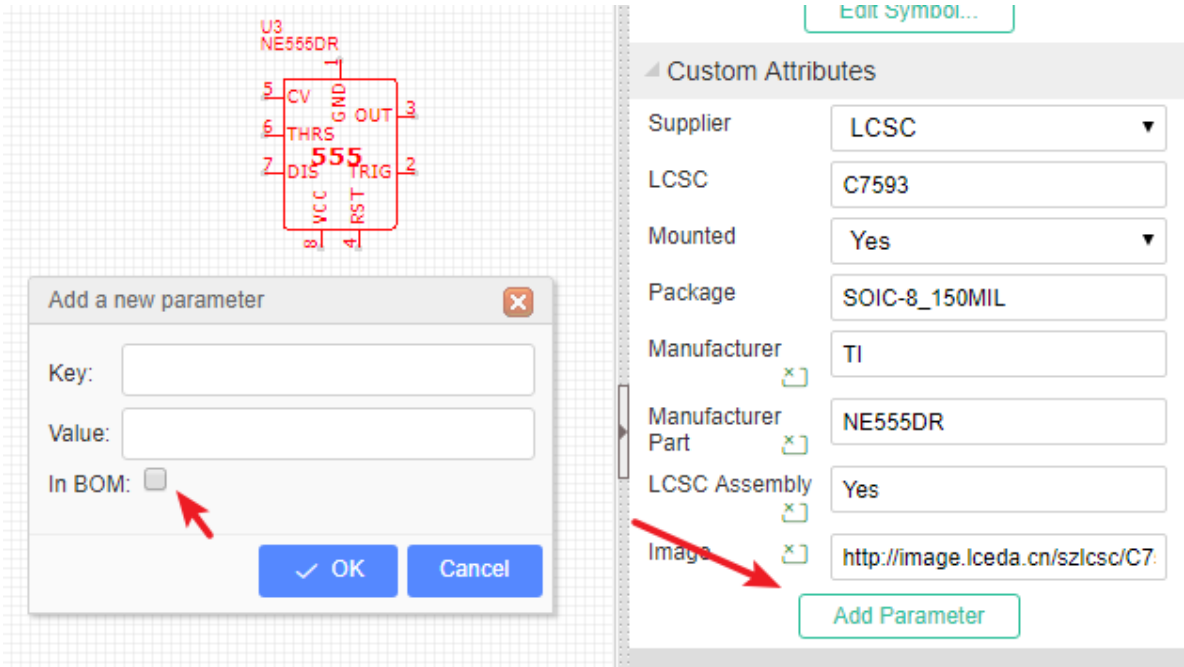
If the component's property "Convert to PCB" is set as "No", it will not appear at footprint manager.

2.Custom Attributes:

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

Define BOM Parameters

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

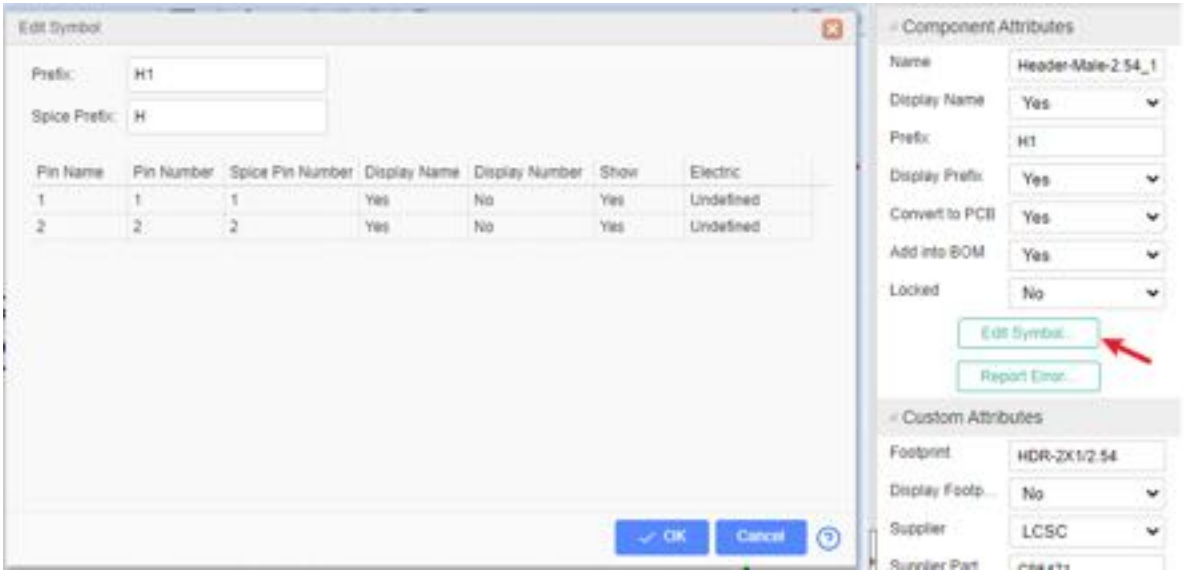


Modify Symbol Pinmap Information

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

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

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



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

Prefixes and Pin Numbers

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

PCB Prefix and Spice Prefix

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

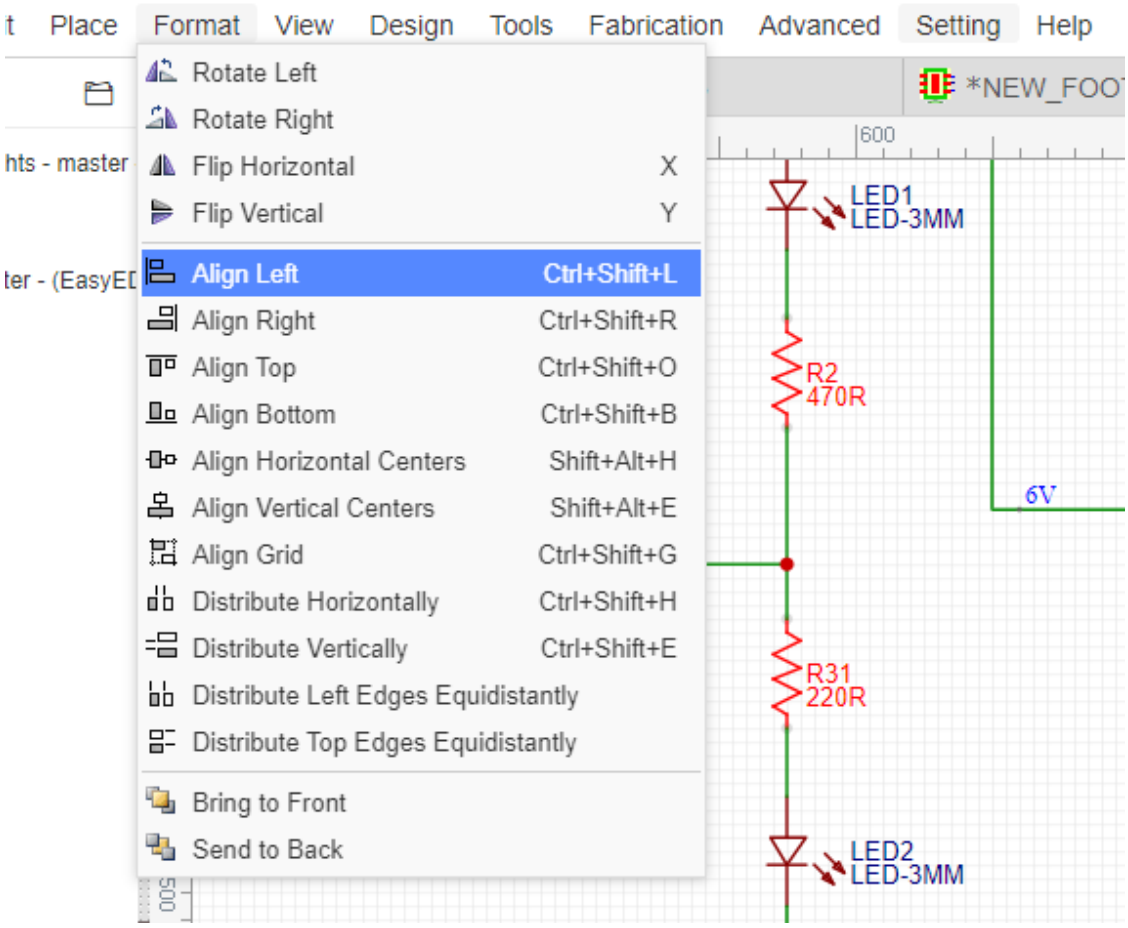
Component Adjust

Adjusting Components

About adjusting components you can:

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

4. Align the components:



Rotating the Prefix and Value (Name) of components

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

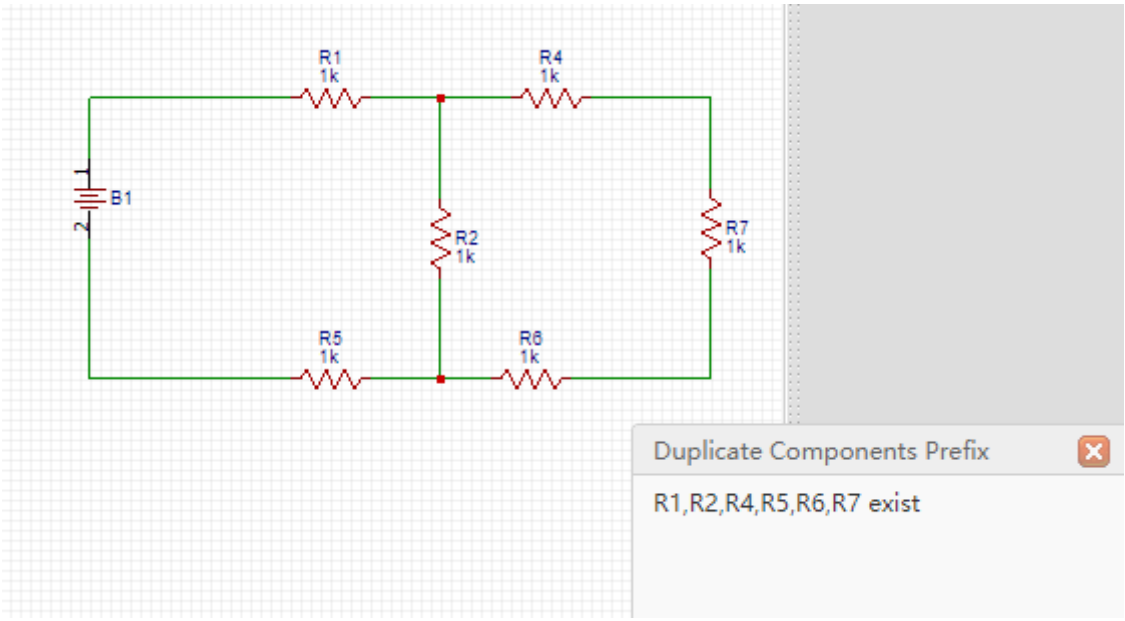
Components Prefixes

Prefix Start

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

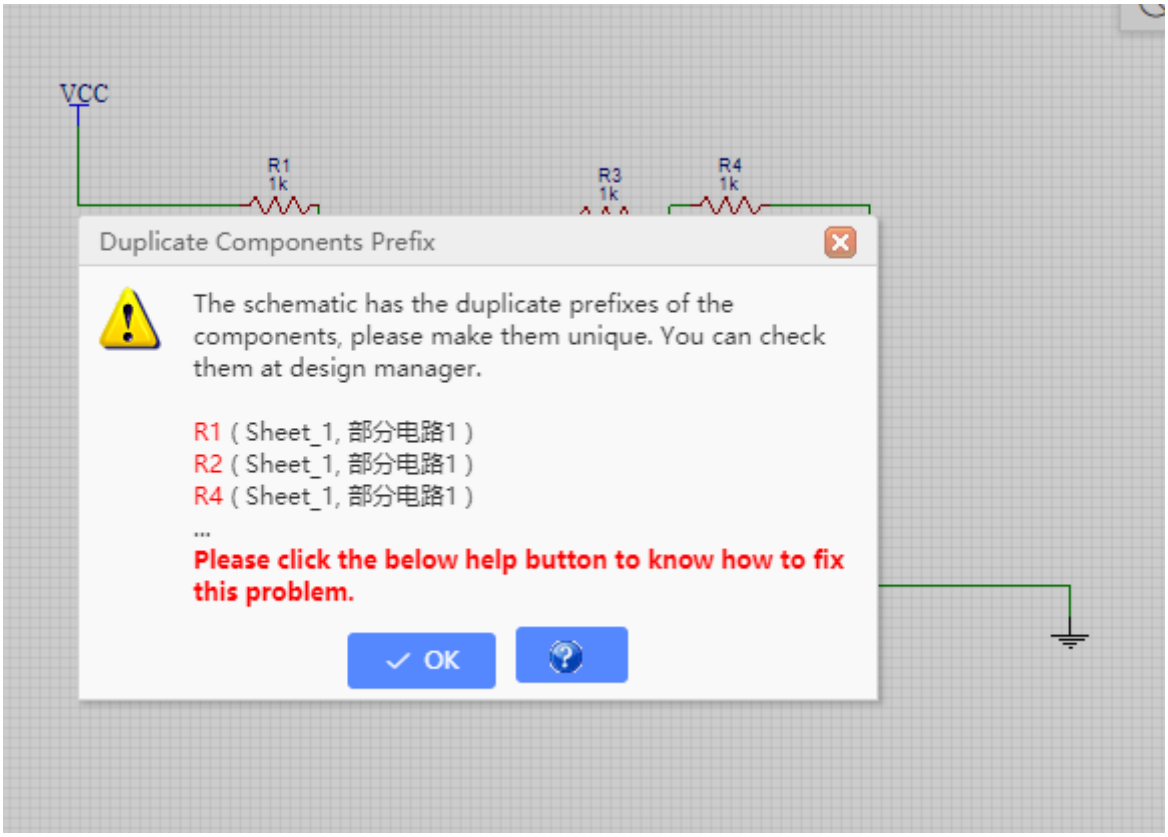
Prefix Conflict Error

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



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

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

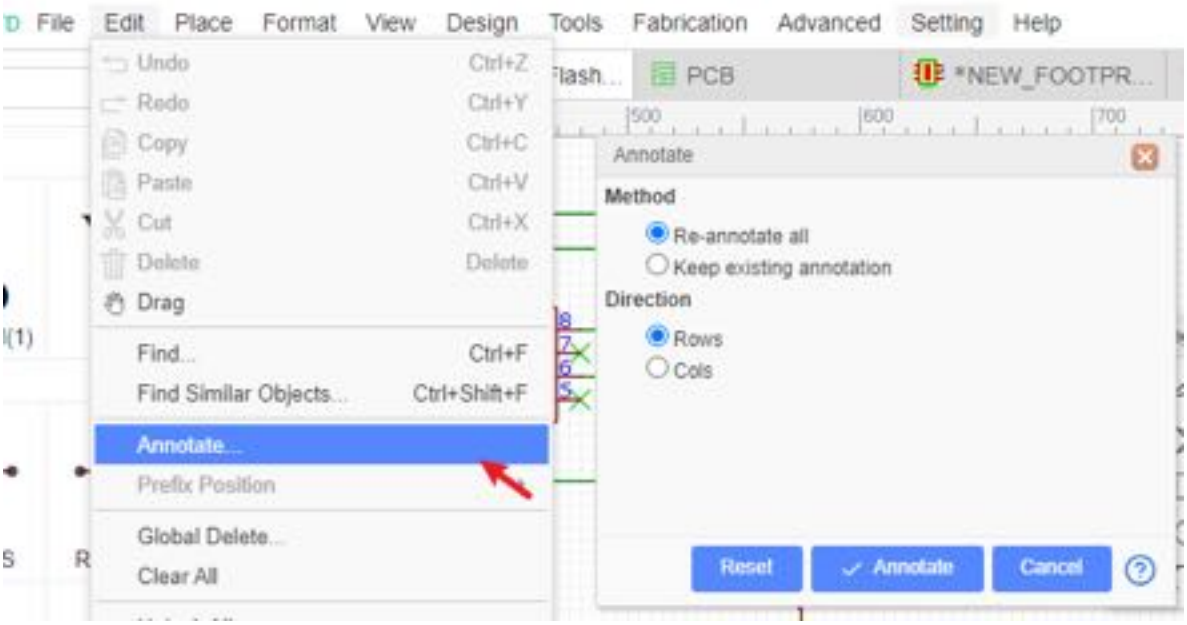


In the above image, you can find the two identical copies of the same schematic, which when you Convert Project to PCB, EasyEDA will try to merge into a single schematic, so every item will have 2 copies.
To fix this, you just have to create a backup project and remove or better still save backup copies of your schematics to that project.

Annotate

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

Via: **Top Menu > Edit > Annotate**



Various Annotate possibilities are available:

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

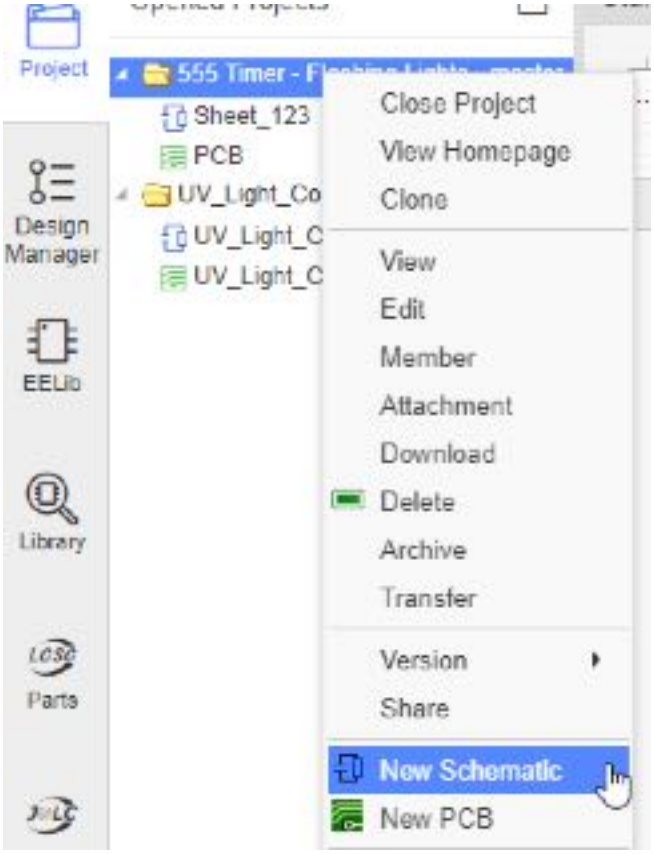
Note:

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

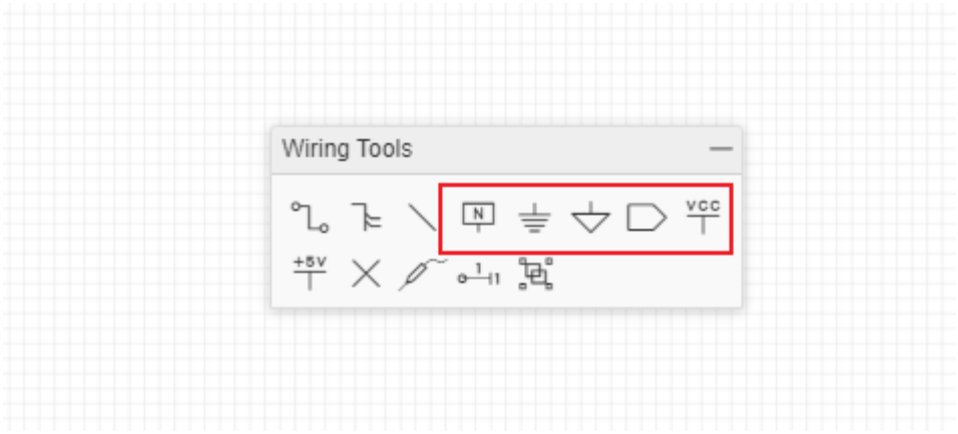
Multi-Sheet

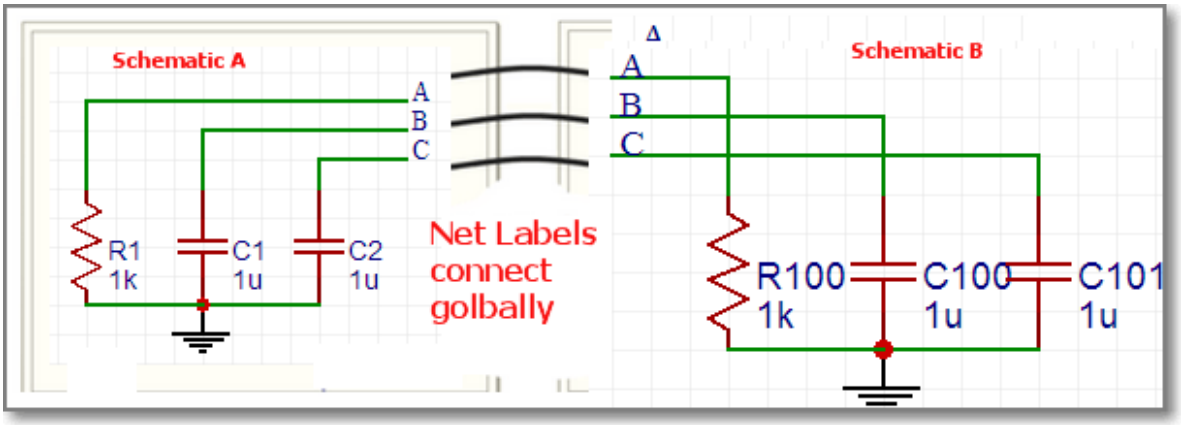
EasyEDA does not support true hierarchical designs but it does support **multi-sheet designs**.

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

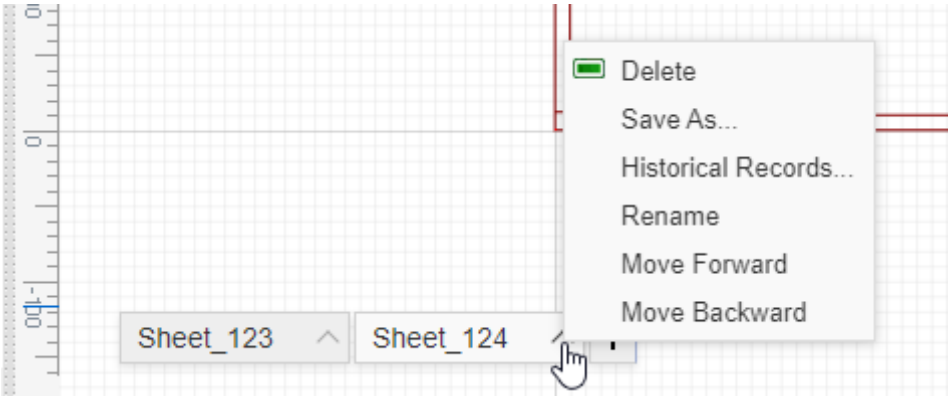


Multi-sheet designs(equivalent to a circuit spread over several pieces of paper), all schematics under the same project will be merged into one when be converted to PCB connecting in **Netlabel**, **Netflag**, **Netport**.

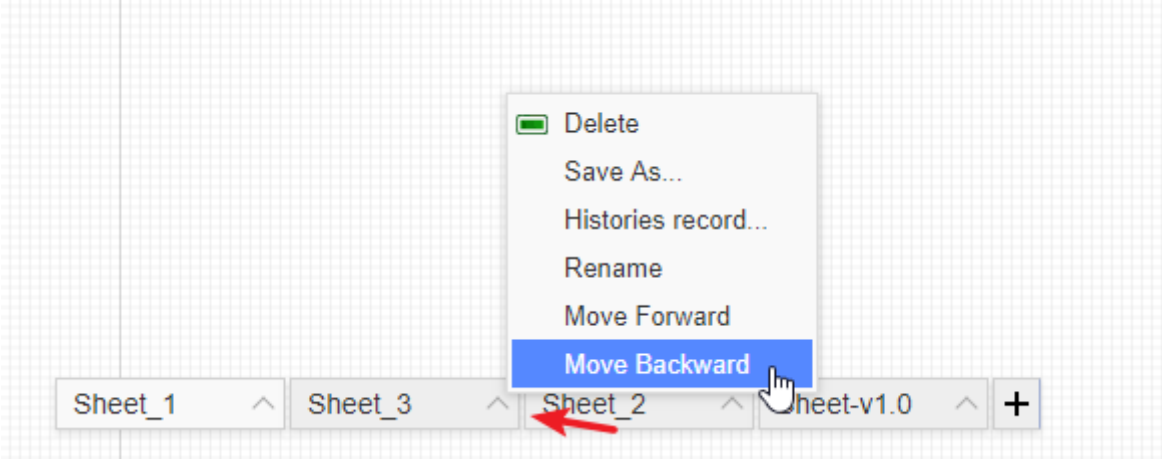




You can click the Sheet tabs on the left-down corner to switch the Sheets, and right-click the sheet tab you can "Save as", check "Histories record", "Move Forward/Backward","Rename" and "Delete" the sheet.



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

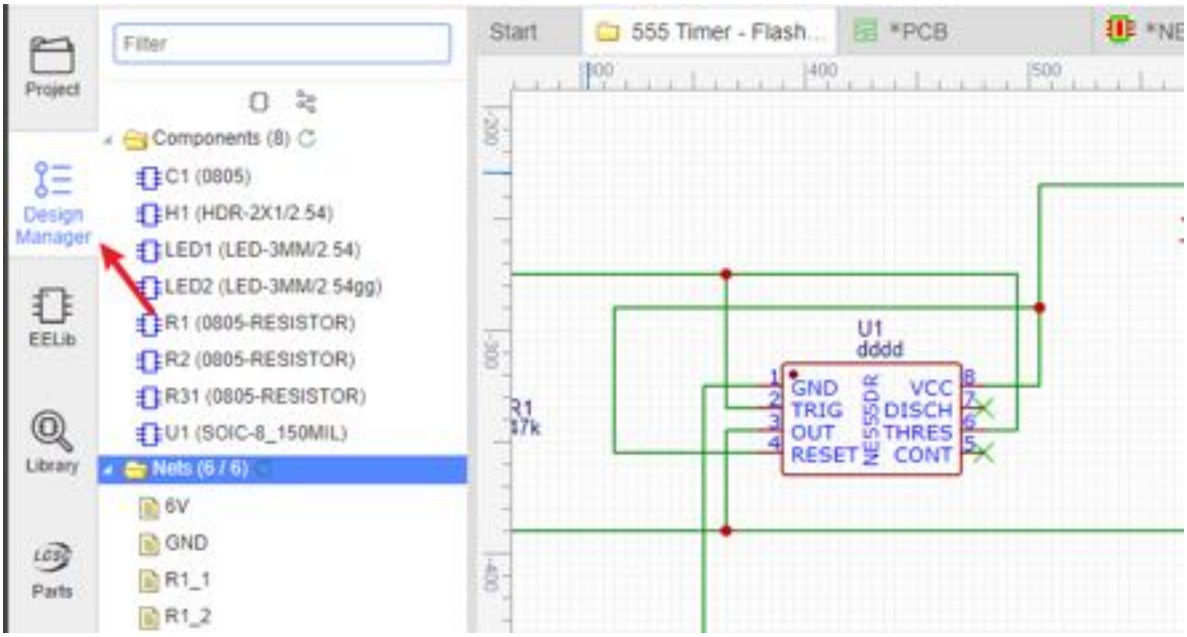


Note:
EasyEDA support global unique prefixes, when you place components in different sheet, the editor will auto annotate the prefix. If you save as a sheet to another project, please make all of the prefixes unique, if the Sheet A has a R1, and the Sheet B has a R1, then you will get a Prefix Conflict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

