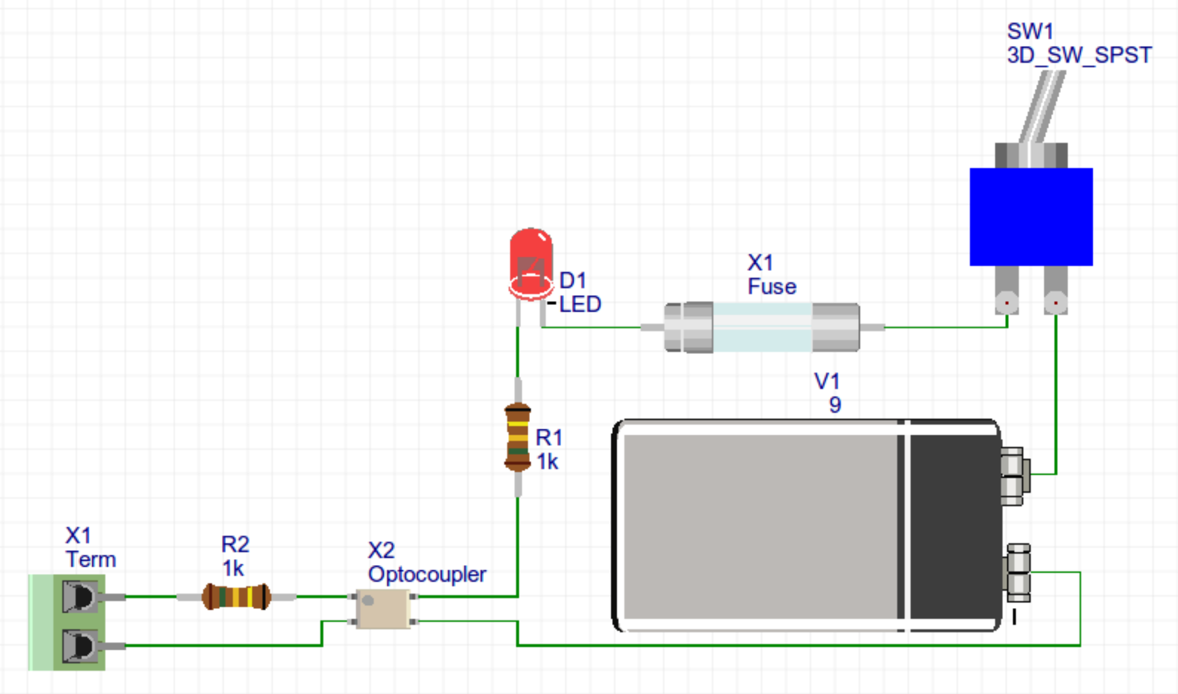


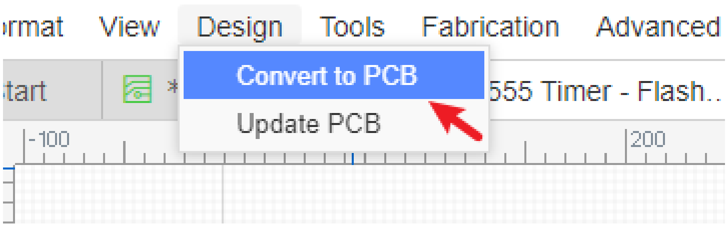
Another powerful feature is that it is possible to import symbols from `Kicad`, `Eagle` or `Altium` libraries.

PCB Layout

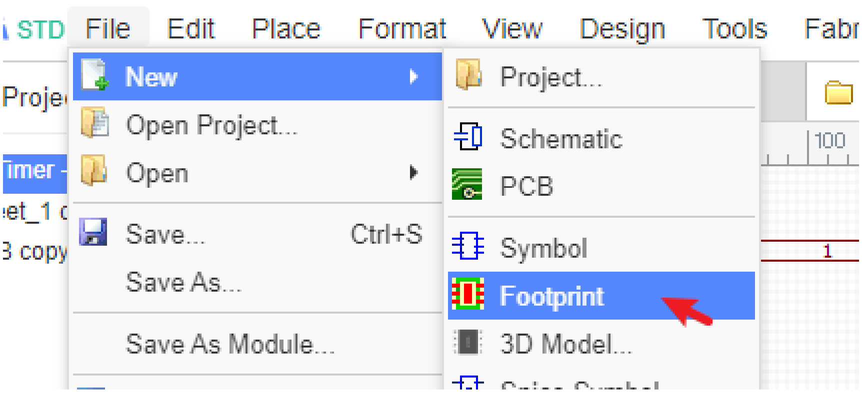
When you are satisfied with your schematic design and simulation results, you can then quickly proceed to produce your finished and populated PCB without leaving EasyEDA.

