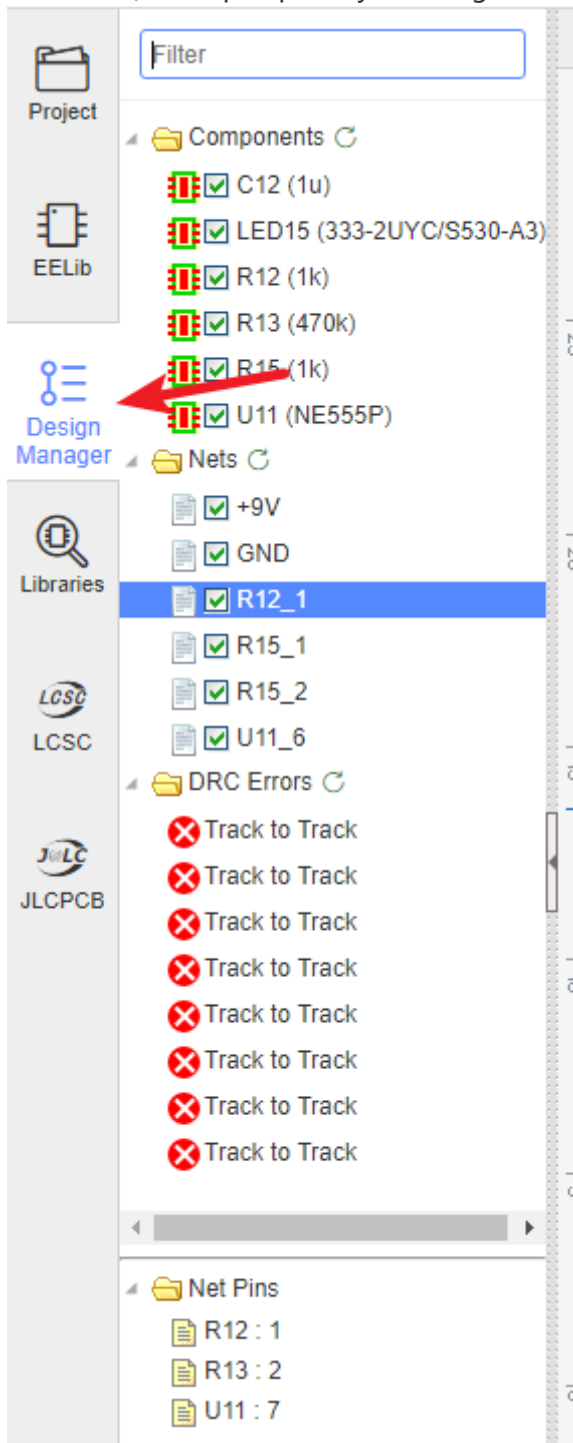
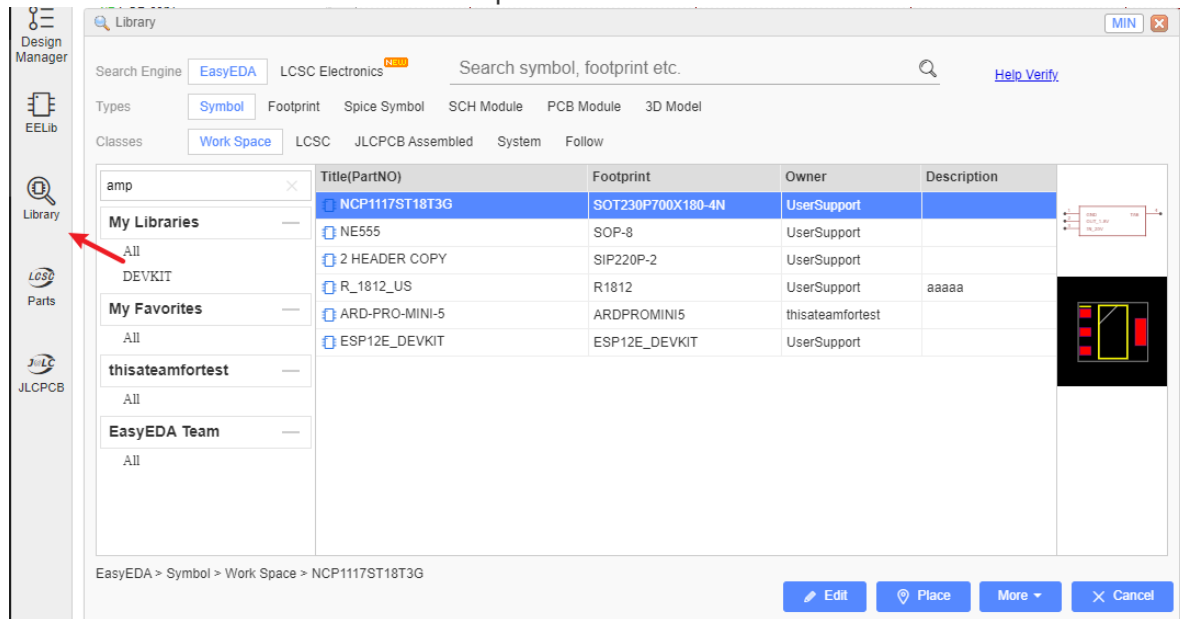