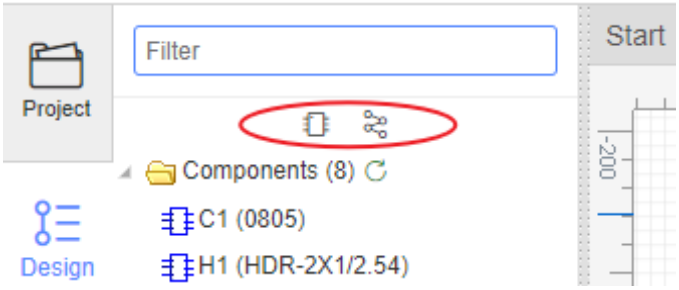
Design Manager

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

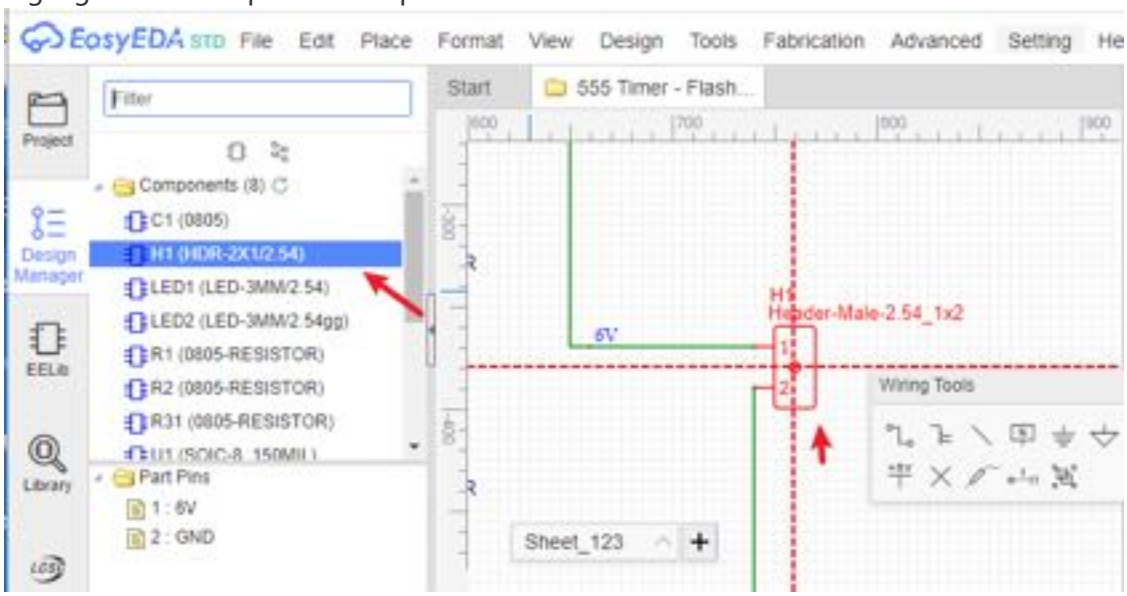
Just press the `CTRL+D` hotkey to open the Design Manager.
or click it via on the left navigation panel:



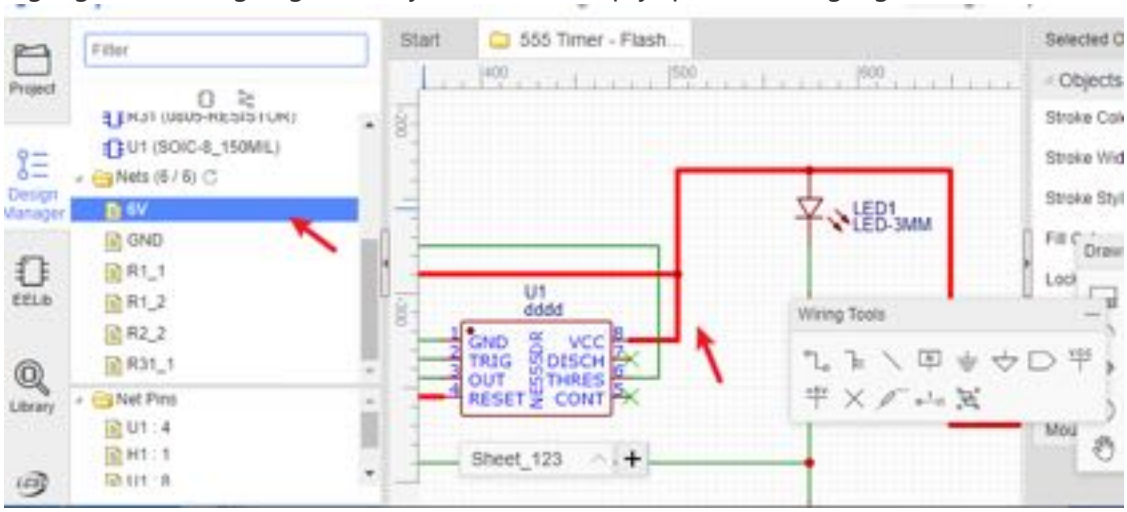
You can click the jump icon to goto the folder quickly.



- **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.

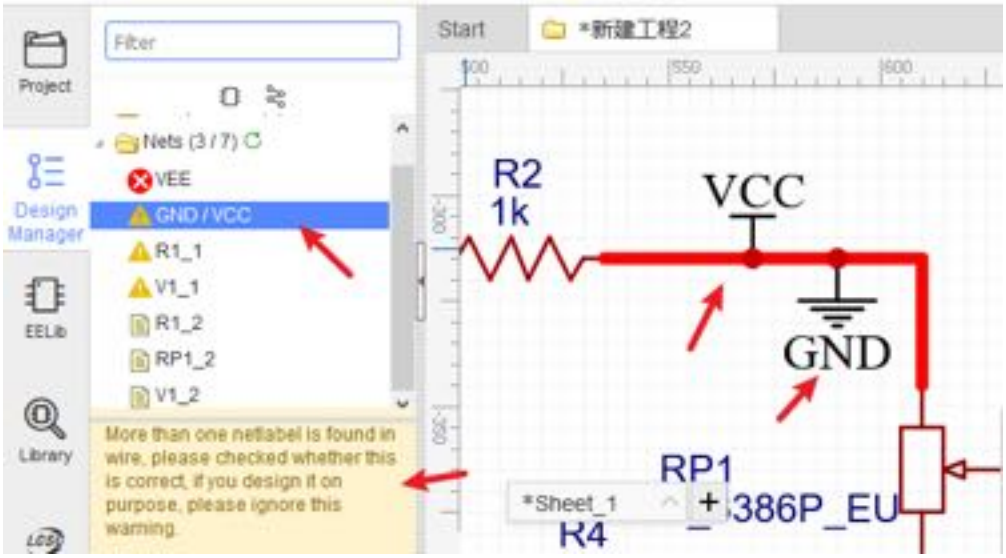


- **Nets**: Lists all the nets in this schematic. A net must connect at least two Pins, or the net name will be marked as a red error. When click the net name, the canvas wire will highlight and being large, when you click the empty space to unhighlight:

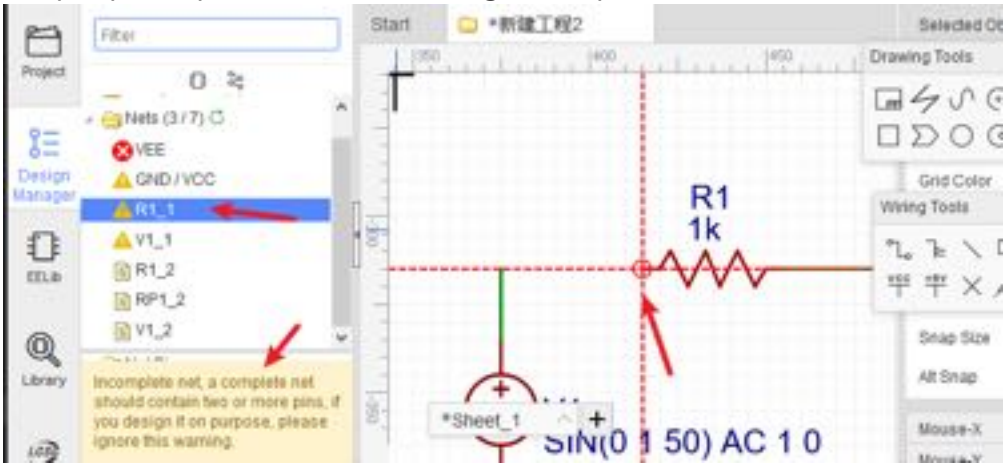


When you click the net name, you will see the tip at the bottom-left corner.

  - 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.

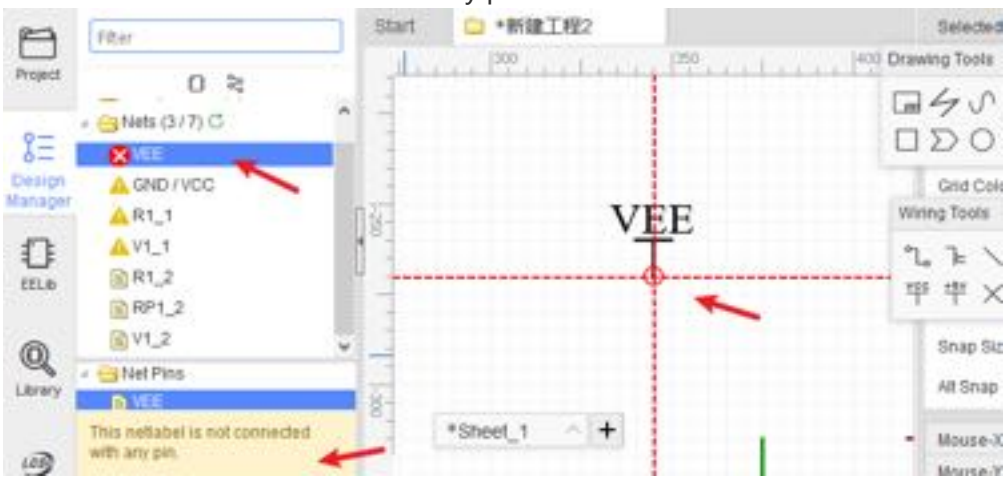


    - The part's pin doesn't place the netlabel, or doesn't connect other pins, or doesn't place No Connect Flag. A completed net must connects two and more pins, so that, you need to modify your net connection. If you don't need to use this pin, please place a No Connect Flag on the pin.

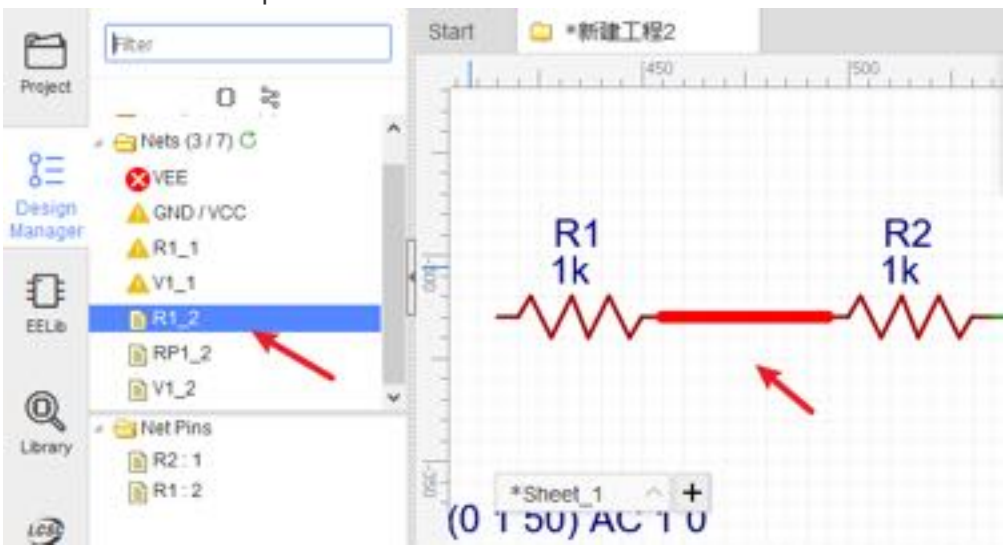


  - Net error: Prompts a red error icon.

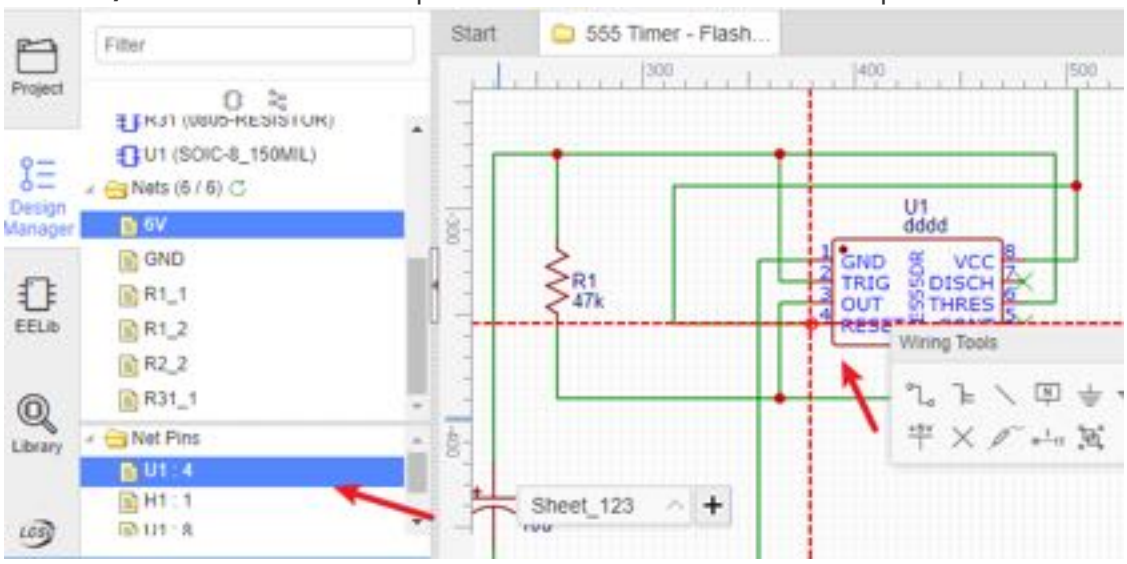
- When Netlabel haven't connected any pins.