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.

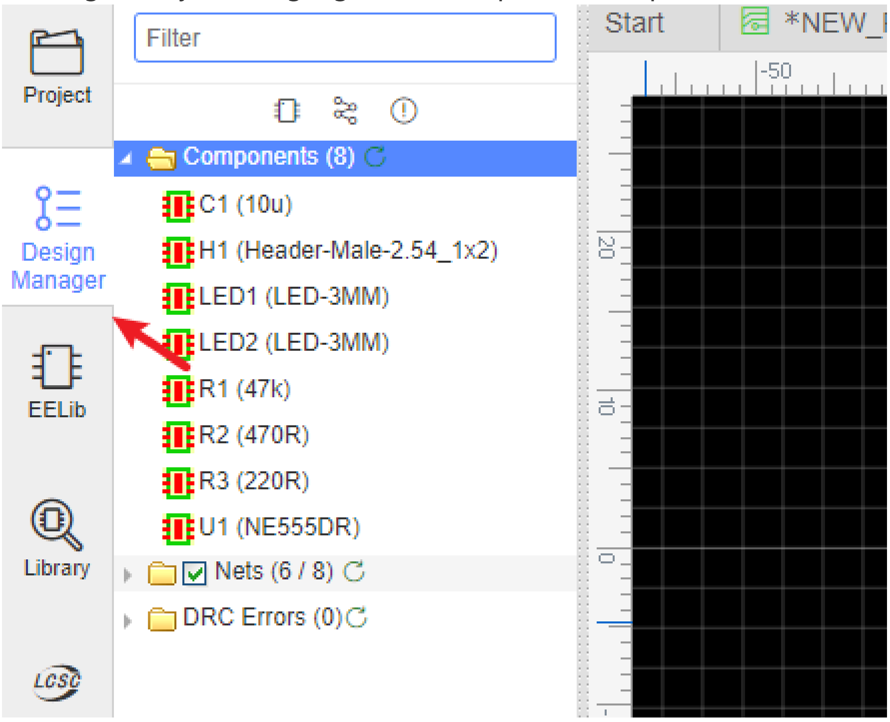
- Passing an EasyEDA Schematic into the PCB Design editor is as easy as clicking a button: Just click the **Convert to PCB** using: "Menu - Design - Convert to PCB".



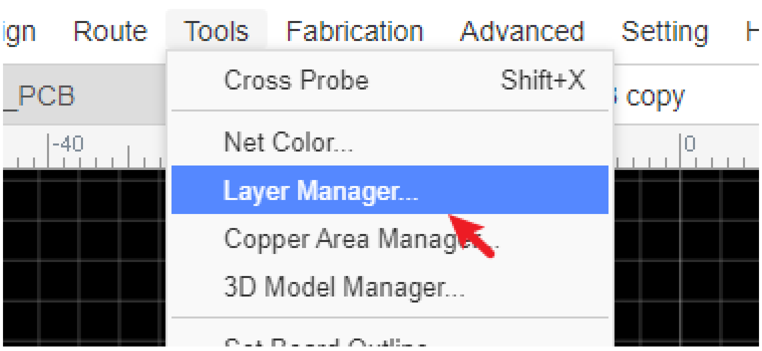
- EasyEDA has extensive component footprints. You can also build up your own library of unusual and specialized parts by copying and modifying existing parts or from scratch using EasyEDA’ s powerful footprint creation and editing tools.