Design Manager, you can check each component and net wiring, and it provides DRC(Design rule check) to help improve your design.



**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://lcsc.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:



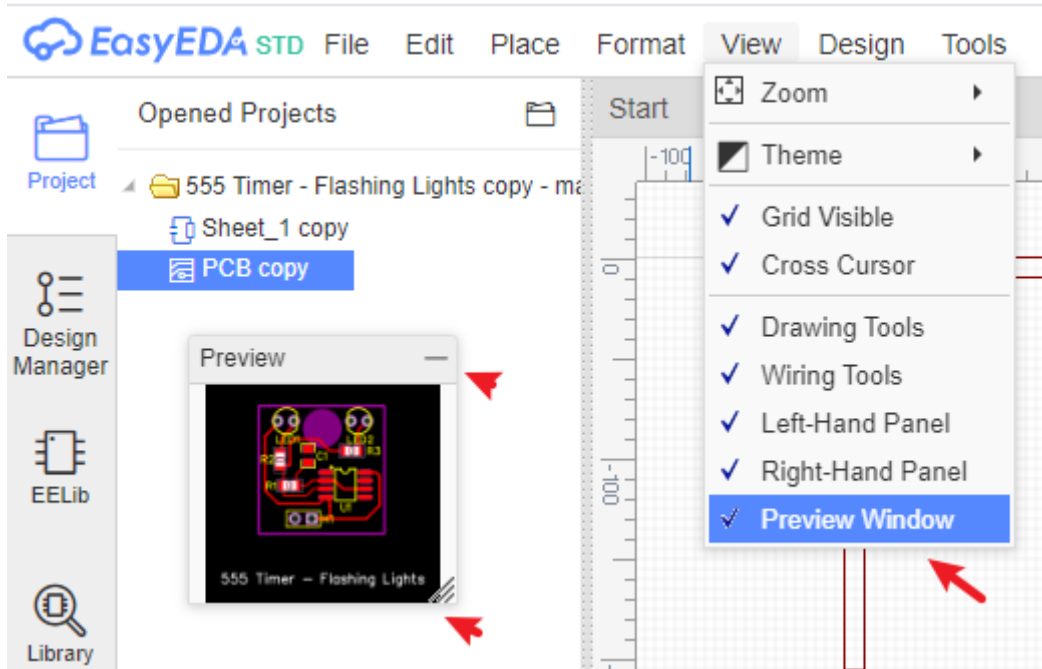
You can find what you need easily and clearly.


## 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:

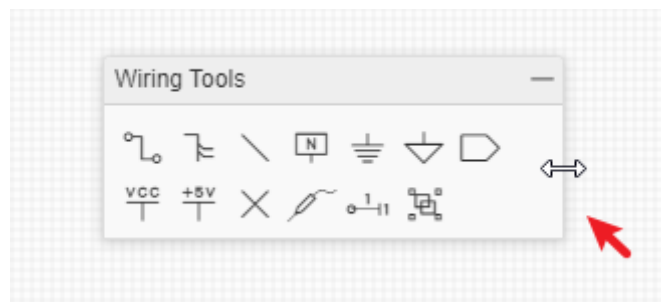
**Top Menu > View > Preview Window.**



- 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.