- Net good: Prompt text small icon. The completed net should connect two pins and more.
  - When the net is completed.



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



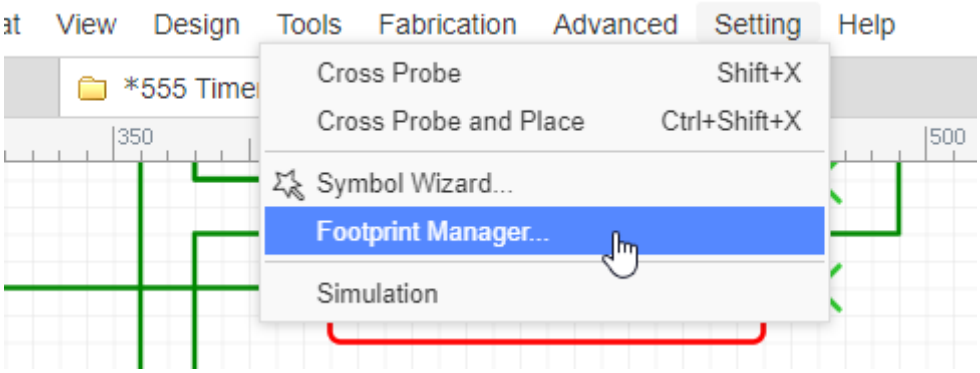
Footprint Manager

Introduction

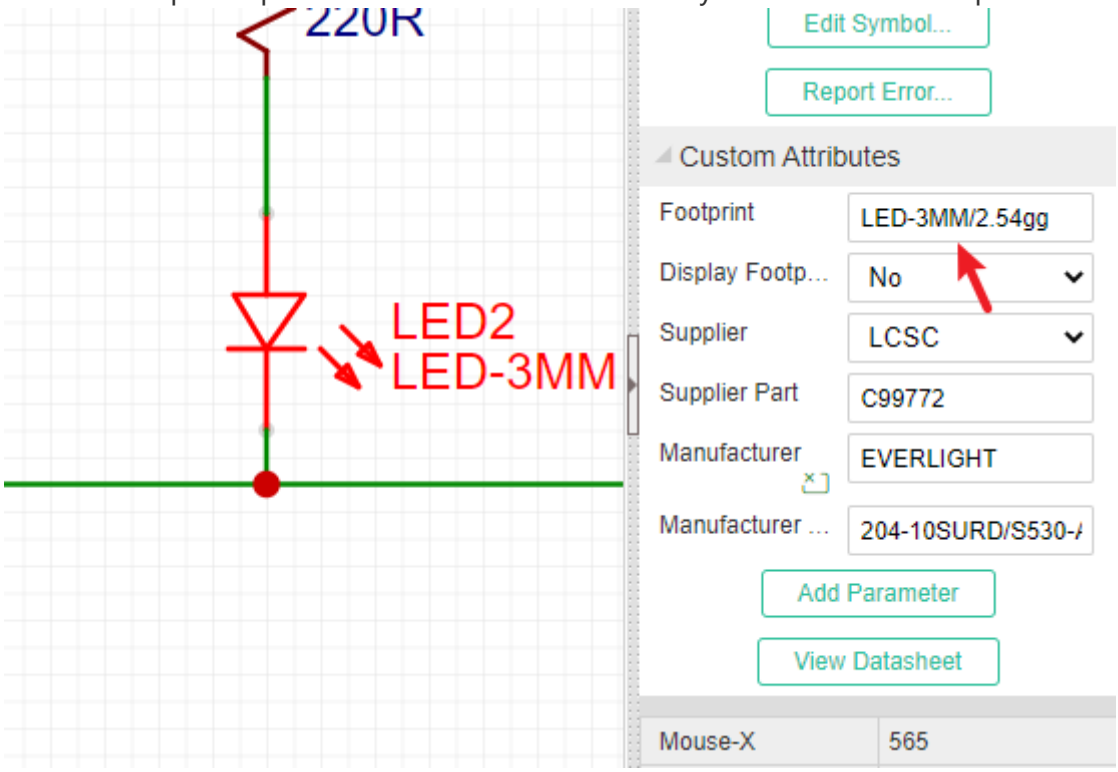
Want to batch modify components? Can't identify the corresponding relationship between component pins and footprint pins? Don't worry, EasyEDA can do this.

There are two ways to open the footprint manager:

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



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



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

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

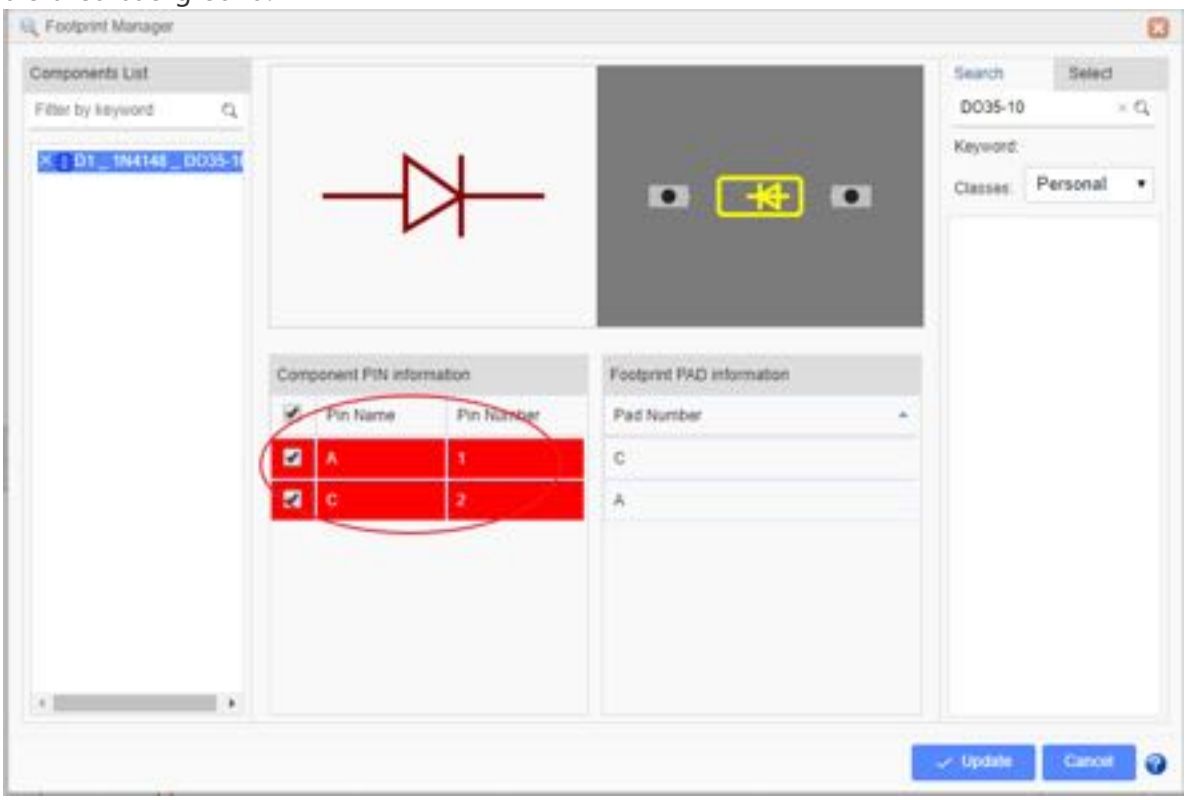
For example, If your part D1 has 2 pins,

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

but you assigned a footprint has 2 pads,

- pad number are `A` and `C`,

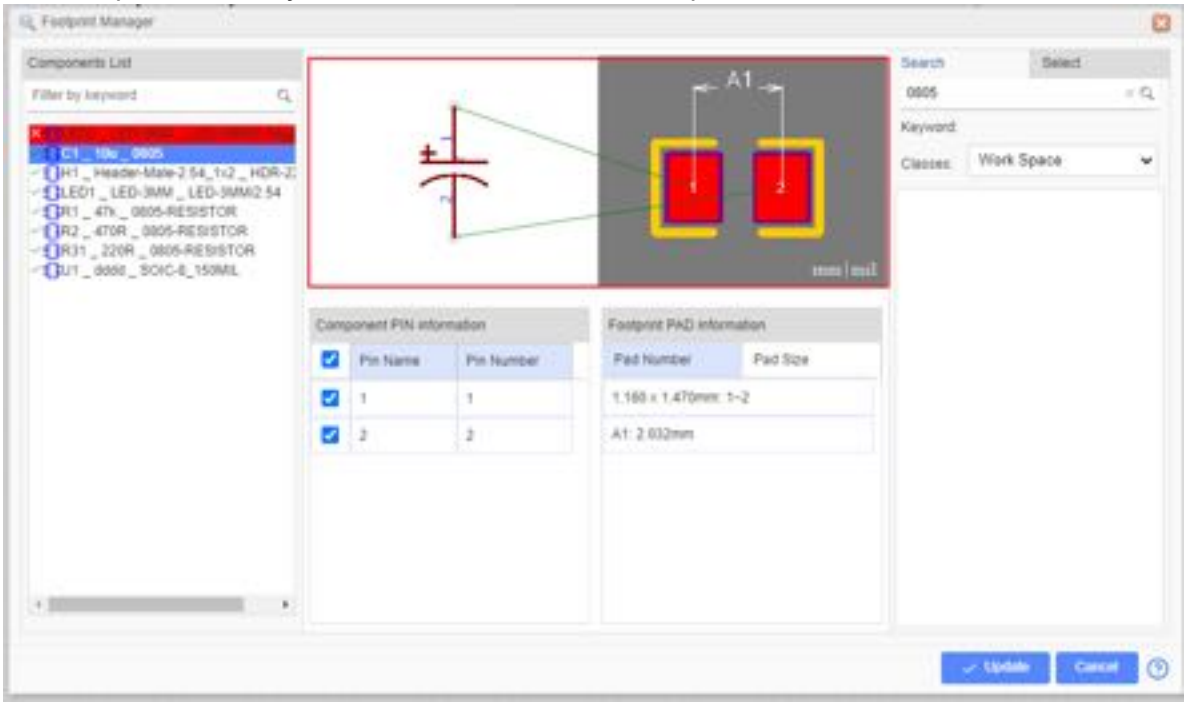
but the part's pin number doesn't match the pad number, so the the footprint manager will alert red background:



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

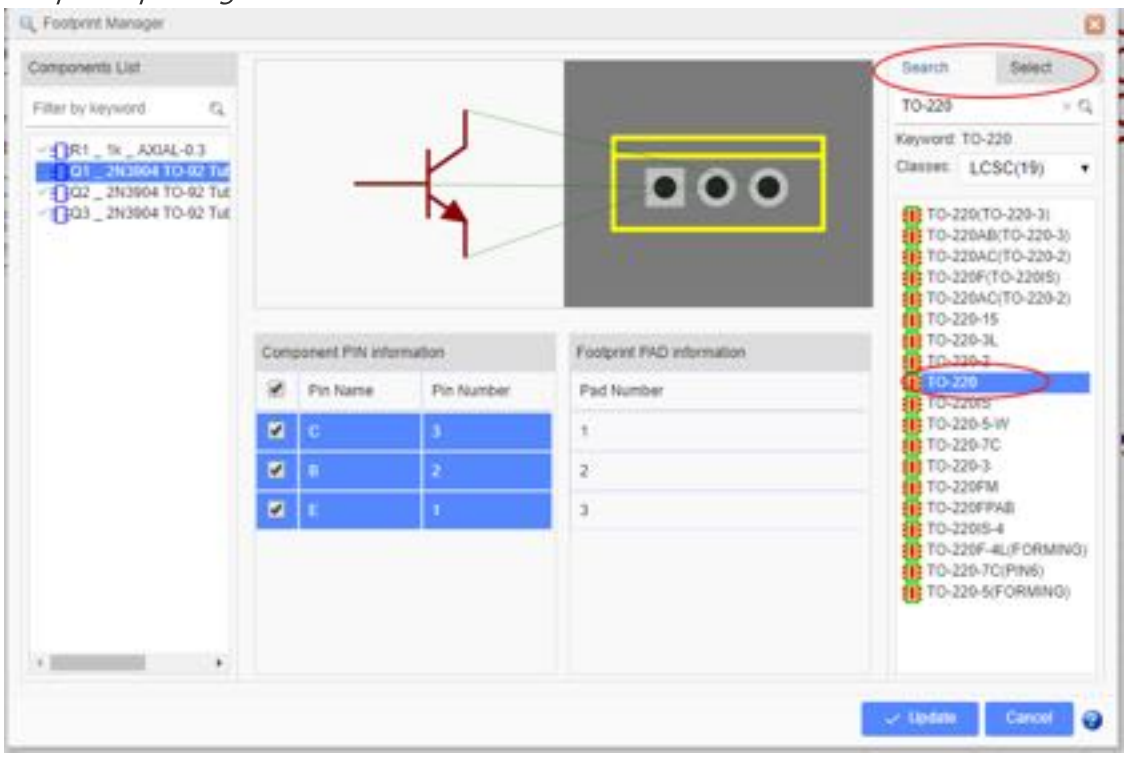
If you want to change the footprint, for example, select a component such as Q1, from **TO-92** TO **TO-220**, you just need to click in the footprint input box. EasyEDA will popup the footprint manager dialog. You can follow the instructions.

