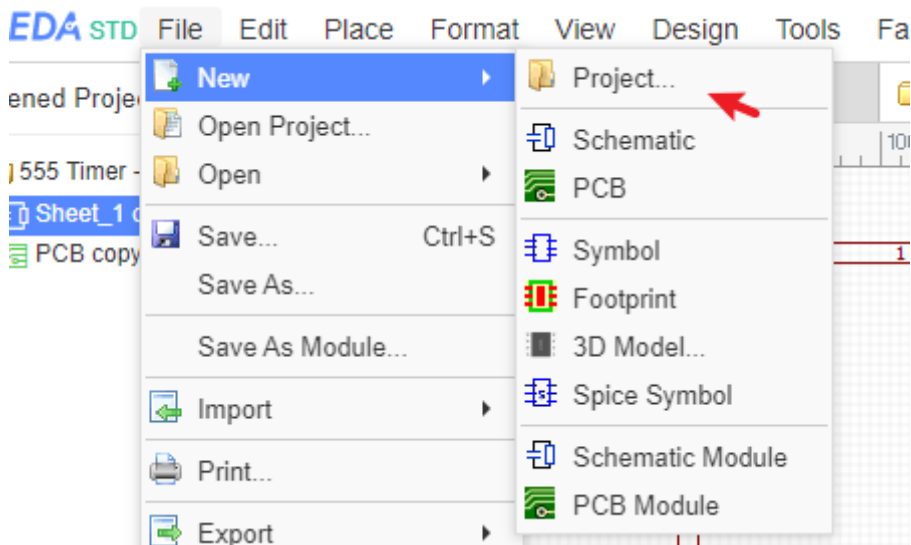


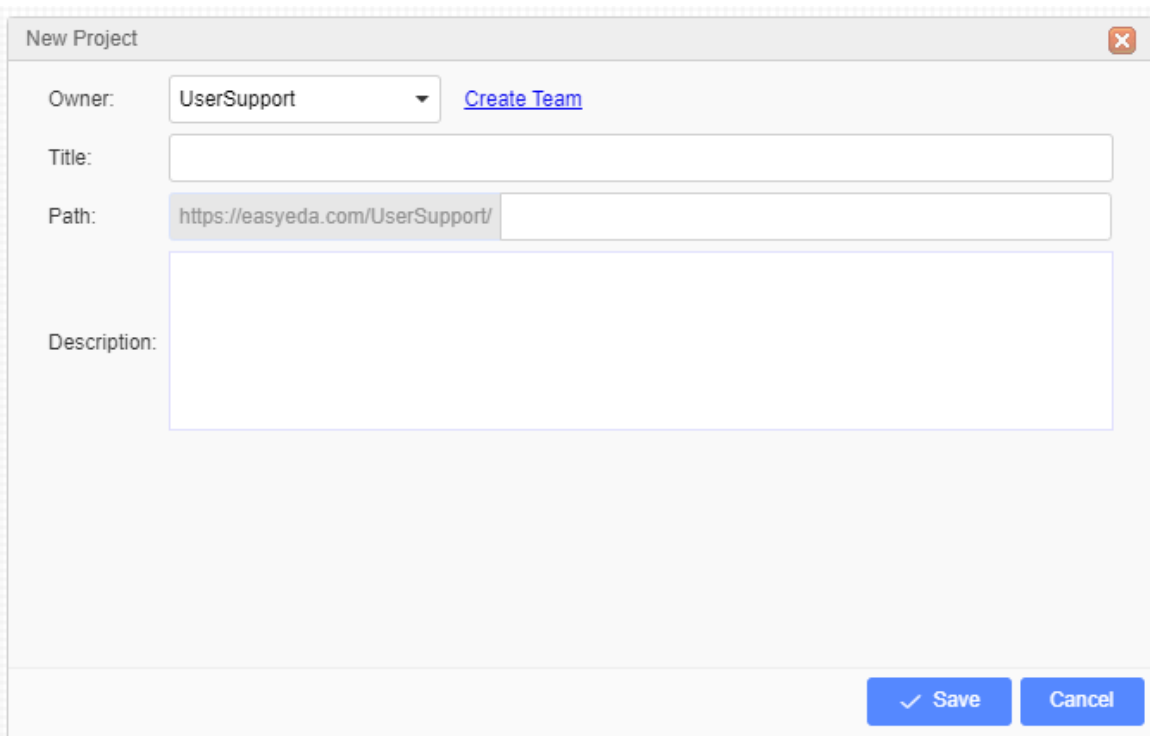
Create Project

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.