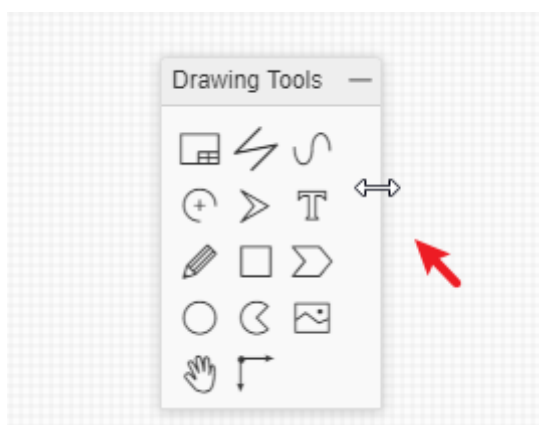
## Wiring Tools

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



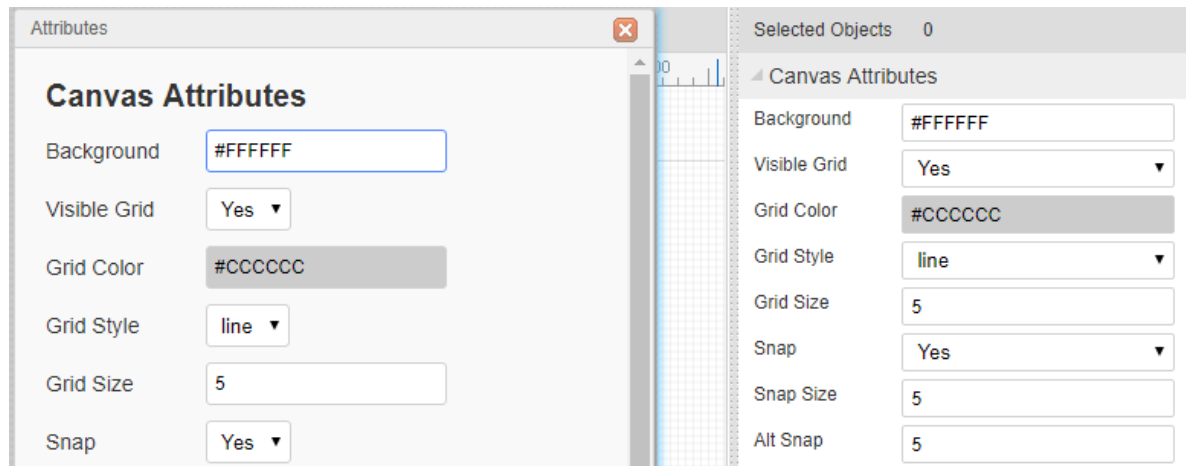
## Drawing Tools

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.