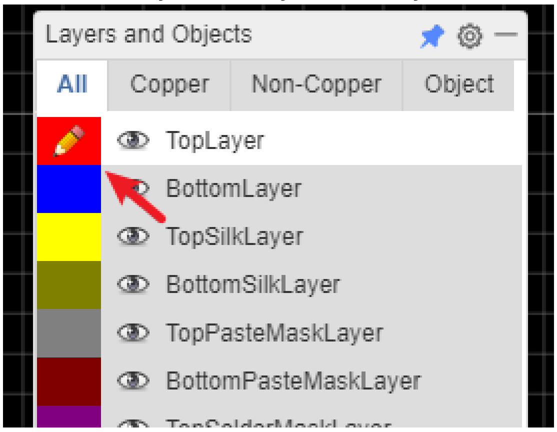
- When working in the PCB Design canvas there is a PCB Design Manager which works in a similar way to the Schematic design canvas, this will help you locate items and navigate your way around.
  Left Navigation Panel > Design Manager
  The PCB Design Manager is a powerful tool for finding components, tracks (nets) and pads (Net Pads).
  Clicking on any item highlights the component and pans it to the center of the window.



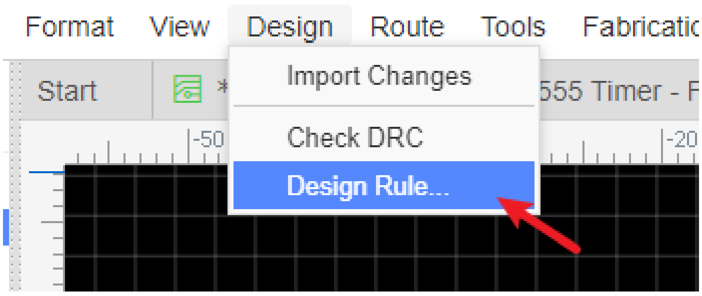
- You can set up layers used in the PCB and their display colours and visibility using **Top Menu - Tools - Layer Manager...**



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

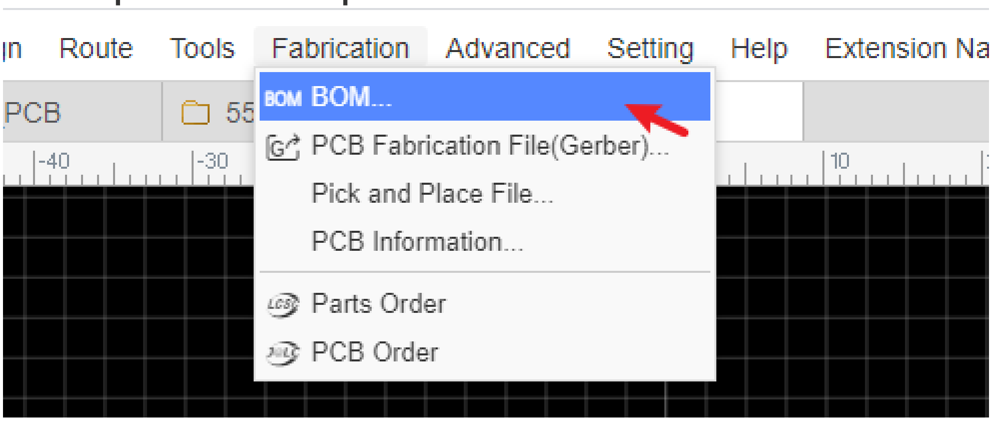


- Default track widths, clearances and via hole dimensions can all be configured in the Design Rule Check dialog which is opened by:
Top Menu > Design > Design Rule...



