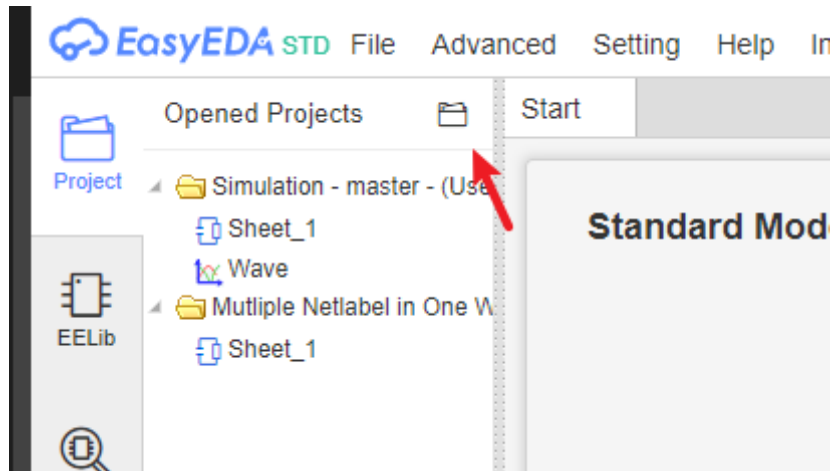
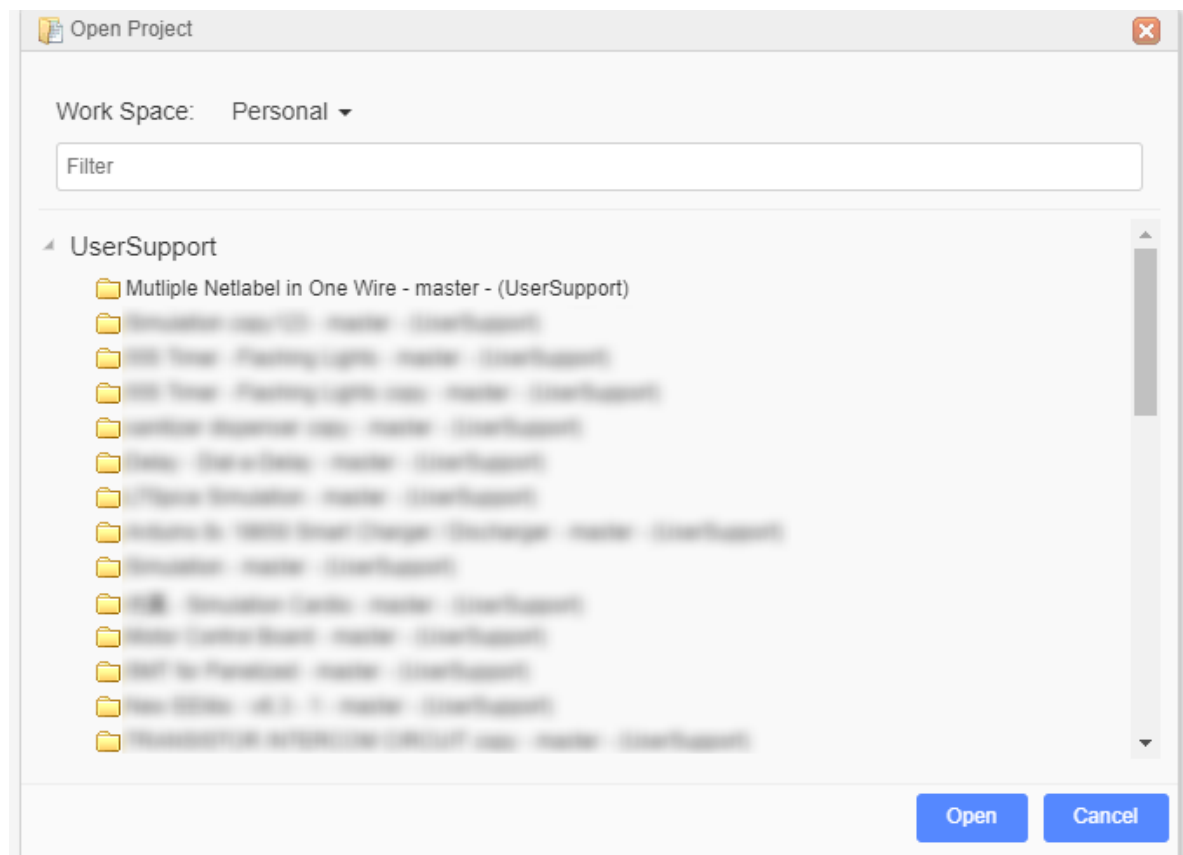


Or click the Opened Project "open project" icon.