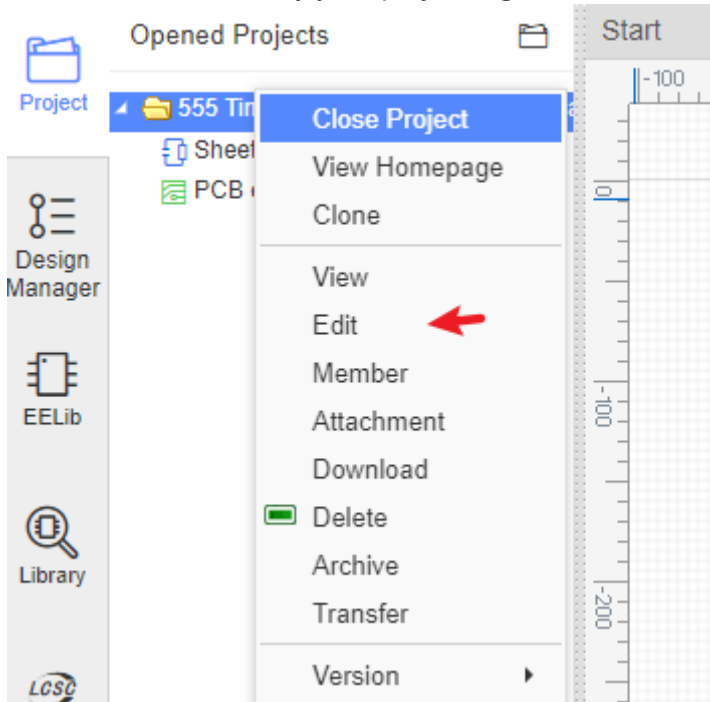
A screenshot of the 'New Project' dialog box in EasyEDA. The dialog has a title bar with a close button. It contains the following fields:

- 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.

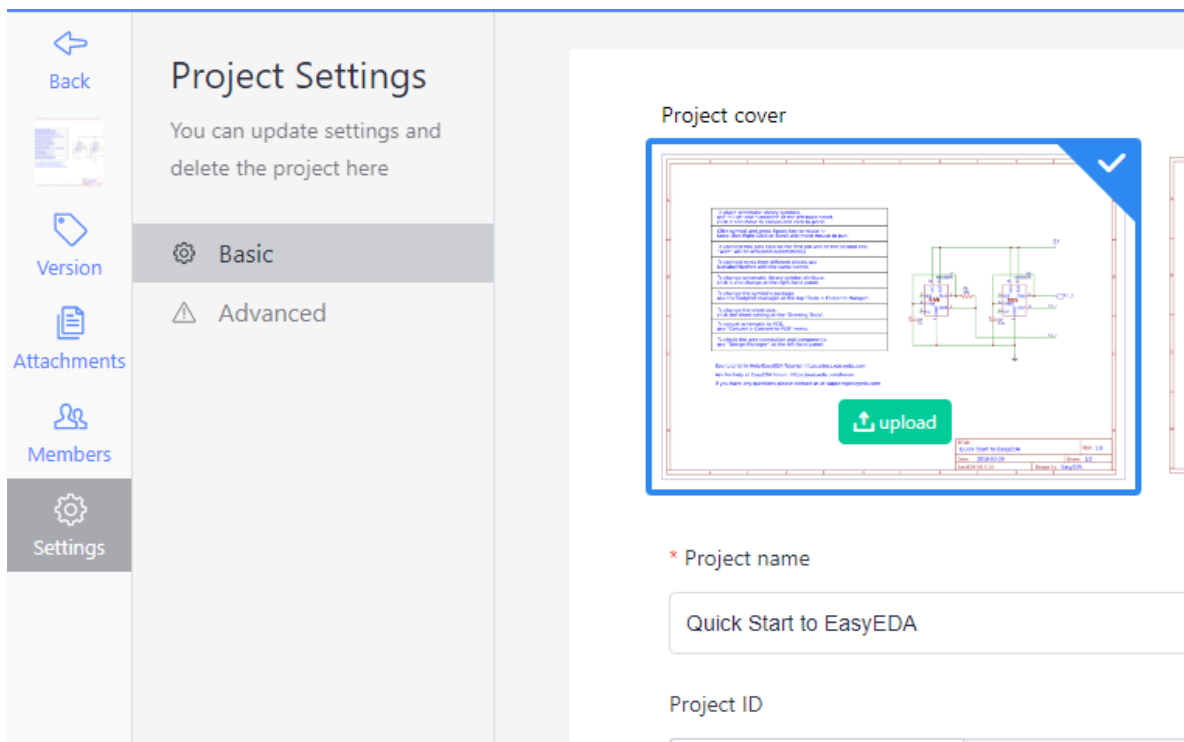
At the bottom right, there are two buttons: 'Save' (with a checkmark icon) and 'Cancel'.