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

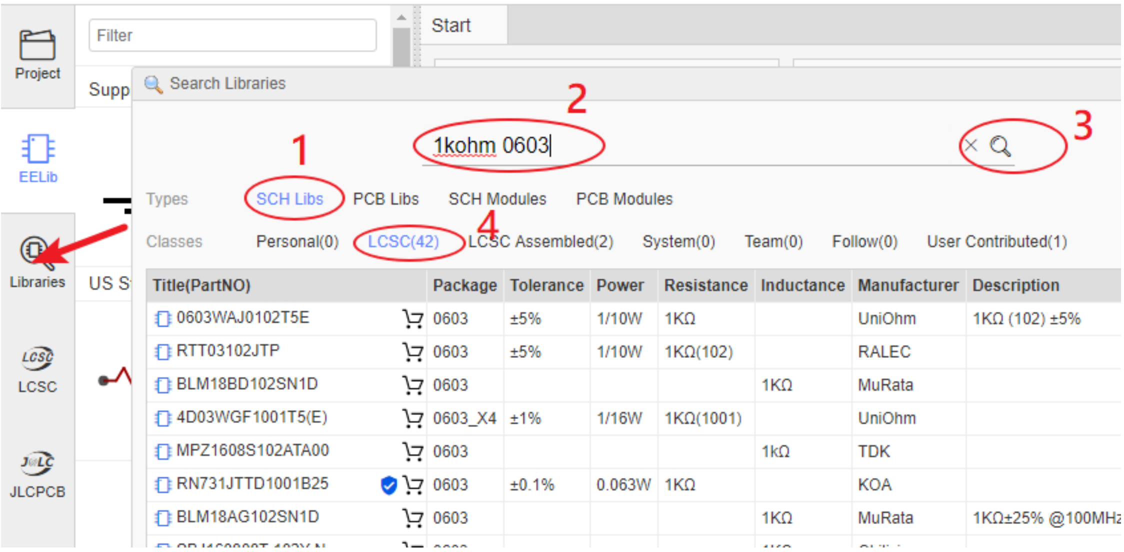
Searching footprints is the same as searching symbols by using **Library** in the Schematic editor.
You can place the selected footprints in the canvas after a successful search.

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.

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.

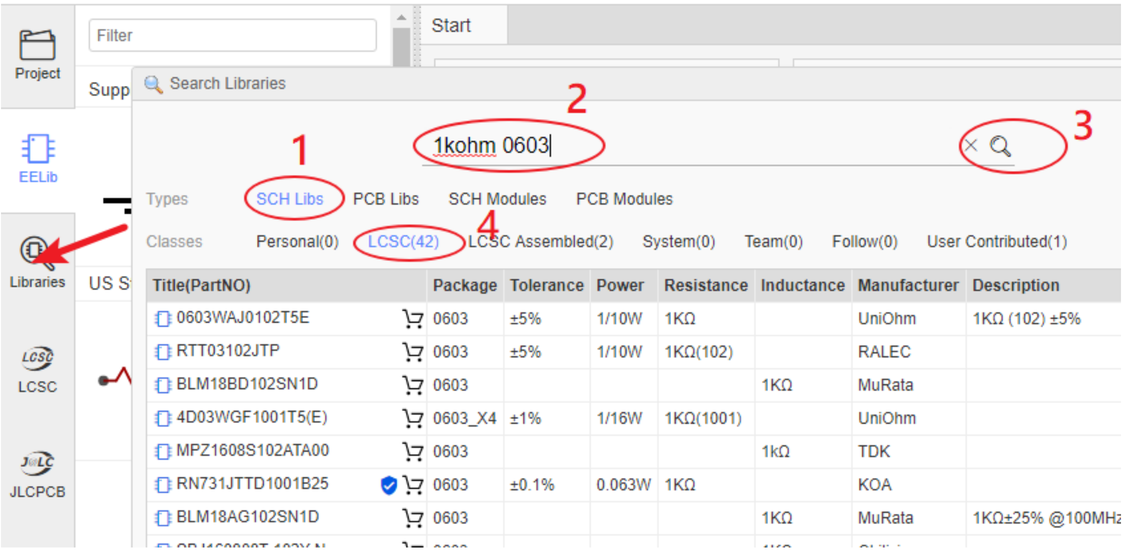
Search footprints

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.