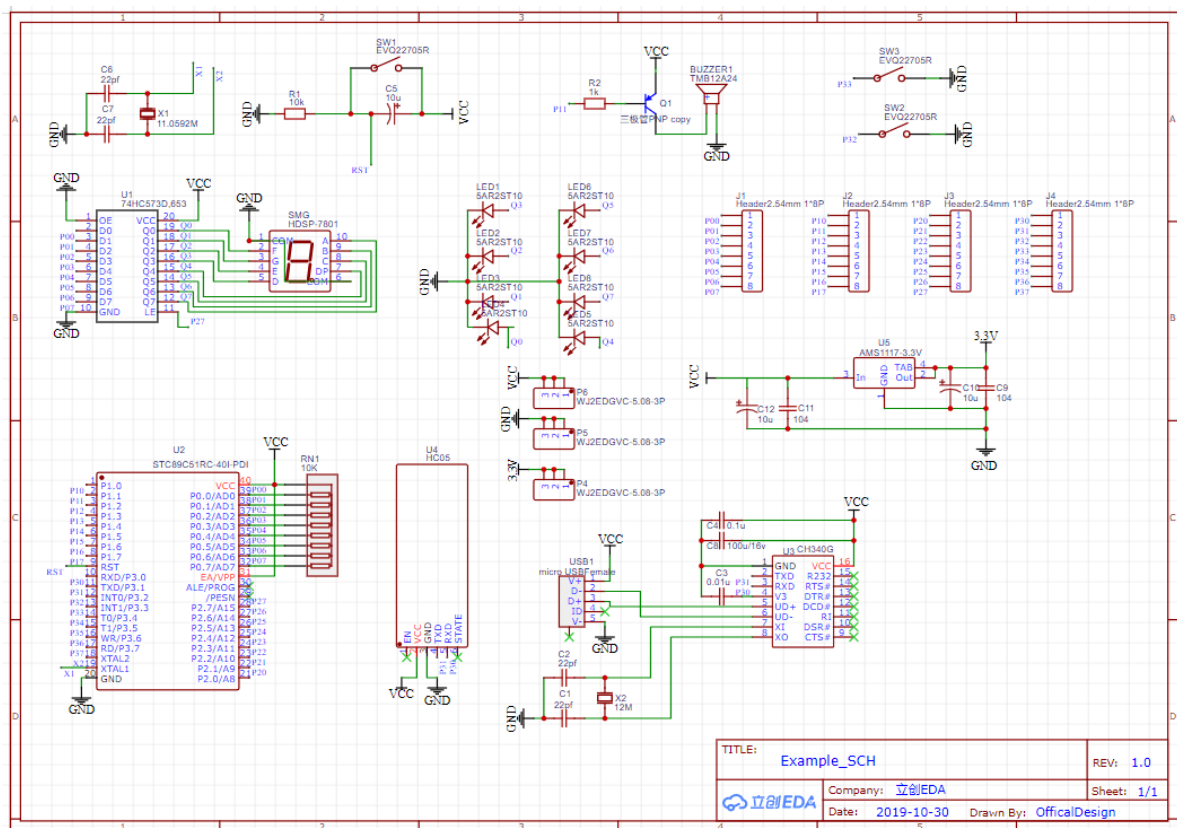


Select the project and open it.



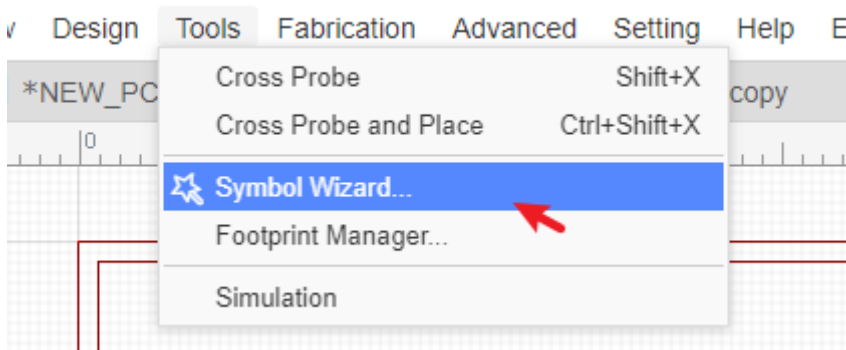
Schematic Capture

EasyEDA can create professional quality schematics.