- 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.*



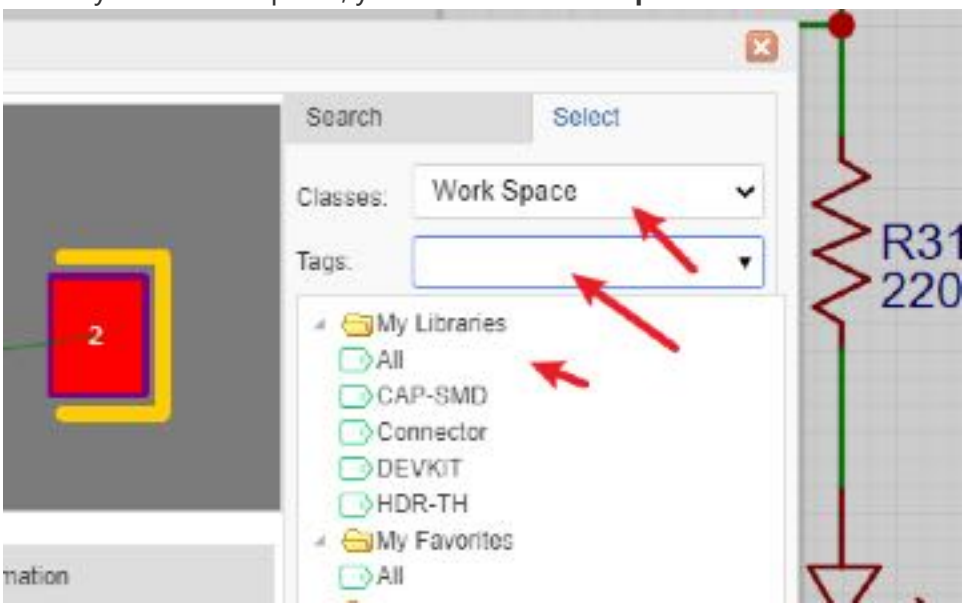
- *When you select a subpart, the others subparts will be selected too, so they will update the footprint together.*
- *If the part's property "Convert to PCB" is set as "No", it will not appear at footprint manager.*

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 commponents 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.



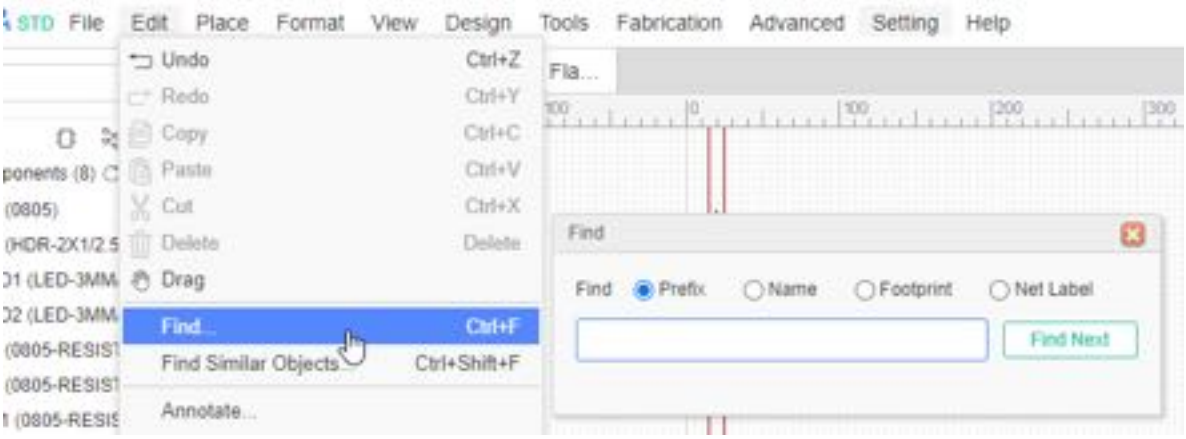
Find Similar Objects

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`)



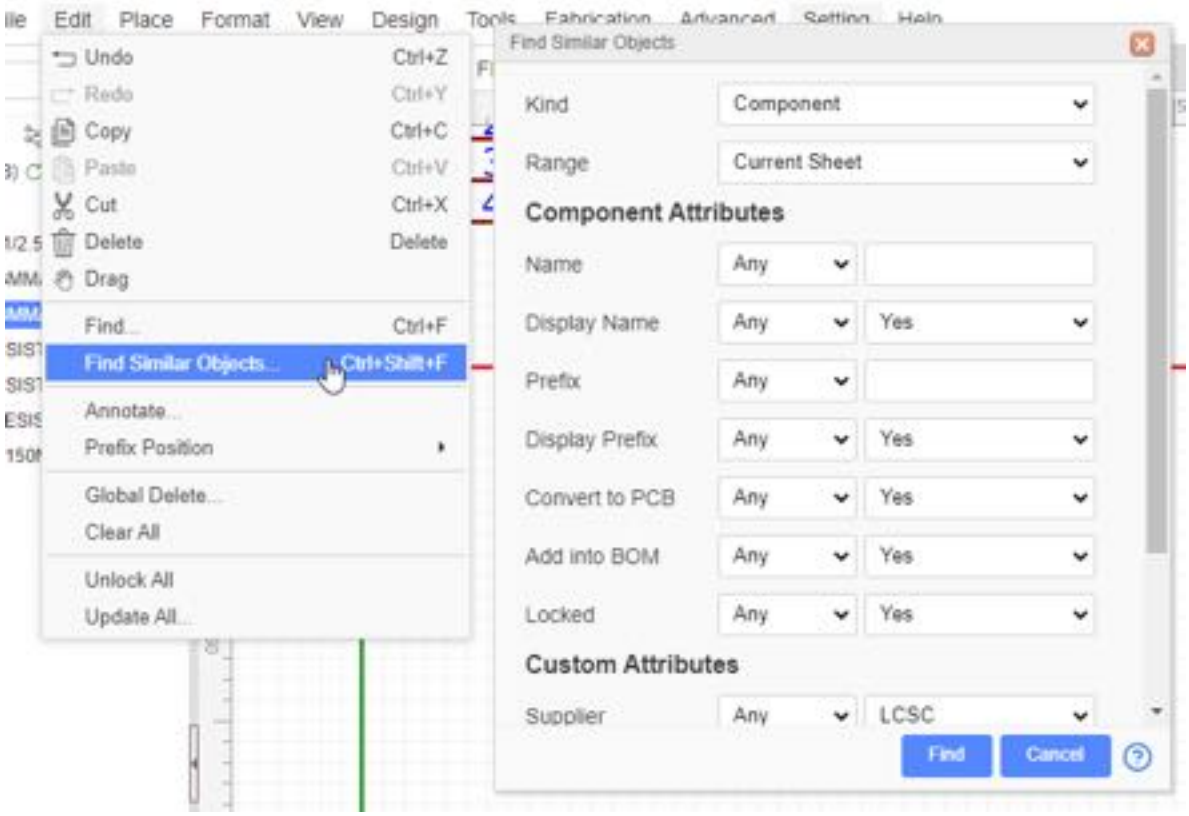
Note: *You have to click OK?in this dialog or use the Enter key.*

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.

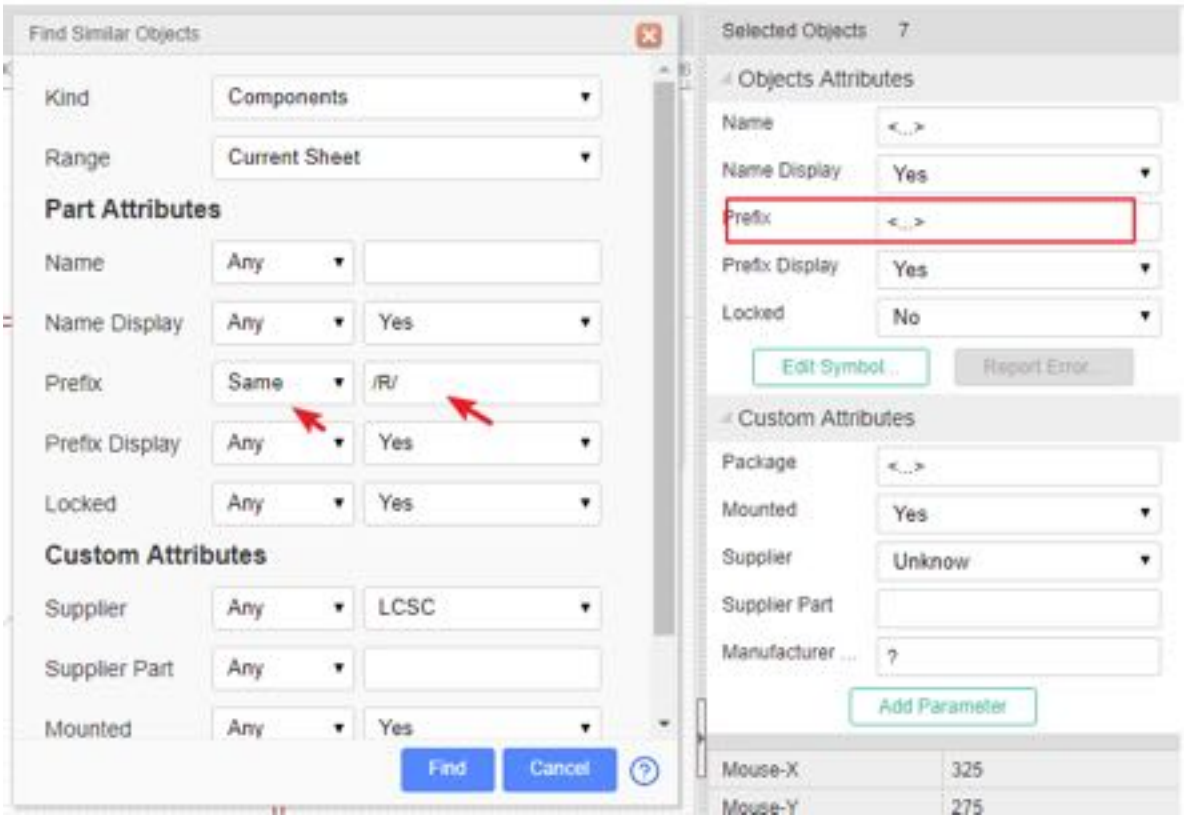
Find Similar Objects

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":



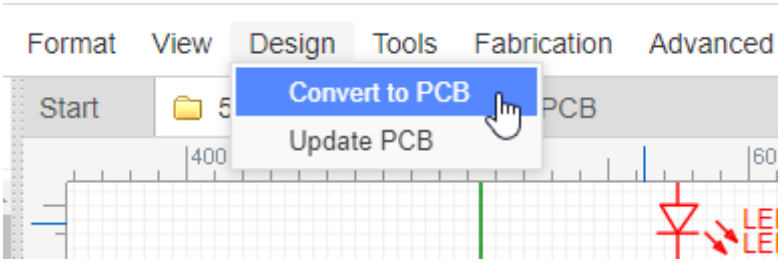
After click the "Find" button, all the siutable objects will be seleted, and the right-hand panel will show all the attributes, the different attributes will show as the `<...>`, you can change the attributes directly, and they will apply to all selected objects.

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

Convert Schematics to PCB

Convert to PCB

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

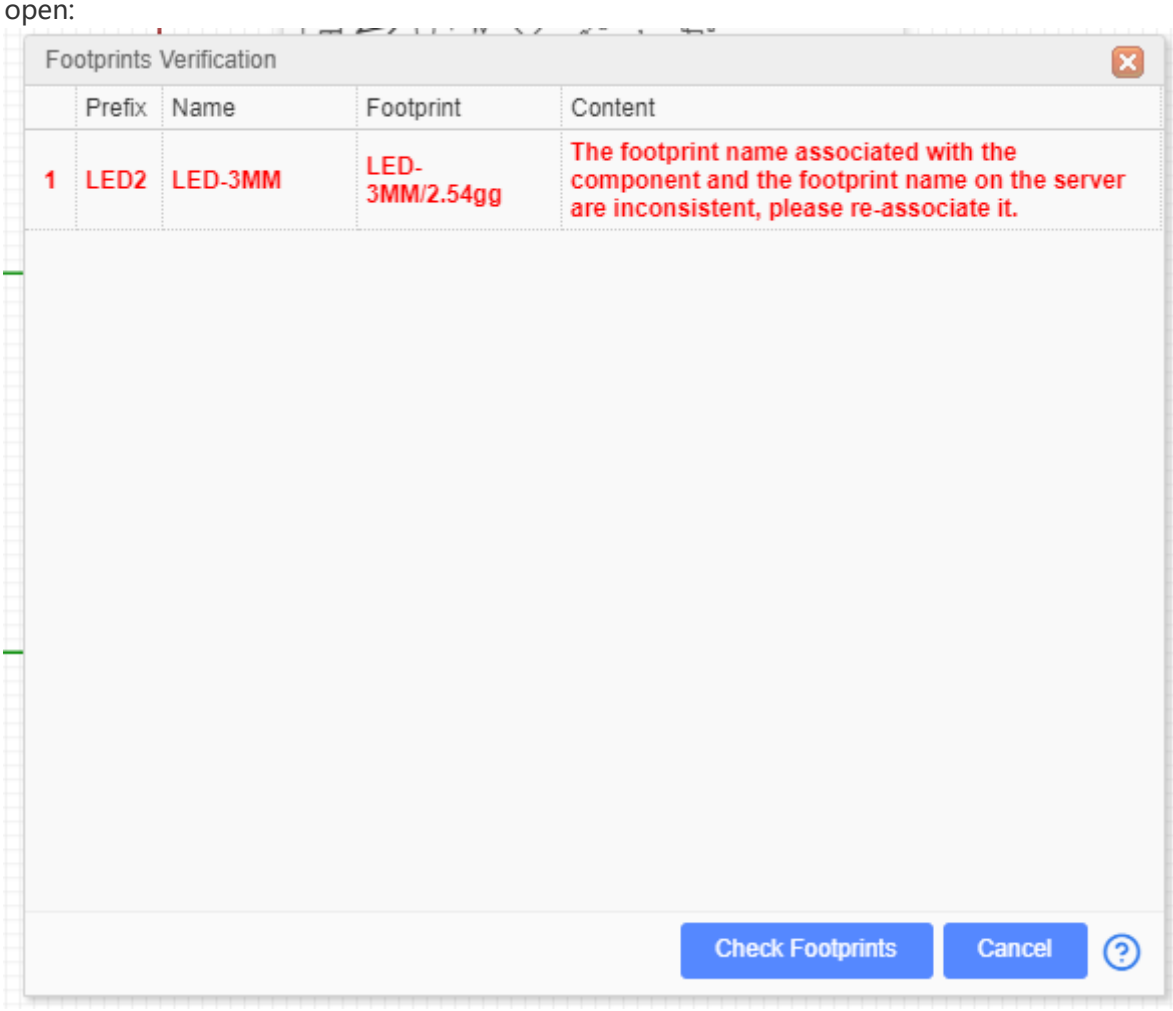


Note:

- *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:

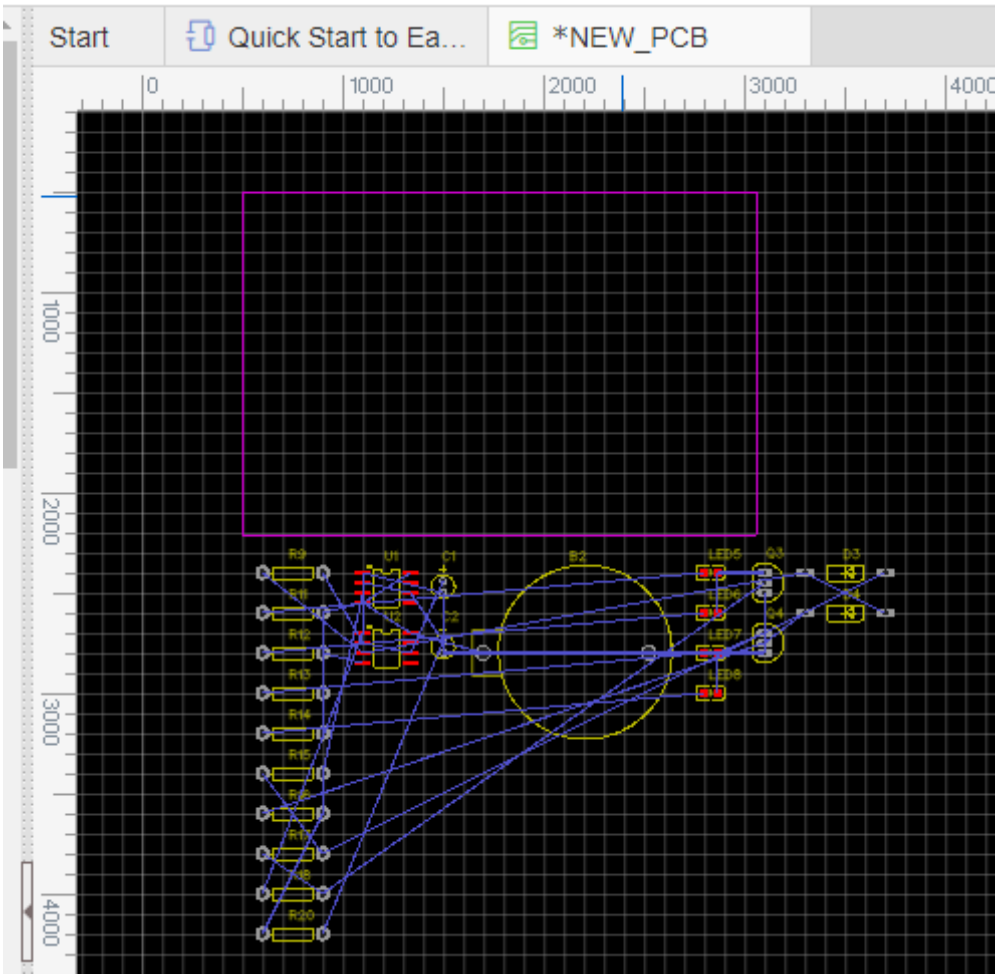


The row in red indicates that EasyEDA can't find a PCB footprint matching the footprint that the schematic symbol is calling for.

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.

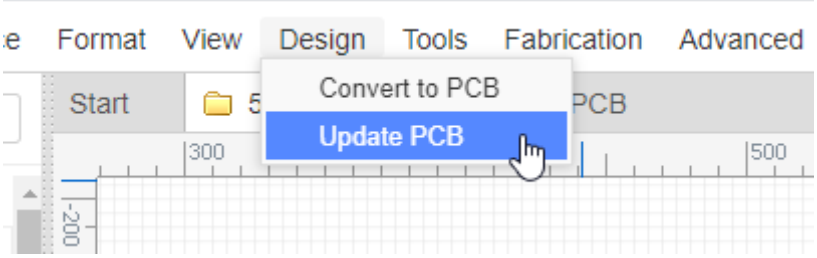
Invalid footprint

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

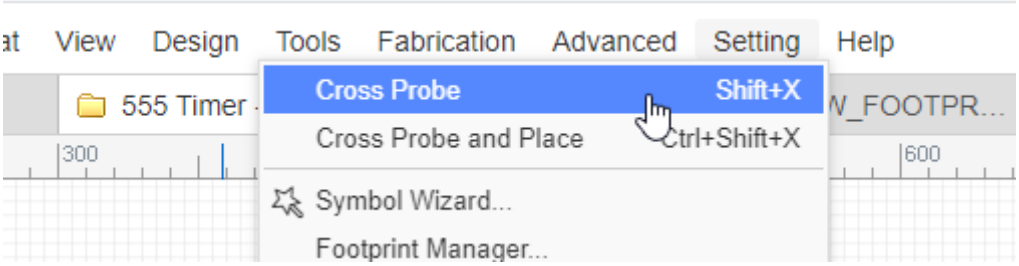
Converting a schematic to PCB can be done using the `Convert to PCB...`, but if you do modifications to the schematic, by using the `Update PCB` button you can immediately be passed forward to update the selected PCB without having the PCB editor window already open or without creating a new PCB file.



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

Cross Probe

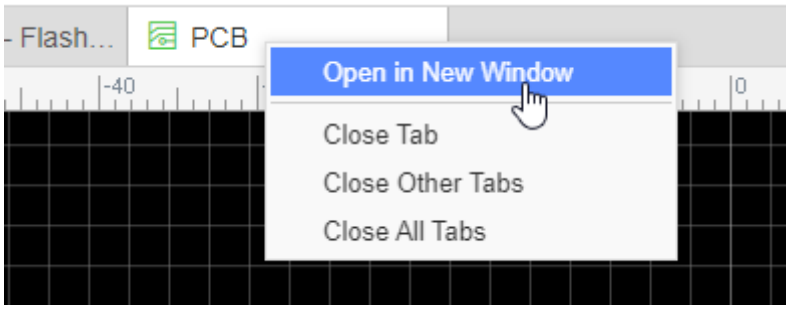
This tool is used to cross probe from chosen objects on the current schematic to its corresponding counterparts in the PCB, or from PCB Footprints to corresponding counterparts in the schematic.



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.

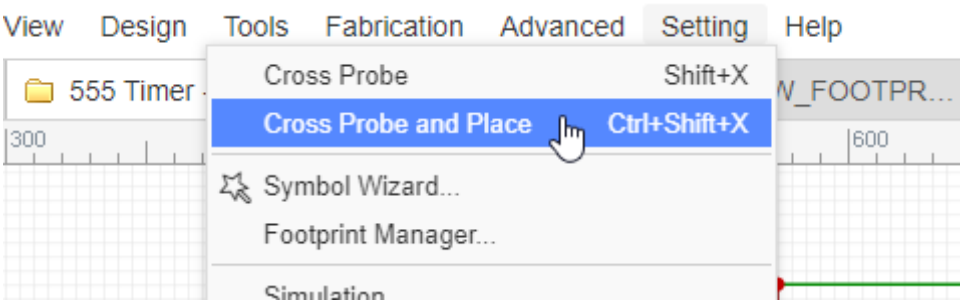
Note:

- *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".

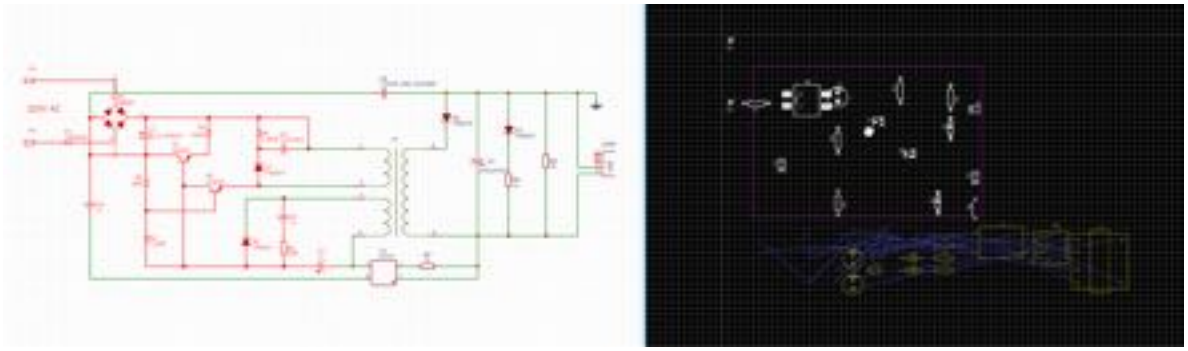
Top Menu> Tools > Cross Probe And Place



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:

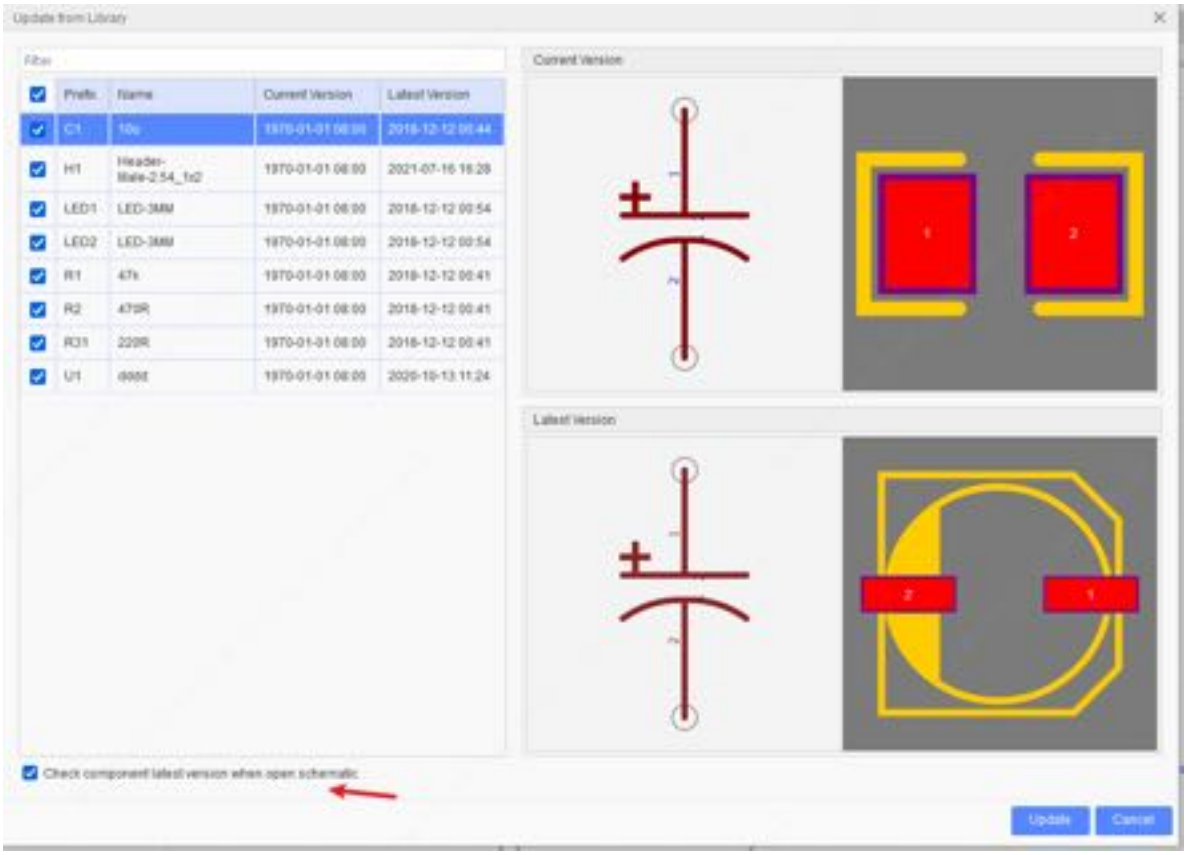
- *You need to open PCB first before using this function in the schematic*

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.



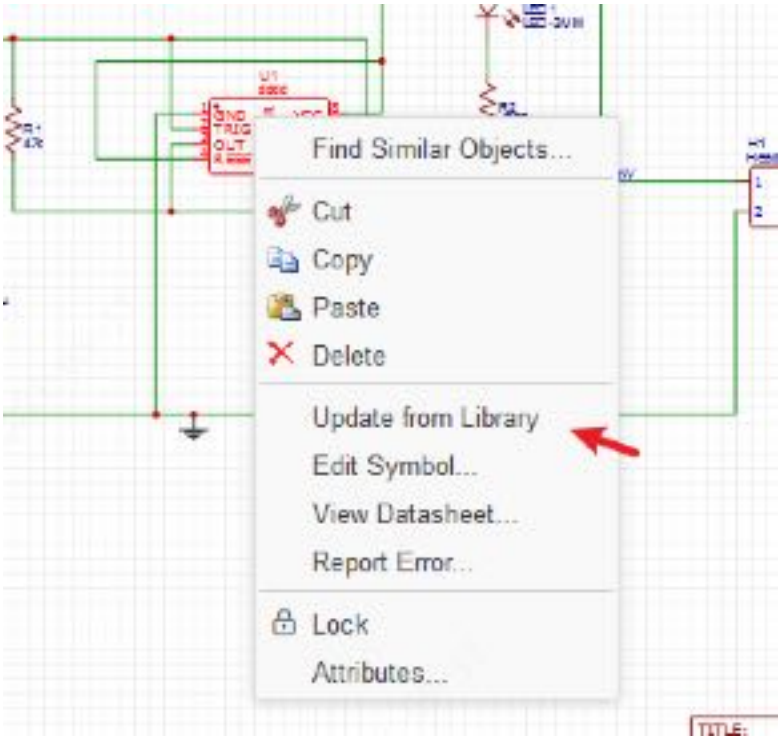
You can setting check the component latest version when open the schematic at bottom option.