- **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:



this will open a web page where you can edit your project:

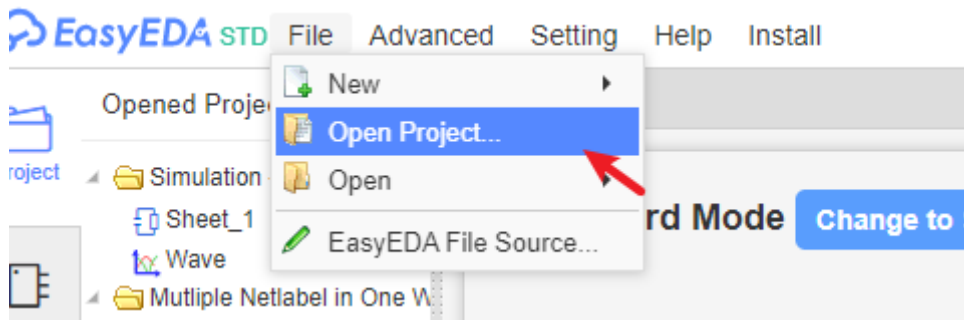


From here, you can change to publish or not, allow other people to comment on your project, or type a more detailed description of the project content. To help you make your project stand out or to maybe simply make a detailed description of your project easier to read, you can use Markdown syntax.

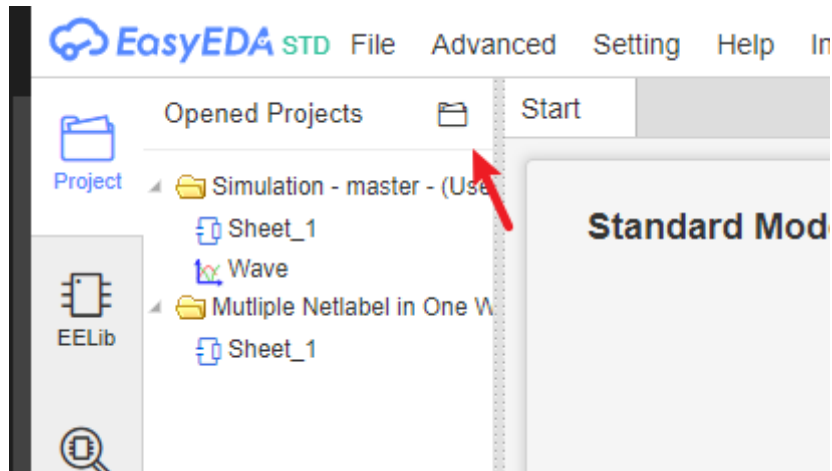
Open Project

You can open your created project using:

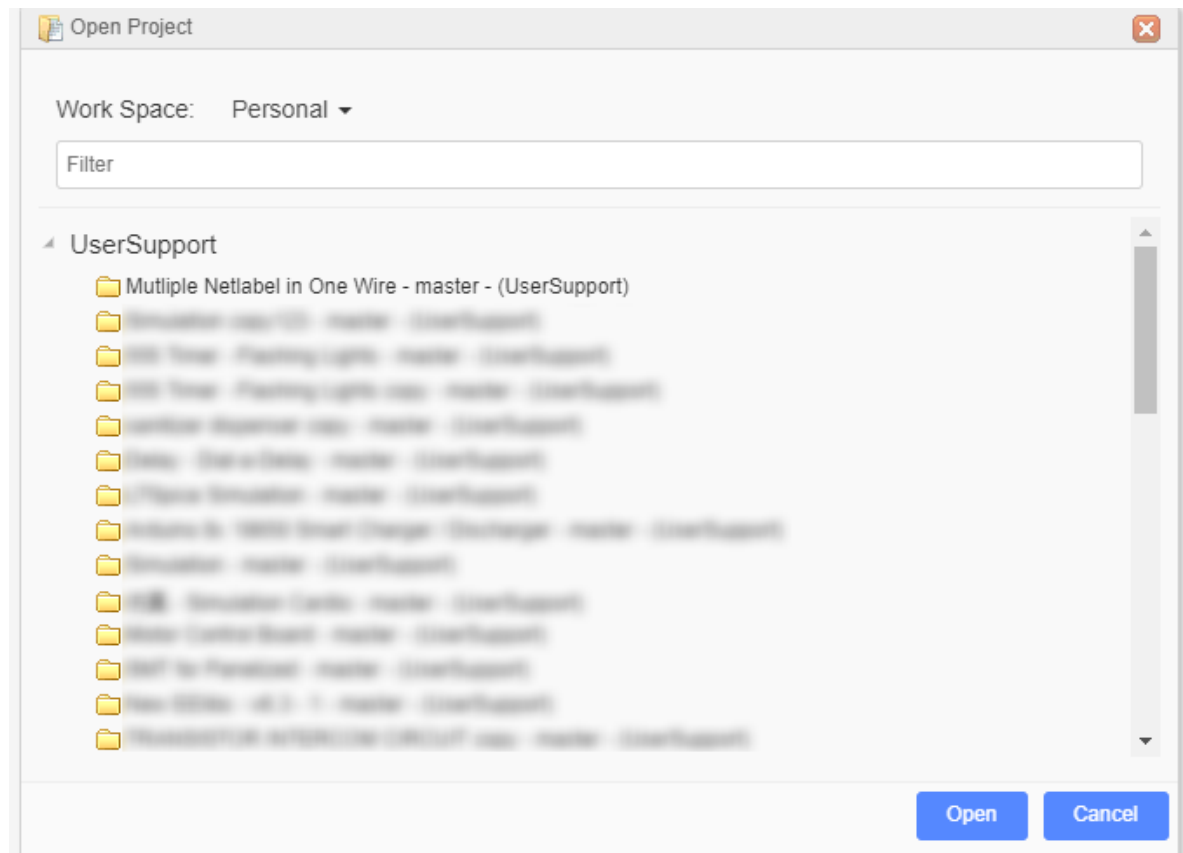
Top Menu - File - Open Project



Or click the Opened Project "open project" icon.

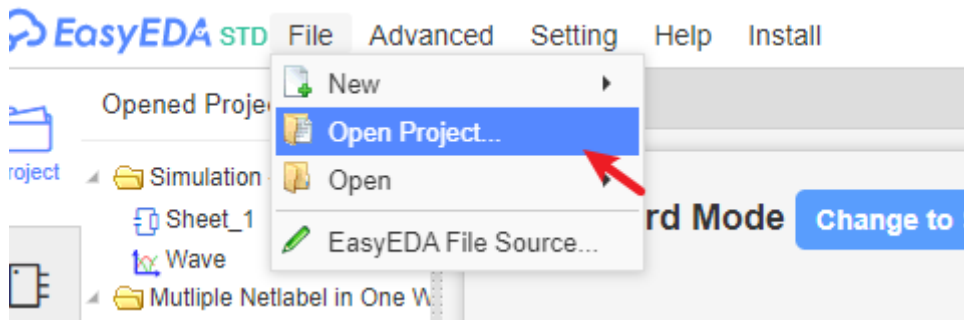


Select the project and open it.