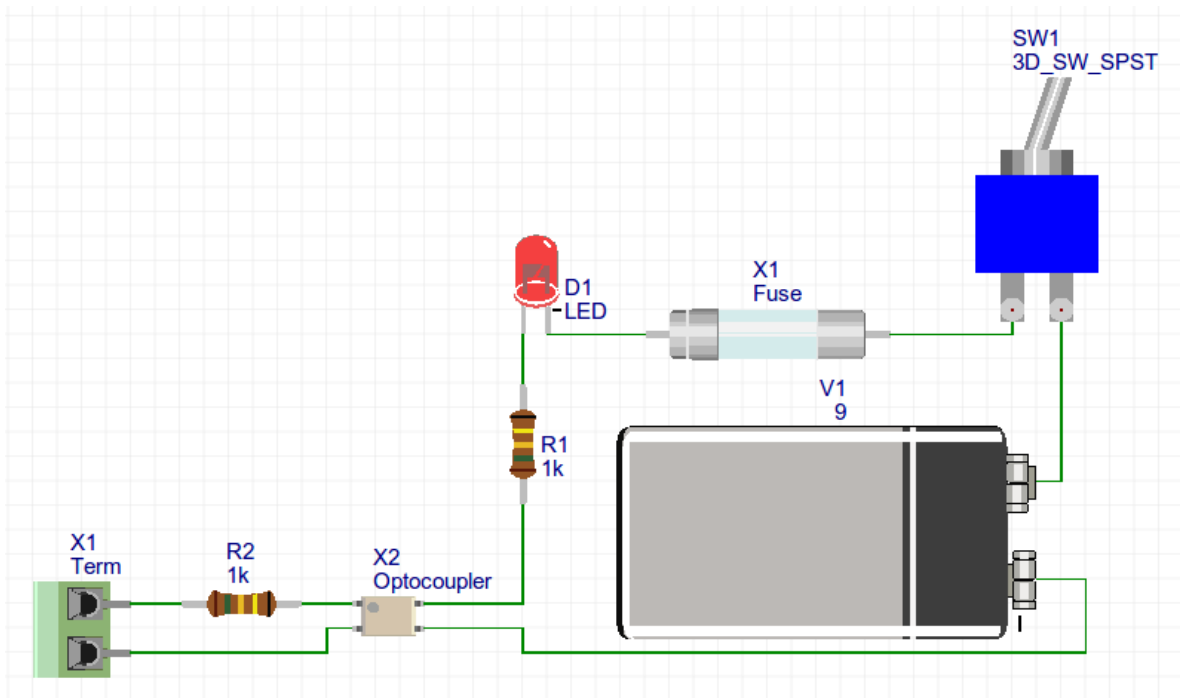
Because EasyEDA has some simple but powerful drawing capabilities, you can create your own symbols either by copying existing symbols into your own library and then editing and saving them, or by drawing them from scratch.

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:



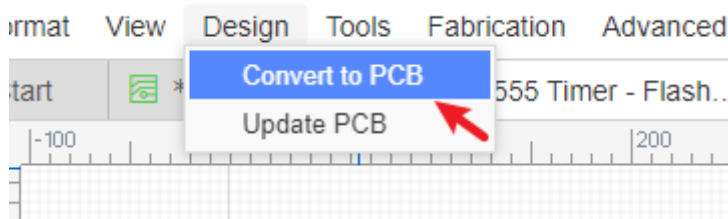
Another powerful feature is that it is possible to import symbols from [Kicad](#), [Eagle](#) or [Altium](#) libraries.