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 compoennt, you can right-click it and update

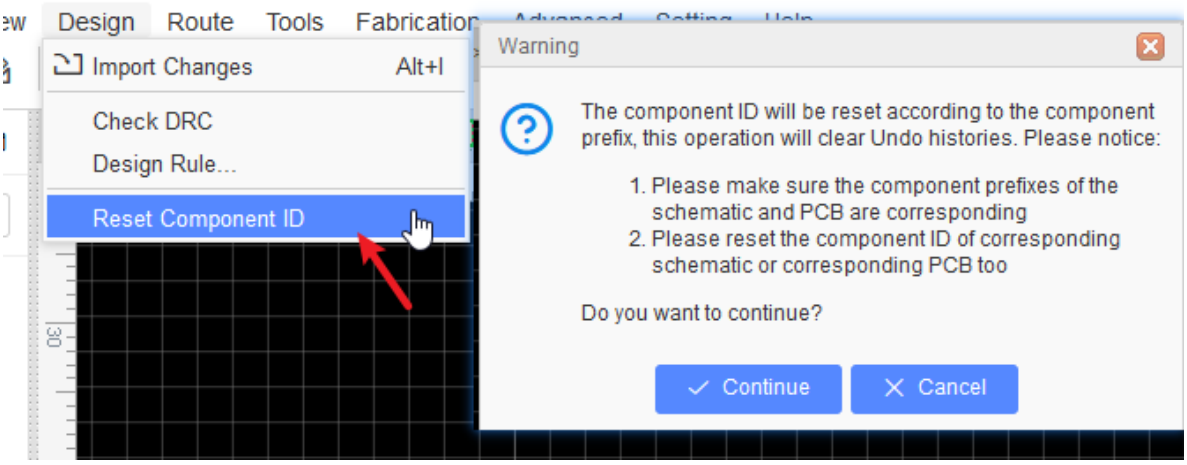


Reset Component ID

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 deleted and add new component 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:

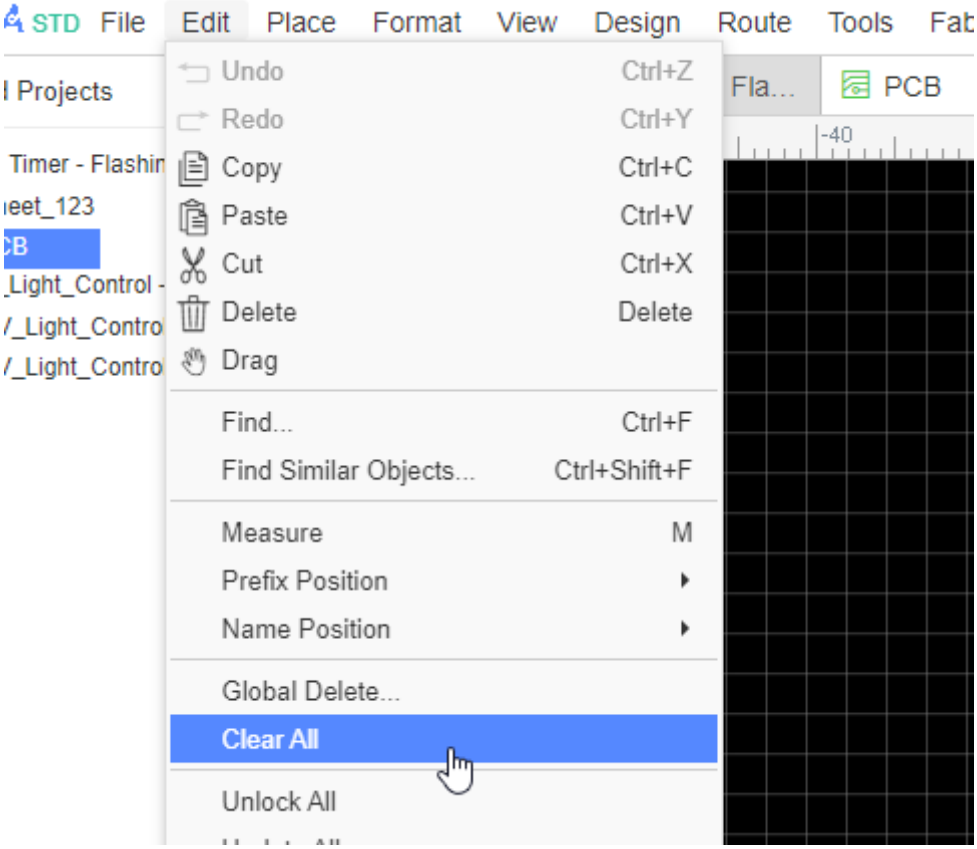
- 1. Make sure that the schematic diagram and the component prefix of the PCB match. The reset ID is reset based on the component prefix.

- 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.

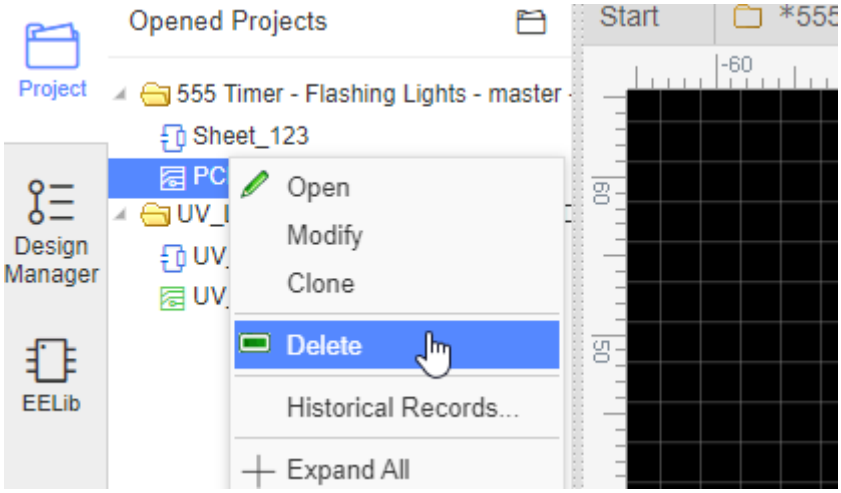
Global Delete

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

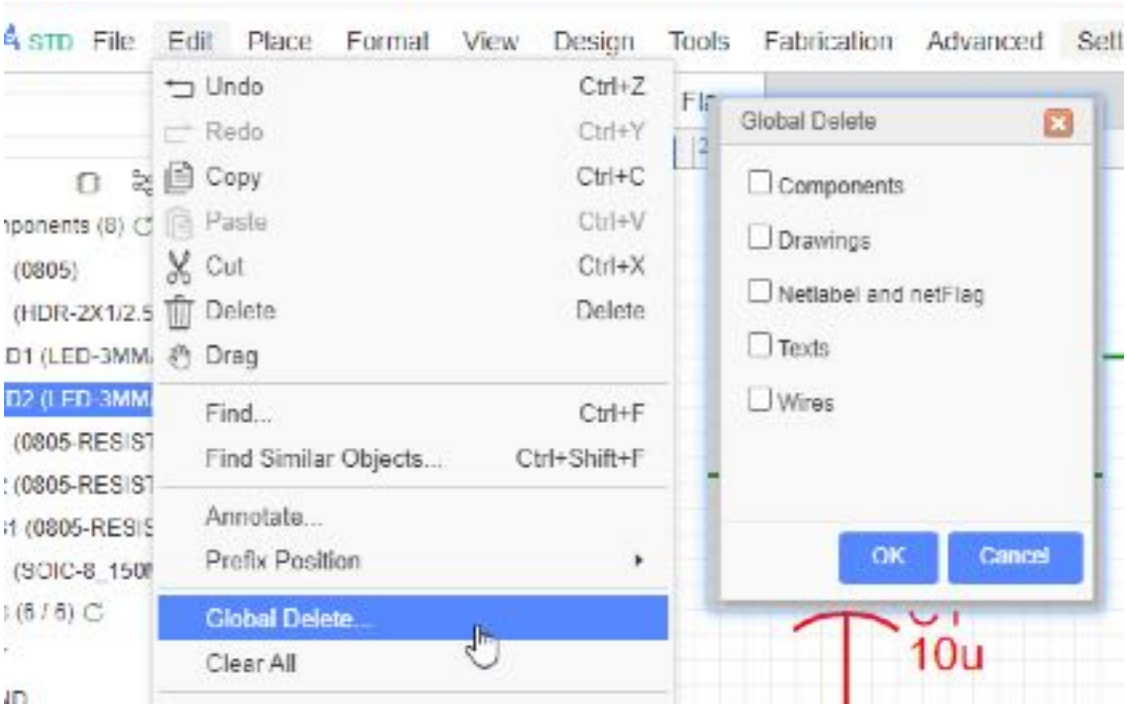
- **Top Menu > Edit > Clear All**, or CTRL + A select all and then press Delete key.



- Delete the document and create a new one.



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

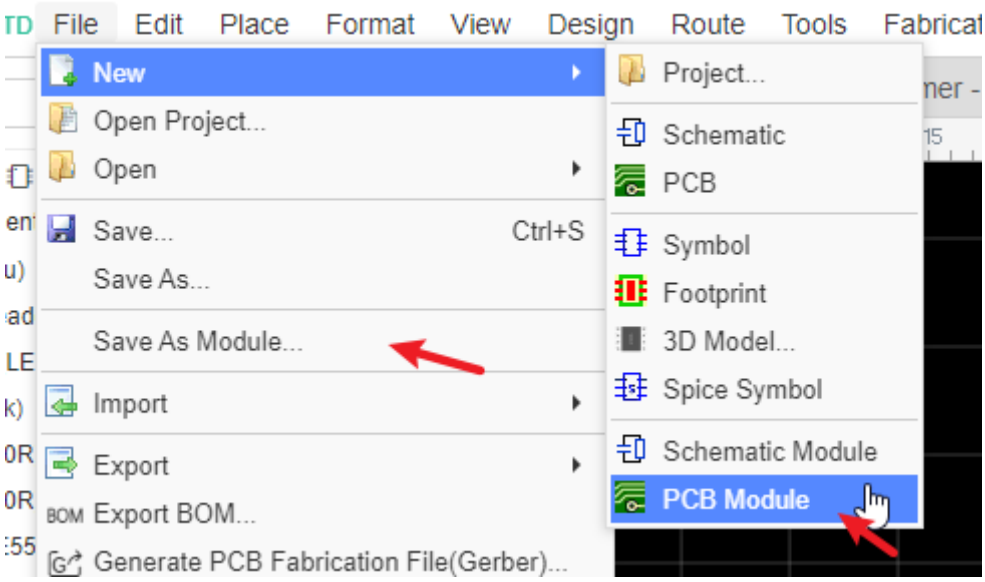


Schematic Modules

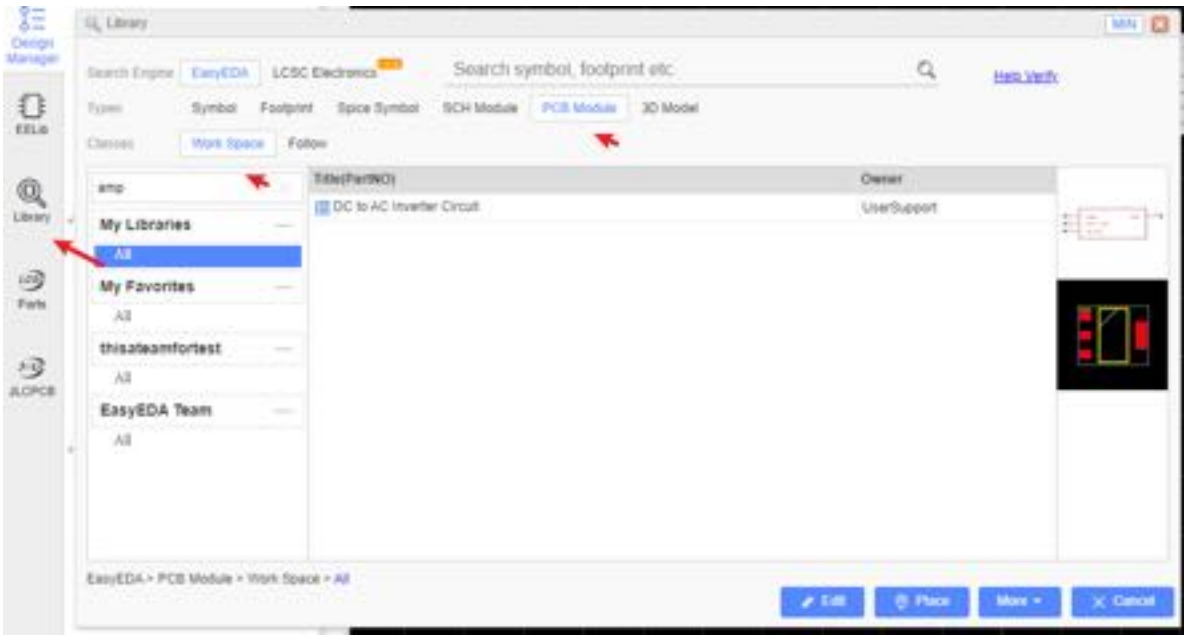
Copying codes is an easy job for coders, now copying and reusing a schematic or PCB is easy. Take a power supply unit for example, you can save this unit as a schematic module.
Via **File > Save as Module**:
EasyEDA support create the PCB modules, it seems schematic module.