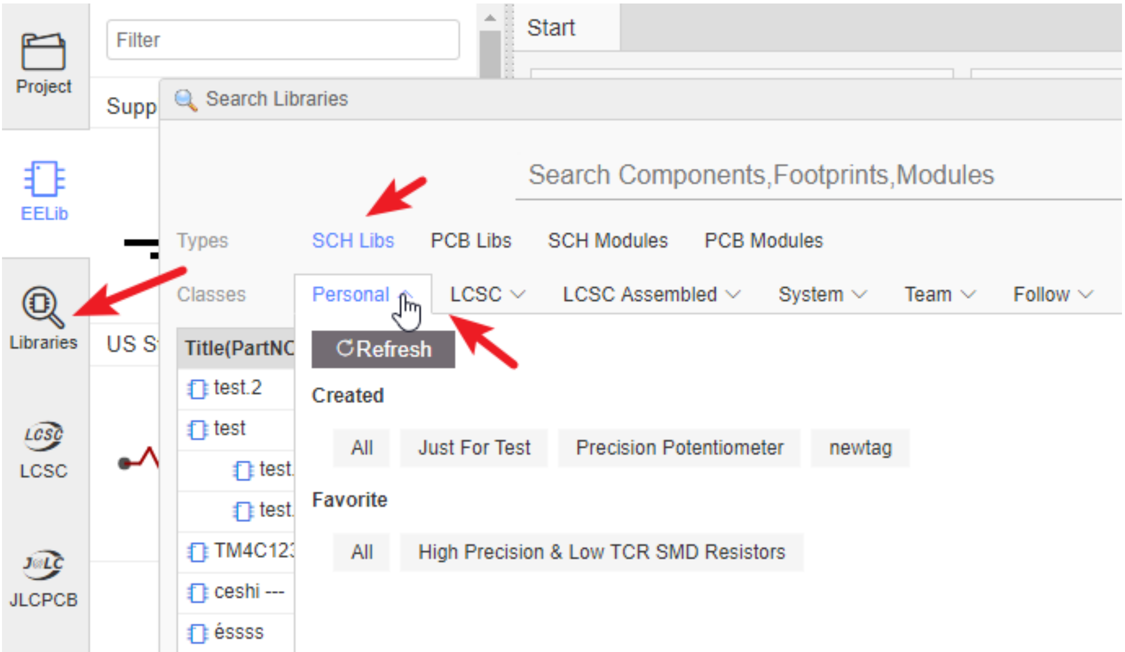
Libraries Management

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!

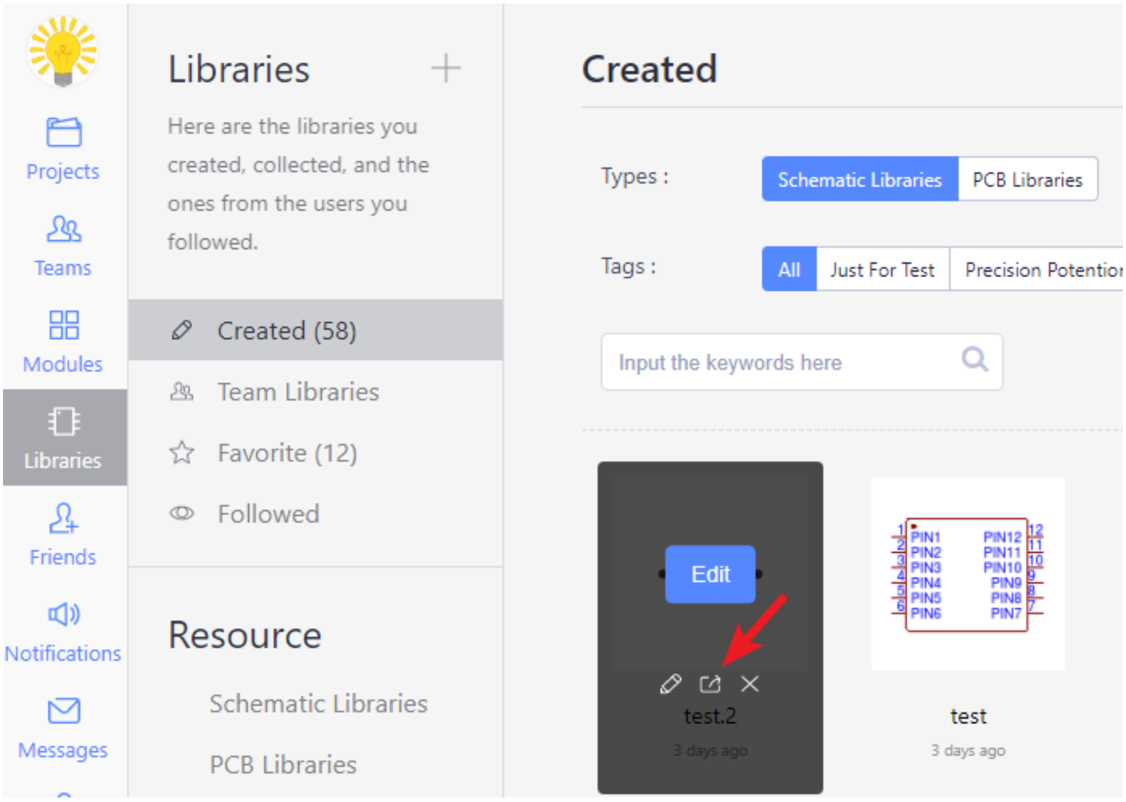
With these libraries you can enjoy using EasyEDA without having to spend so much time hunting for or building schematic symbols and PCB footprints.