PCB Layout

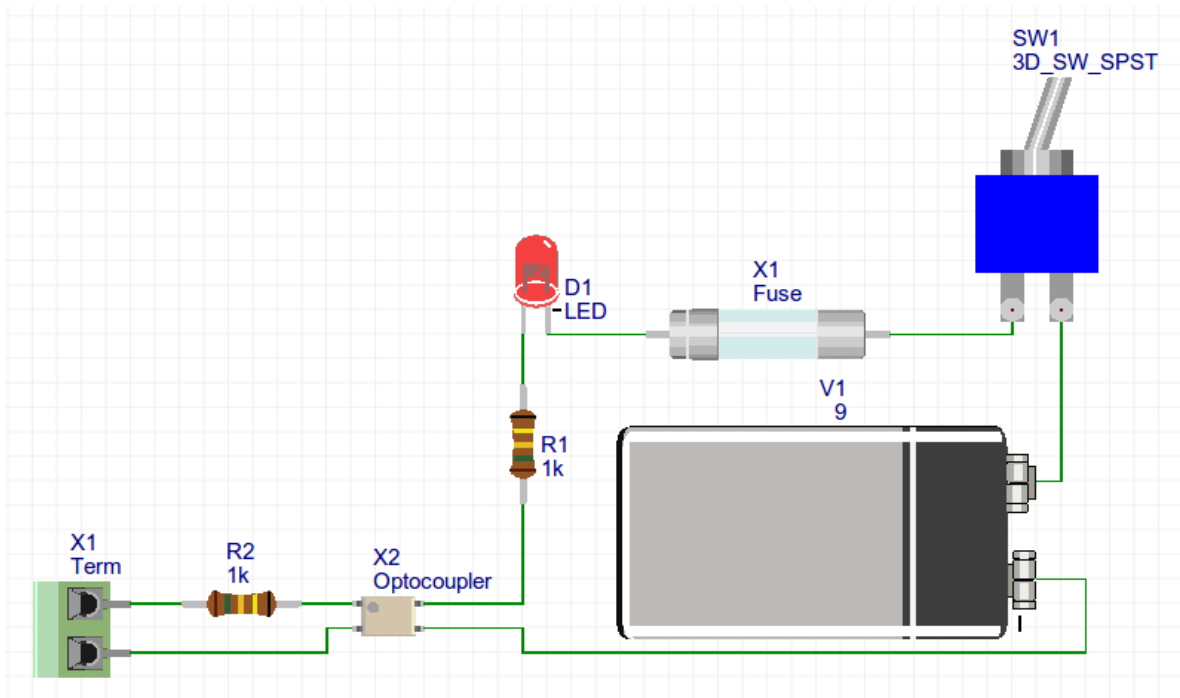
When you are satisfied with your schematic design and simulation results, you can then quickly proceed to produce your finished and populated PCB without leaving EasyEDA.

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



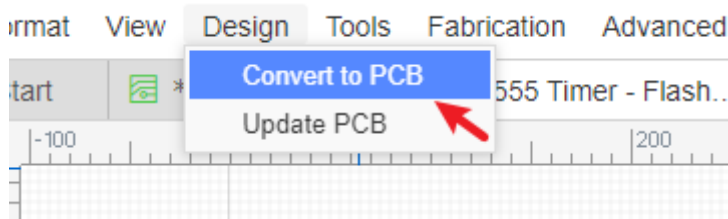
Another powerful feature is that it is possible to import symbols from [Kicad](#), [Eagle](#) or [Altium](#) libraries.

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.

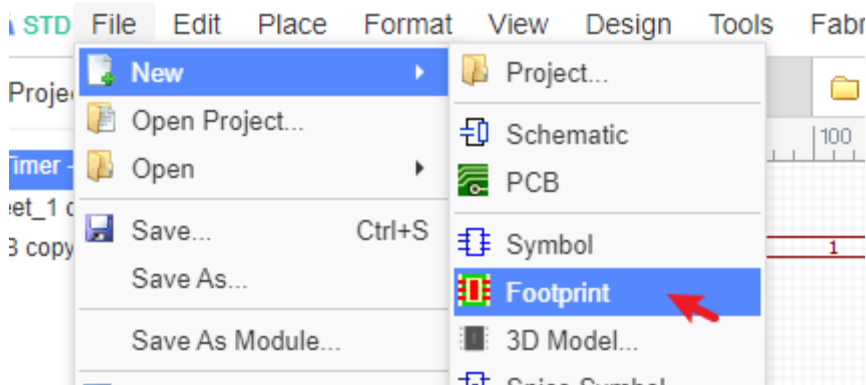
EasyEDA's PCB Design canvas helps you to quickly and easily lay out even complex multilayer designs from schematics you have already created in the Schematic canvas or directly as a layout with no schematic.