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**

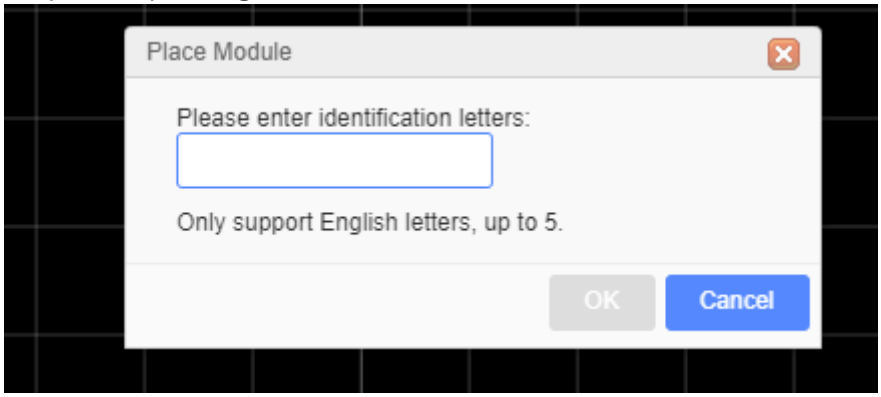


How to use

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 multipe chanel placing.

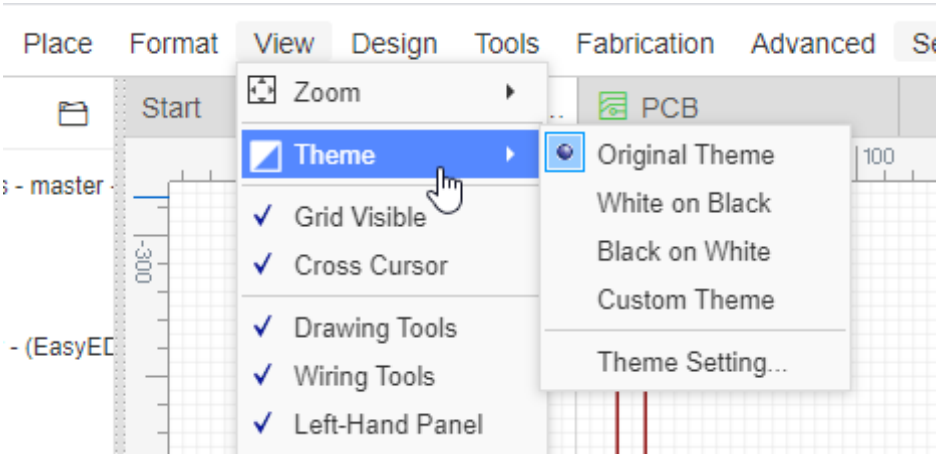
Notice:

- Module composes by tracks and components, it doesn't same as symbol binding footprint, the schematic module can not binding PCB module, after placing, the module will be separated by many objects, only the symbol and footprint can be corresponding via component ID, that is why you need to make the identication 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 Definded: 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.

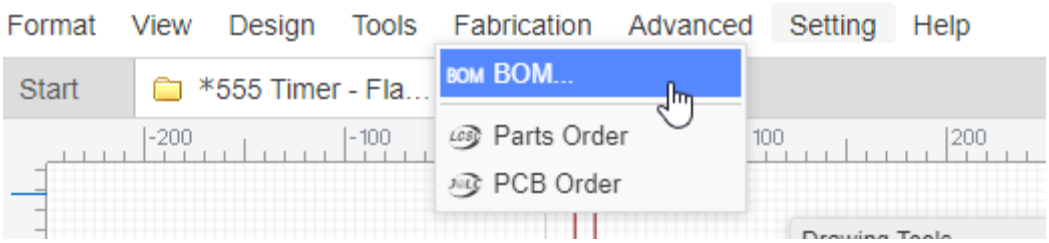
My theme Settings: You can apply "My theme" on: 1. Creating New Schematic, 2.Opening Existed Schematic.

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.

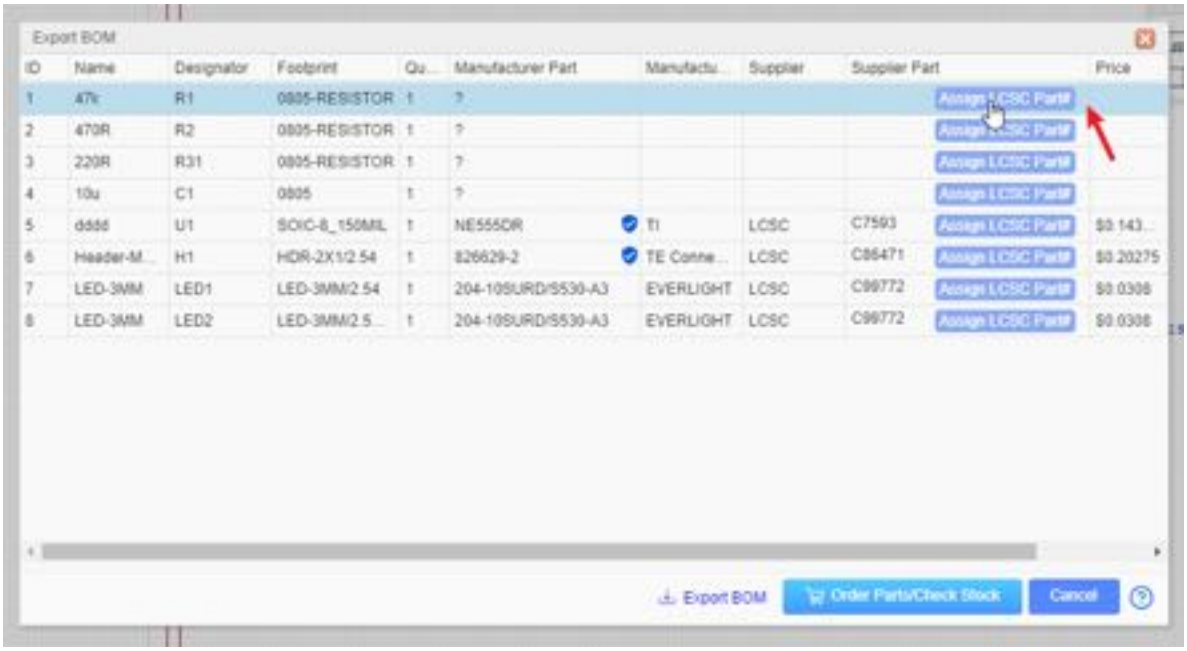
Export BOM

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

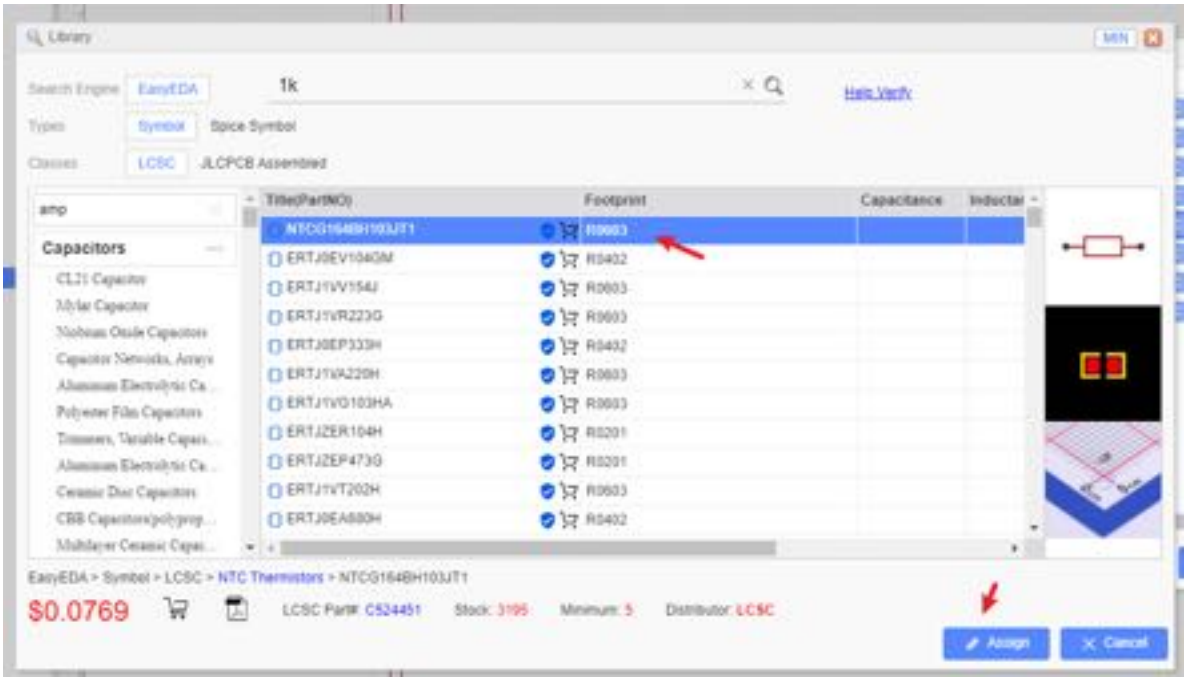


After clicking the BOM export option, the dialog below will open.

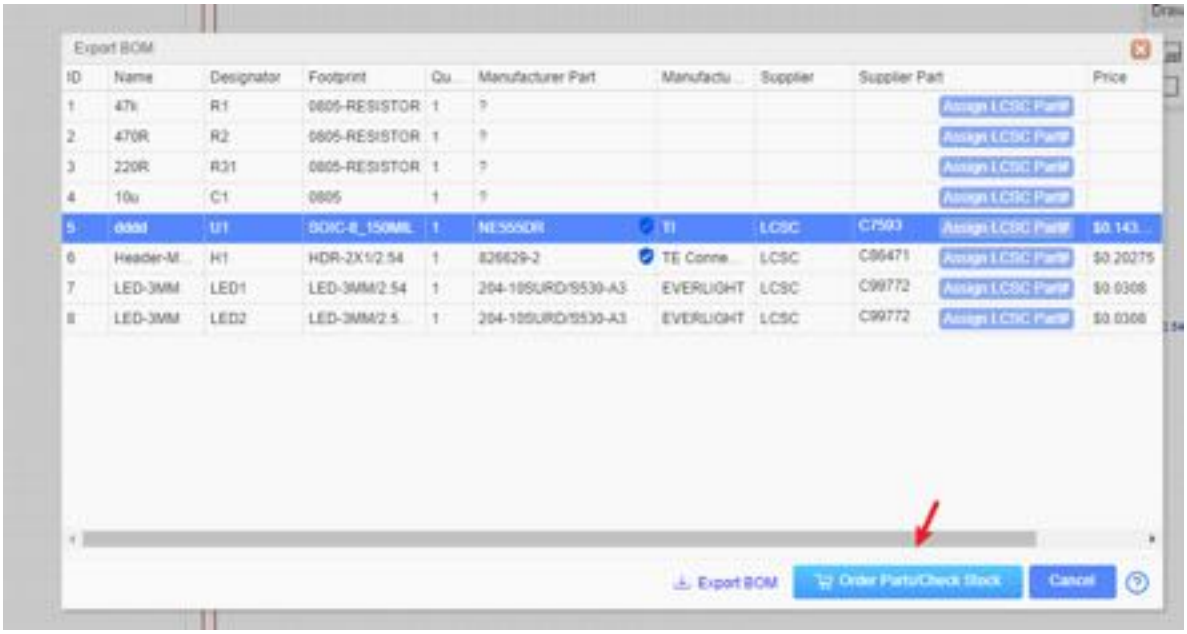
In this dialog，you can click the buttom to assign LCSC part's order code for your components.



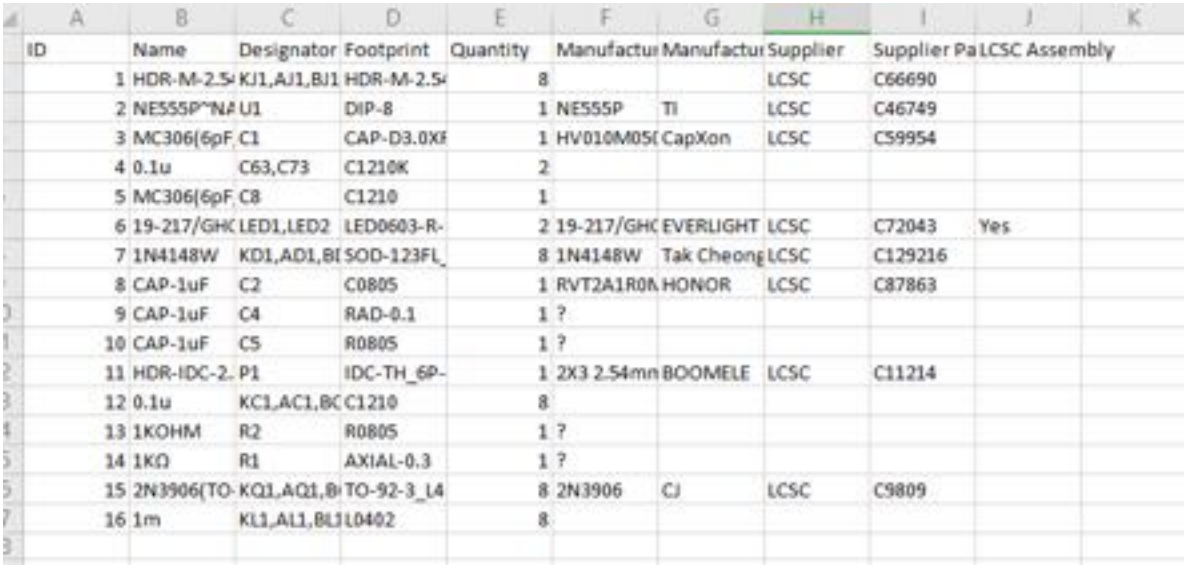
After clicking on the assign icon，the components and footprints search dialog will pop up, and you can choose which component you want to assign.



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-2.5	KJ1,AJ1,BJ1	HDR-M-2.5	8			LCSC	C66690	
2	NE555P~NA	U1	DIP-8	1	NE555P	TI	LCSC	C46749	
3	MC306(6pF	C1	CAP-D3.0XF	1	HV010M05(	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,B	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 NetList

EasyEDA can export the netlist for the whole active project:

File > Export NetList > Spice...