- **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

Version-Control

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.

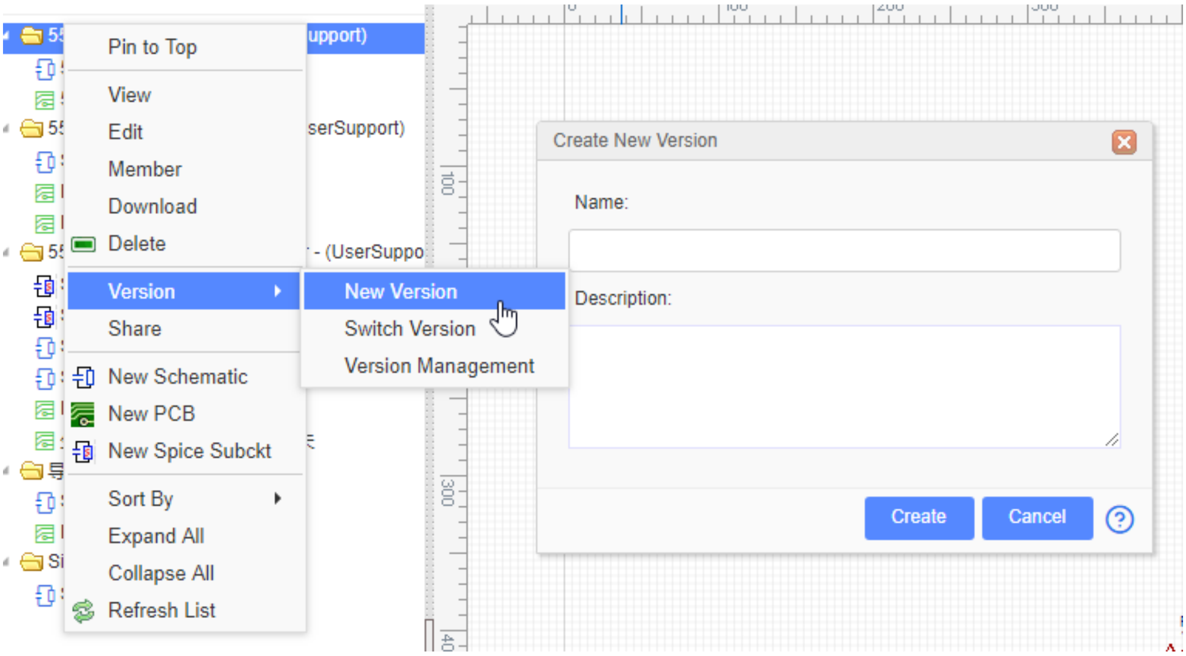
You can create up to 10 versions for every project. To create a new version, you must first delete an older version.