- 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.

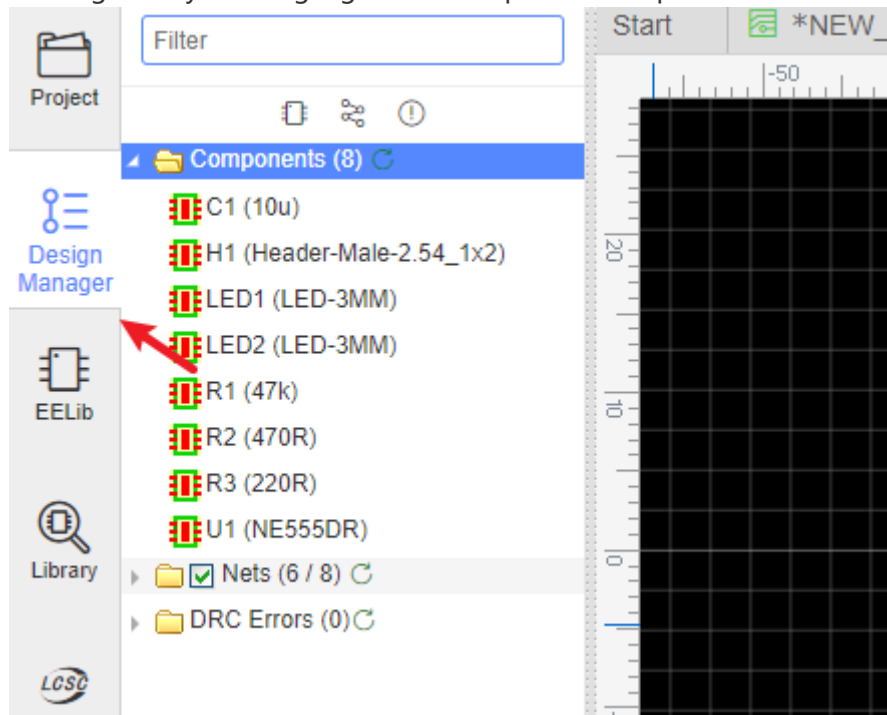


- When working in the PCB Design canvas there is a PCB Design Manager which works in a similar way to the Schematic design canvas, this will help you locate items and navigate your way around.

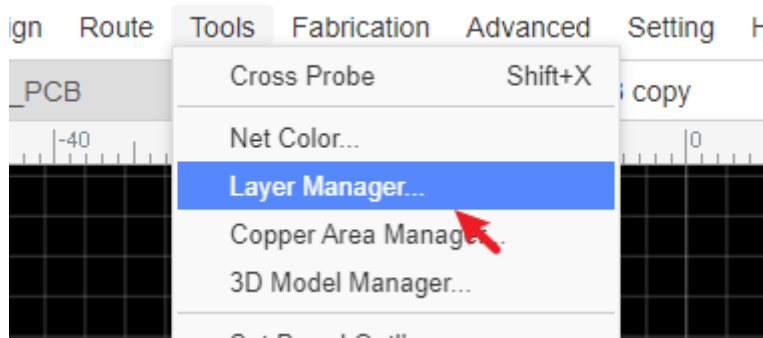
Left Navigation Panel > Design Manager

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

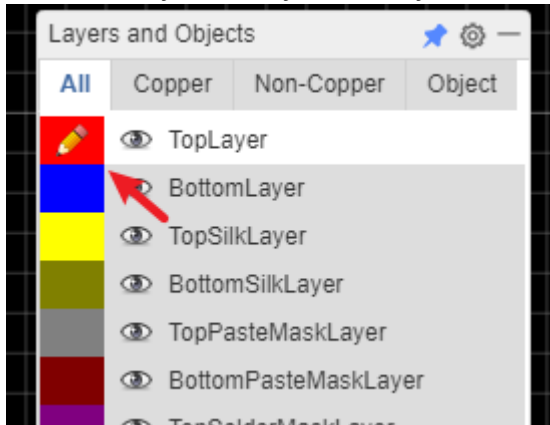
Clicking on any item highlights the component and pans it to the center of the window.