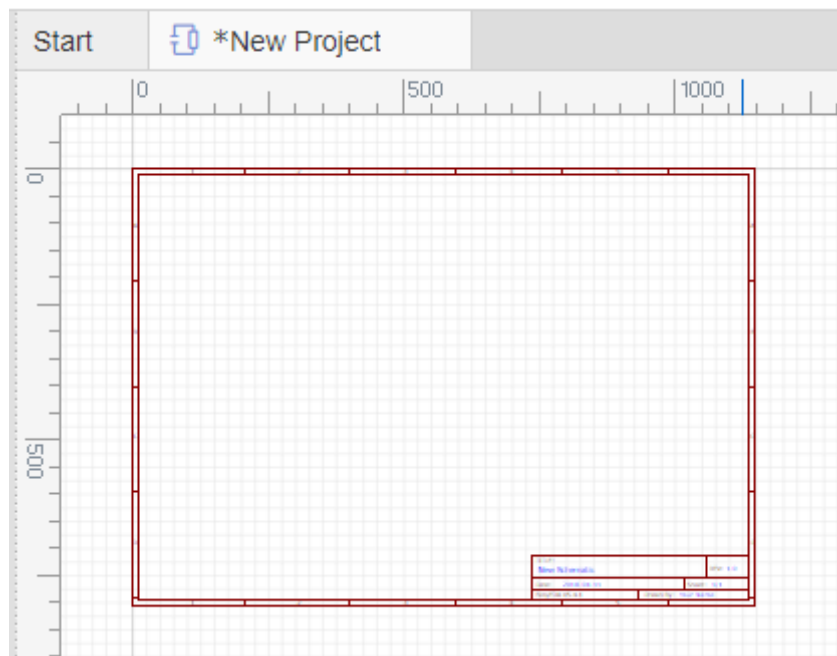


The background and grid colors and the style, size, visibility and snap attributes of the grid can all be configured.

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.