Create New Version

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

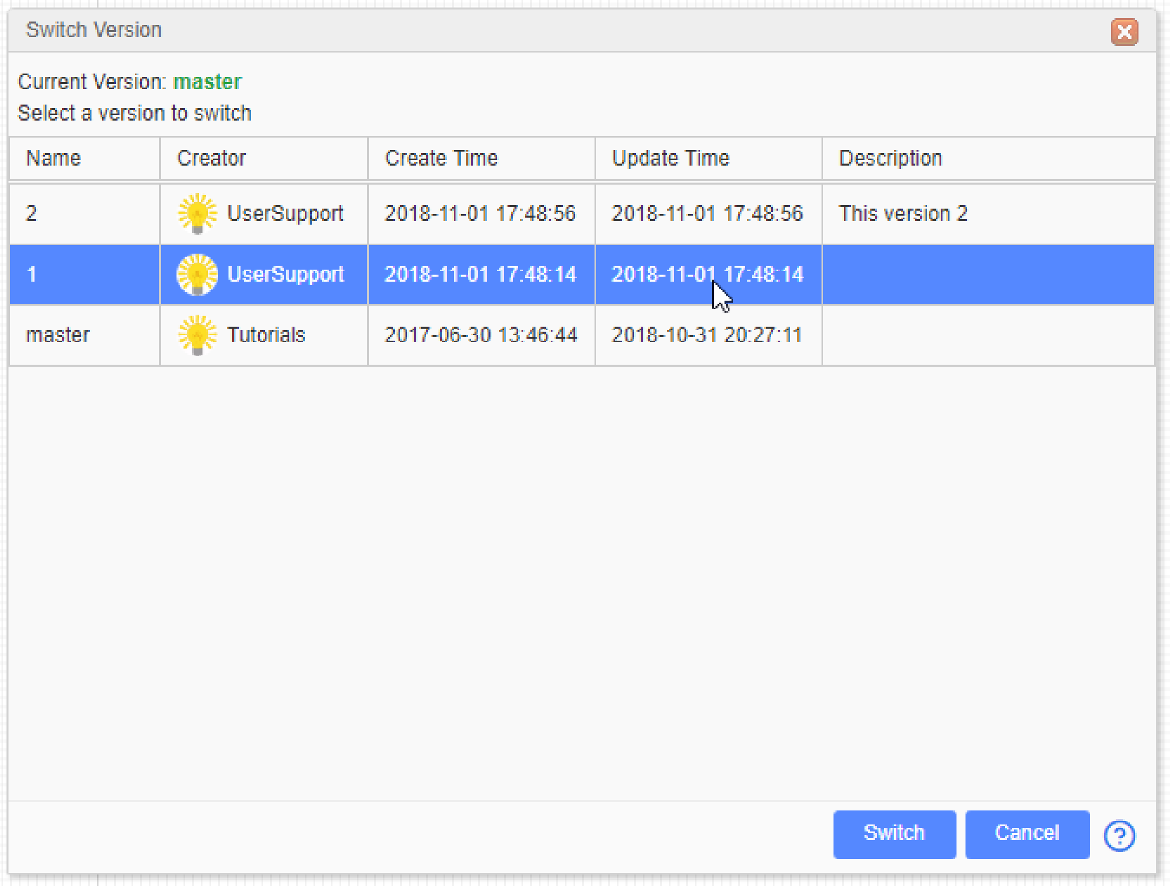
In the new version dialog, you type the version's name and description, and then create it.