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

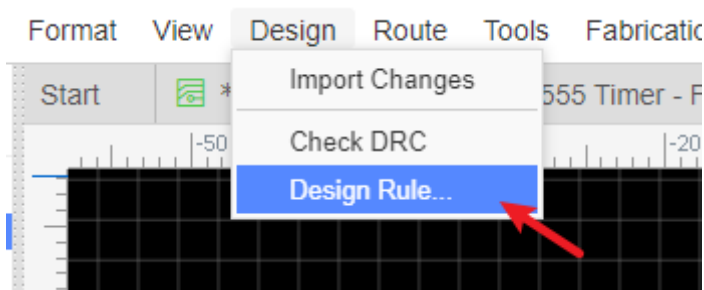


The active layer and layer visibility can be selected using the Layers Toolbar.



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

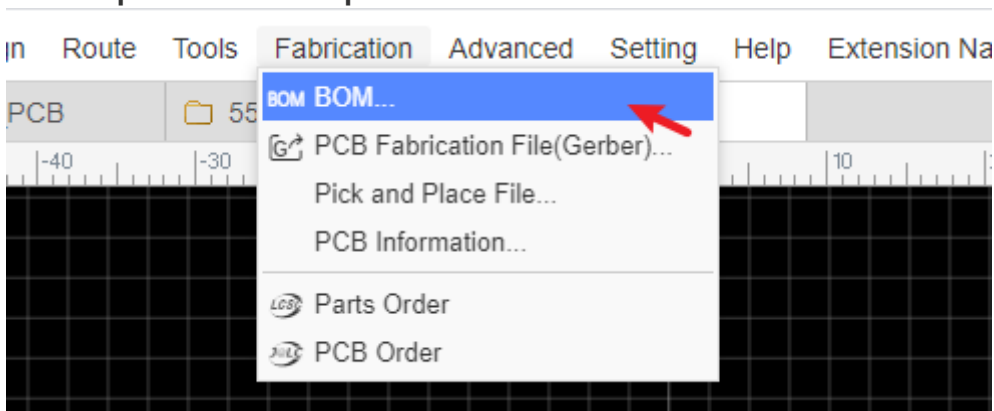
Top Menu > Design > Design Rule...



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

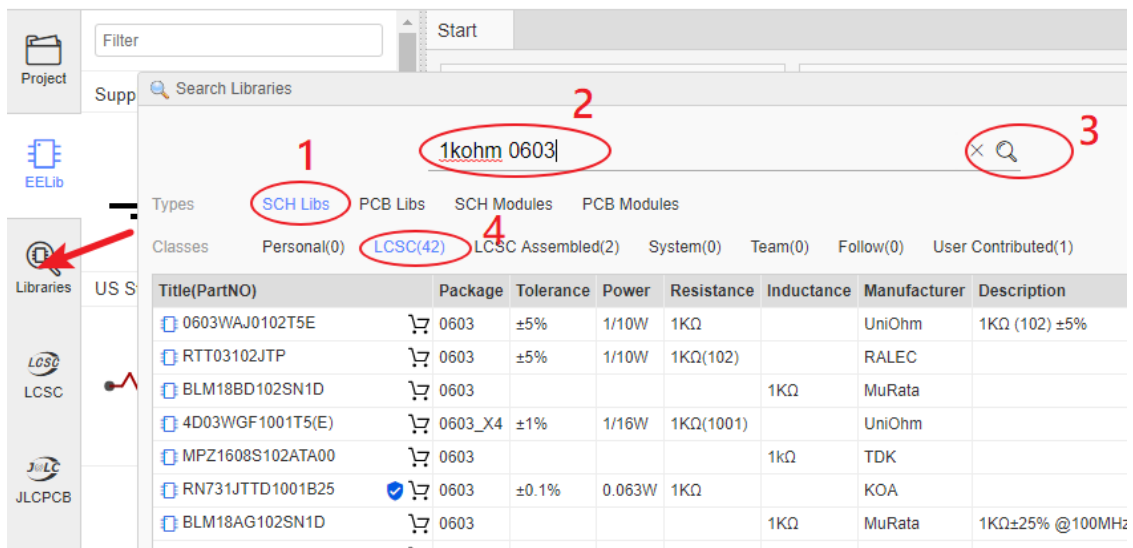
Thanks to the Free and Open Source Kicad Libs and some Open Source Eagle libs, EasyEDA now has 700,000+ components, which should be enough for most projects!

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.