---

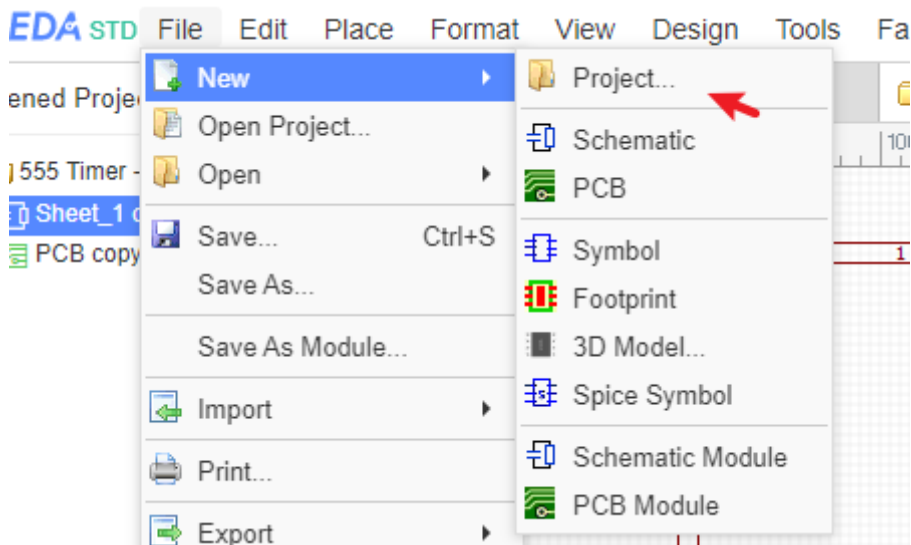
## How to Create a New Project or File

---

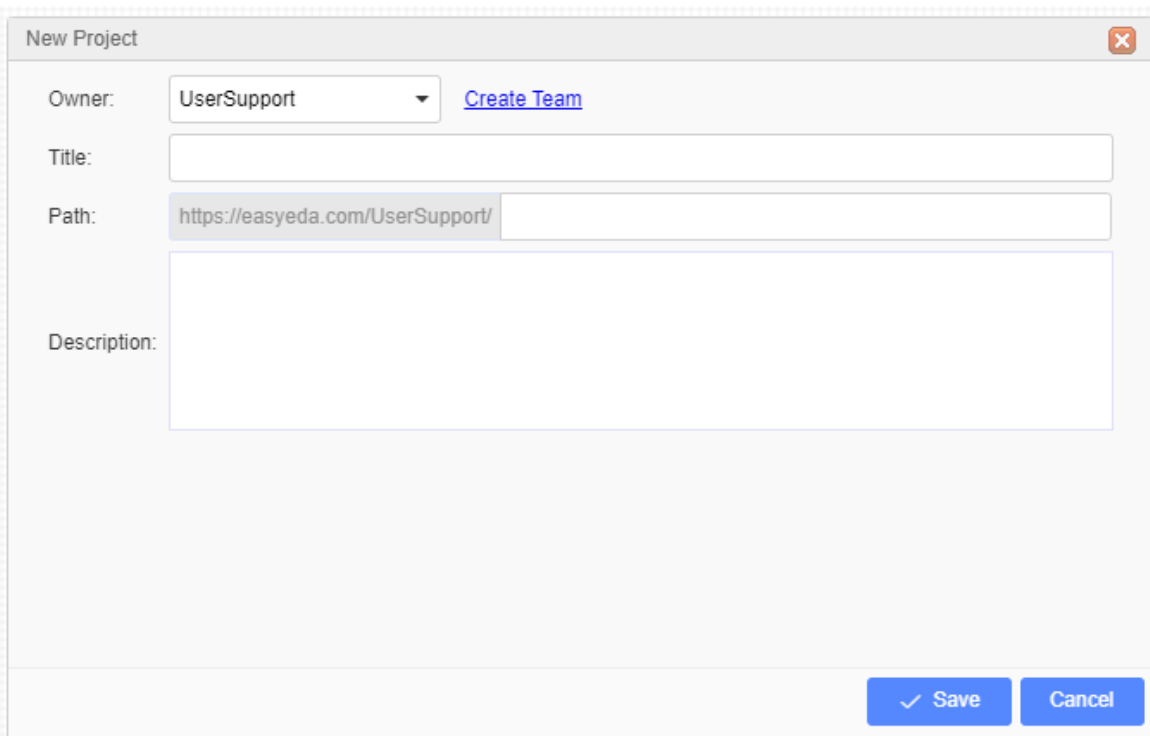
# Create Project

After logging in, you can create a new project:

**File > New > Create a new project/Schematic..etc**

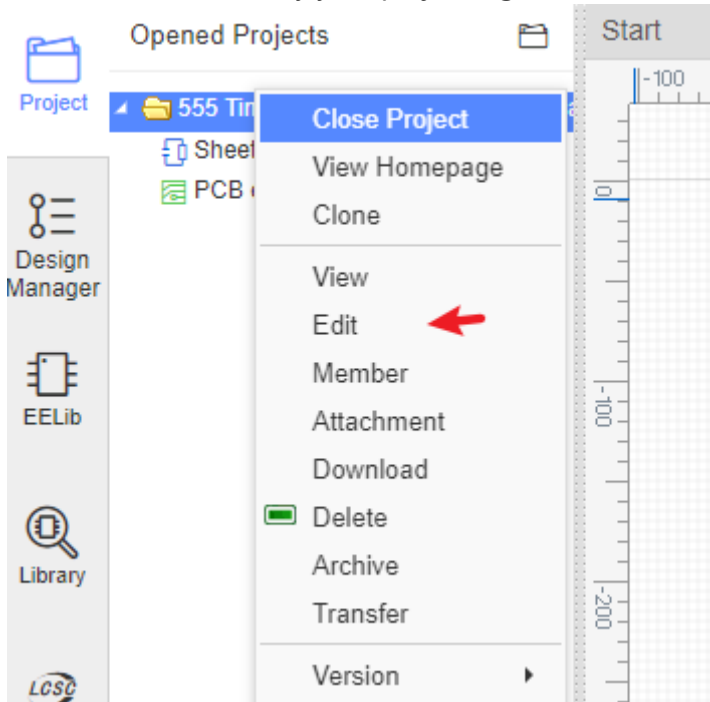


The Project concept is important in EasyEDA because it is the foundation of organizing your designs.