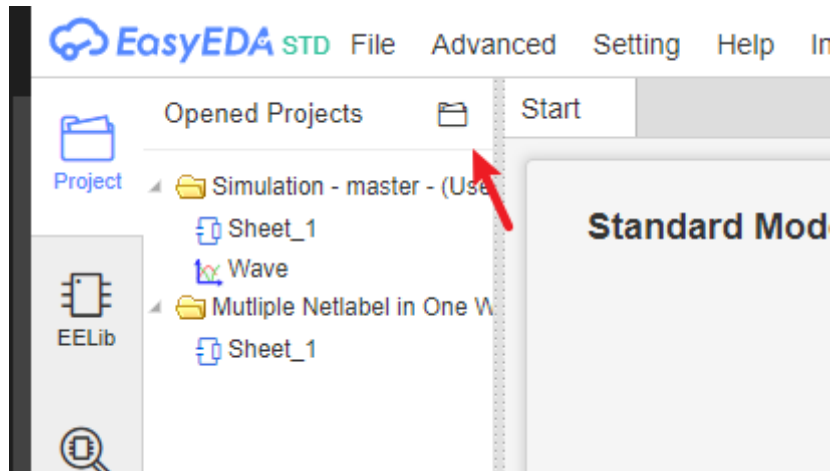


Schematic Capture

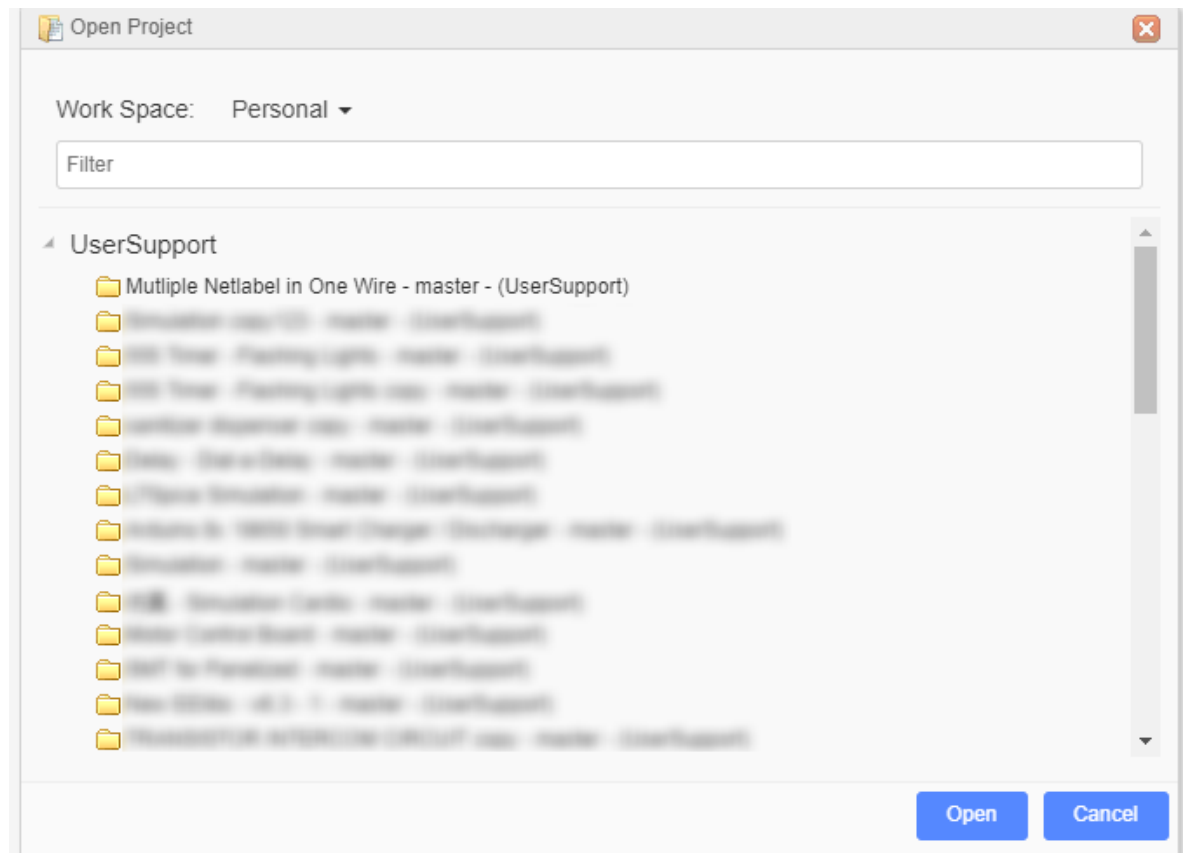
EasyEDA can create professional quality schematics.