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



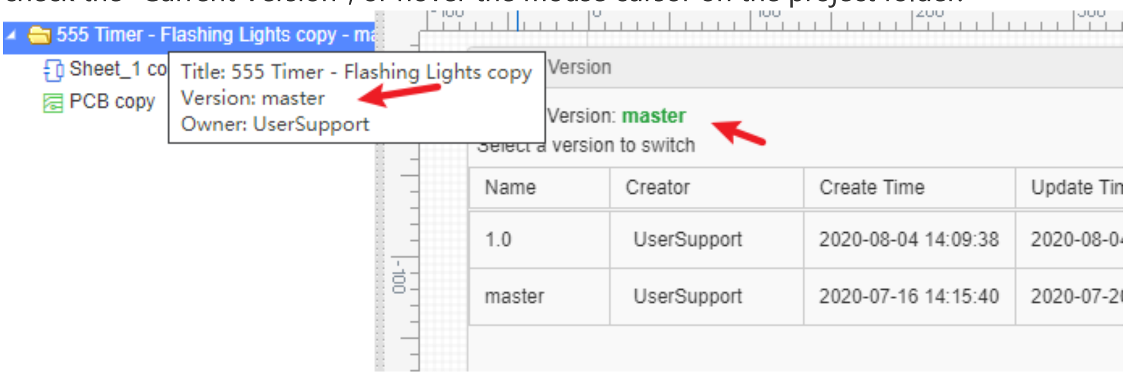
Switch Version

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



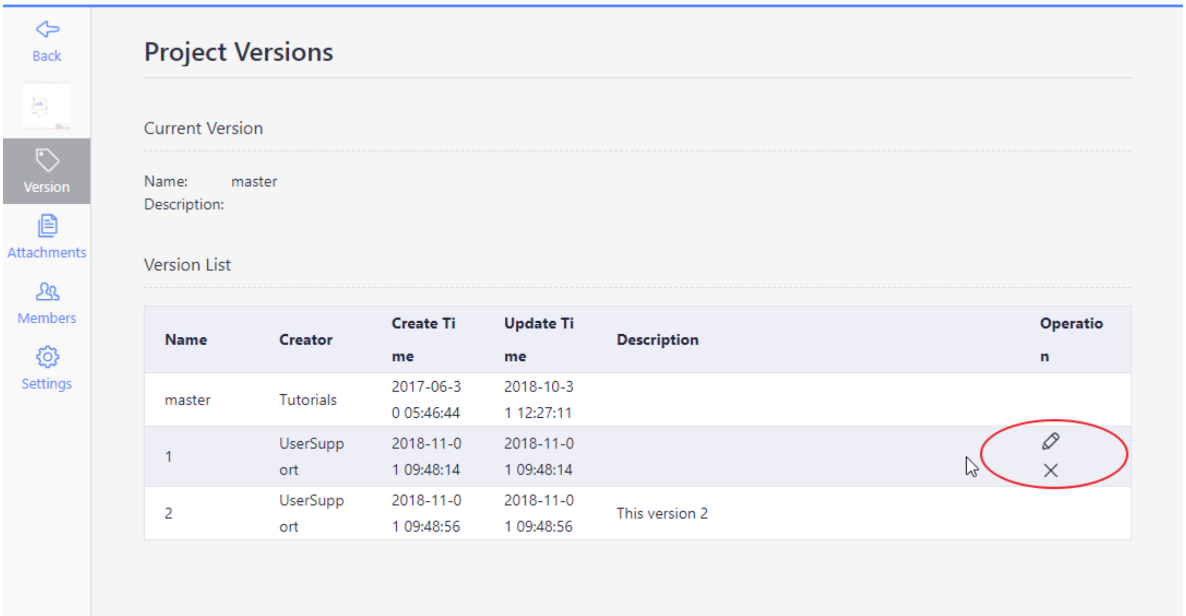
Note:

- 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.



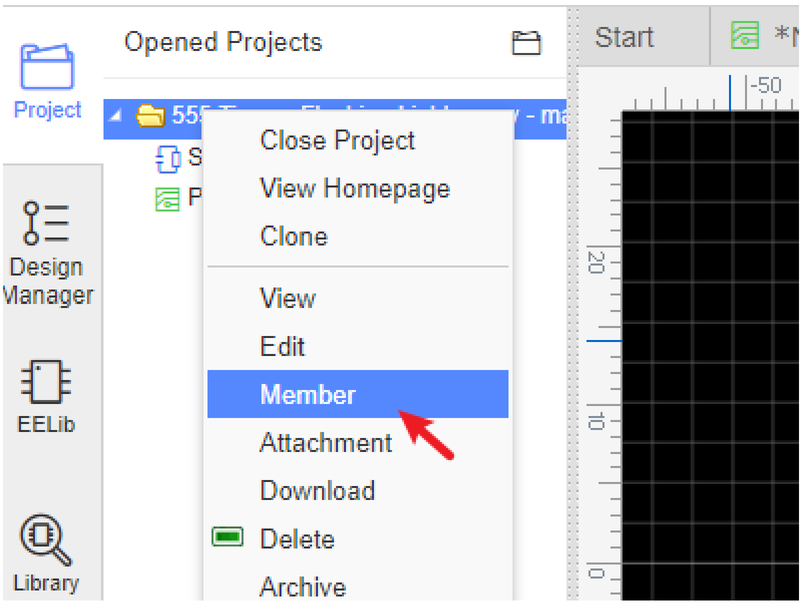
Project Member

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