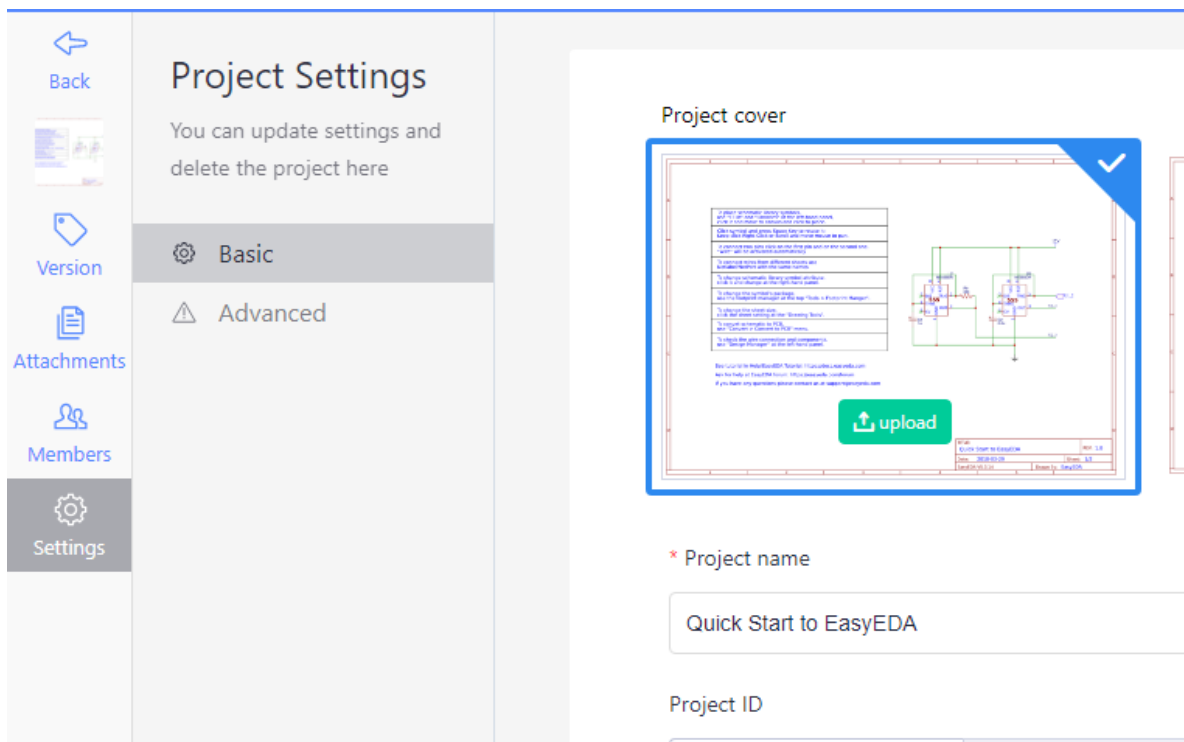
A screenshot of the 'New Project' dialog box in EasyEDA. The dialog has a title bar with a close button. It contains several fields: 'Owner' with a dropdown menu set to 'UserSupport' and a 'Create Team' link; 'Title' with an empty text box; 'Path' with a text box containing 'https://easyeda.com/UserSupport/'; and 'Description' with a large empty text area. At the bottom right, there are 'Save' and 'Cancel' buttons.

- **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:

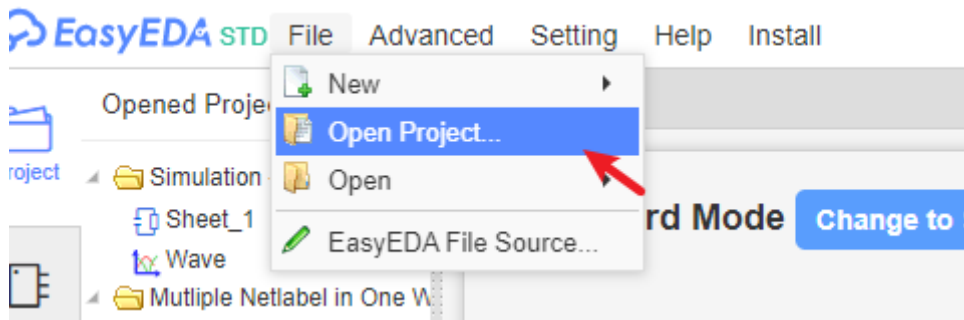


From here, you can change to publish or not, allow other people to comment on your project, or type a more detailed description of the project content. To help you make your project stand out or to maybe simply make a detailed description of your project easier to read, you can use Markdown syntax.

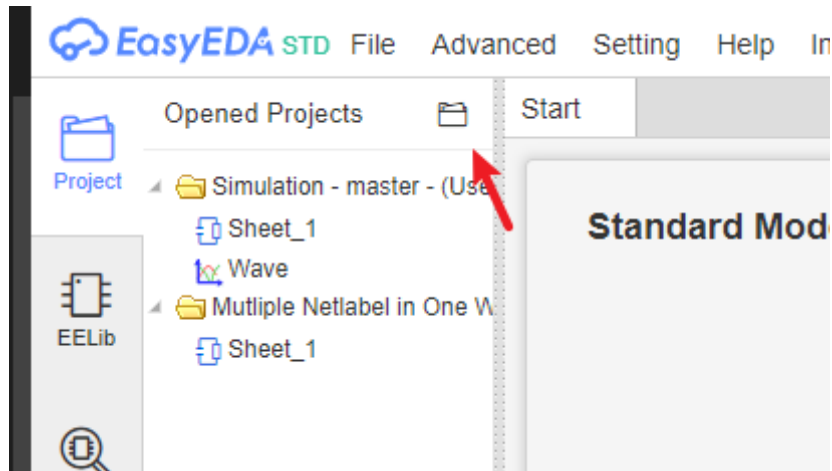
## Open Project

You can open your created project using:

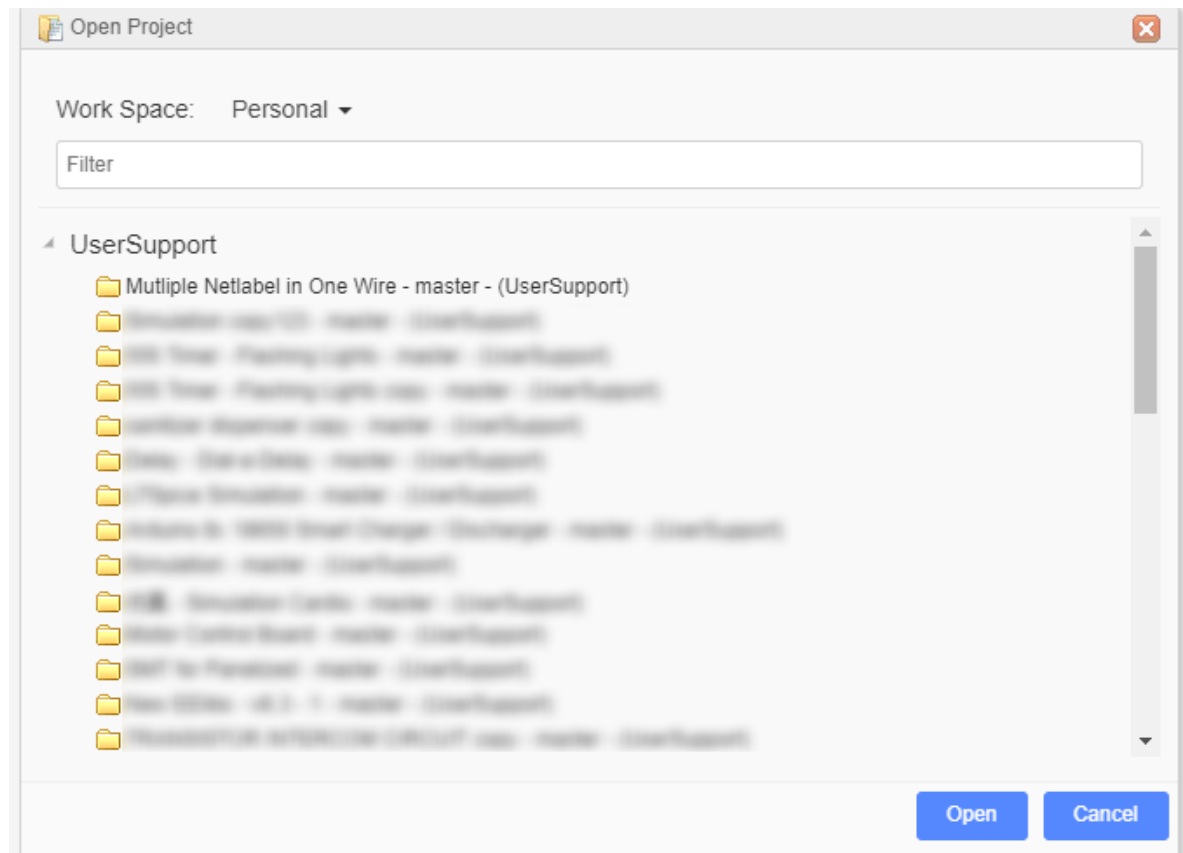
Top Menu - File - Open Project



Or click the Opened Project "open project" icon.



Select the project and open it.



## Schematic Capture

EasyEDA can create professional quality schematics.