Or click the Opened Project "open project" icon.

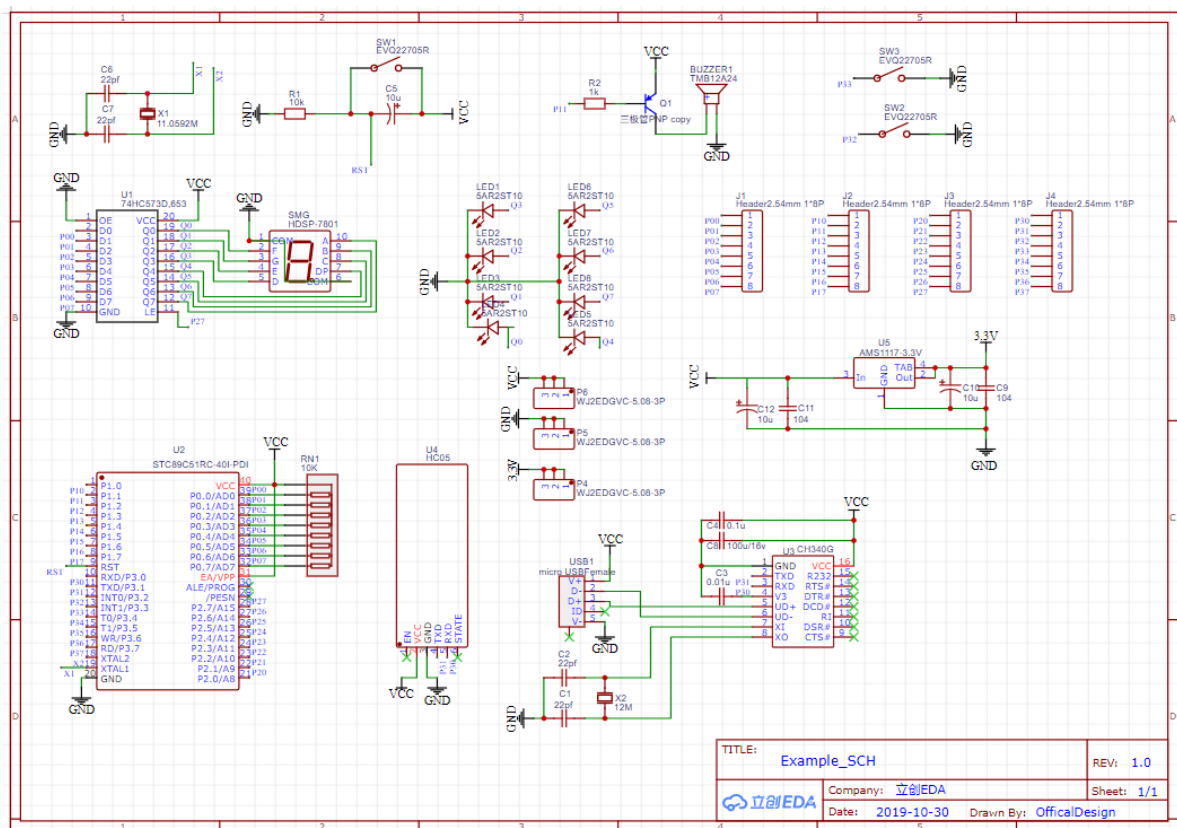


Select the project and open it.



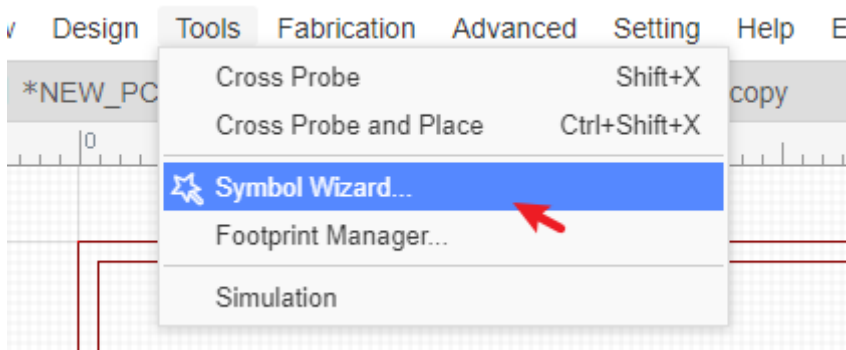
Schematic Capture

EasyEDA can create professional quality schematics.



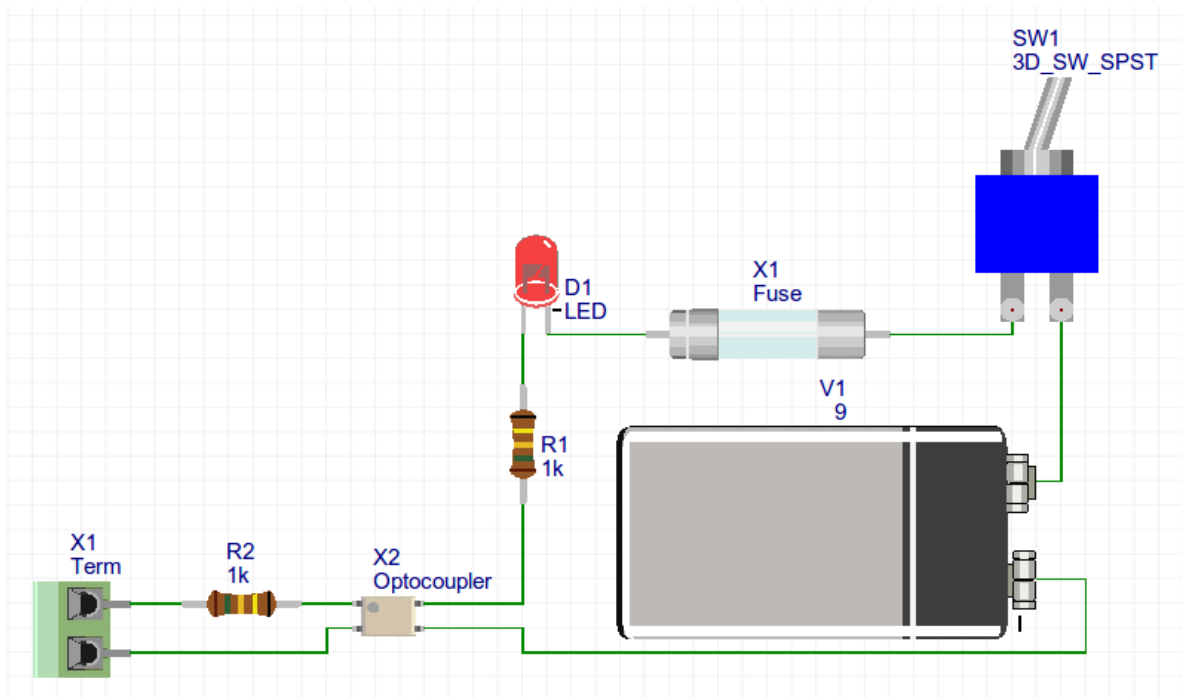
Because EasyEDA has some simple but powerful drawing capabilities, you can create your own symbols either by copying existing symbols into your own library and then editing and saving them, or by drawing them from scratch.

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:



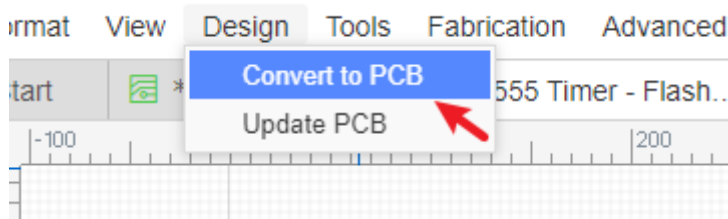
Another powerful feature is that it is possible to import symbols from [Kicad](#), [Eagle](#) or [Altium](#) libraries.

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