- At present, the EasyEDA PCB fabrication and PCB Assembly service is transferred to JLCPCB. Although EasyEDA is part of the same company group, for any PCB order problems please contact with JLCPCB using the email address above.

Parts Order Problems:

- support@lcsc.com

- EasyEDA provides direct links to thousands of components which can be ordered directly from LCSC. For any problems with LCSC parts orders, please contact LCSC using the email address above.

All Other Inquiries About EasyEDA:

- Tutorials: EasyEDA tutorial
- User forum: EasyEDA forum
- If you find a problem whilst using EasyEDA please first post to the forum to ask for help. If the issue cannot be resolved there, please download and attach the EasyEDA source file of your Schematic, PCB or Project together with a clear, step-by-step description of how to repeat the issue to:

  support@easyeda.com

Notice

EasyEDA team may not have the time or resources to help you fix all your problems; we may just be able to help you to fix problems commonly encountered by newbies, such as using a drawing polyline in place of a wire, finding a spice model for a simulation or selecting the right PCB footprint.

- Please note that although some browsers or plug-ins allow you to use gestures, EasyEDA does not work with gestures, so you should disable this function.
- Simultaneous editing is not yet fully supported: care must be taken because the last save by any collaborator overwrites all previous saves.
- When signing up for an account with EasyEDA, please take a few moments to think about your username because this is the name that other users will see on your designs and posts if you choose to share them or make them public. Once you have created an account, you cannot change your username.
- You can use upper and lower case letters, numbers and symbols to make a strong password but please note that the password entry is case sensitive.
- PCBs ordered directly from an EasyEDA project are passed on to and fullfilled by, JLCPCB.

Business Development/Cooperation About EasyEDA:

please contact

dillon@easyeda.com

Address:

- F5, Tianjian Building, No.7 Shangbao Road, Futian District, Shenzhen, Guangdong, 518000, China

Introduction

Introduction to EasyEDA

Welcome to EasyEDA, a great web based EDA(Electronics Design Automation) tool for electronics engineers, educators, students, makers and enthusiasts.

There is no need to install any software. Just open EasyEDA in any HTML5 capable, standards compliant web browser.

Whether you are using Linux, Mac or Windows, it is highly recommended to use Chrome or Firefox as your browser. You can also download EasyEDA client.
EasyEDA has all the features you expect and need to take your design rapidly and easily from conception through to production.

EasyEDA Editor:

https://easyeda.com/editor

Instruction:

- This tutorial document will be updated as changes are made to the EasyEDA editor.

Tutorial for PDF

EasyEDA-Tutorials.pdf

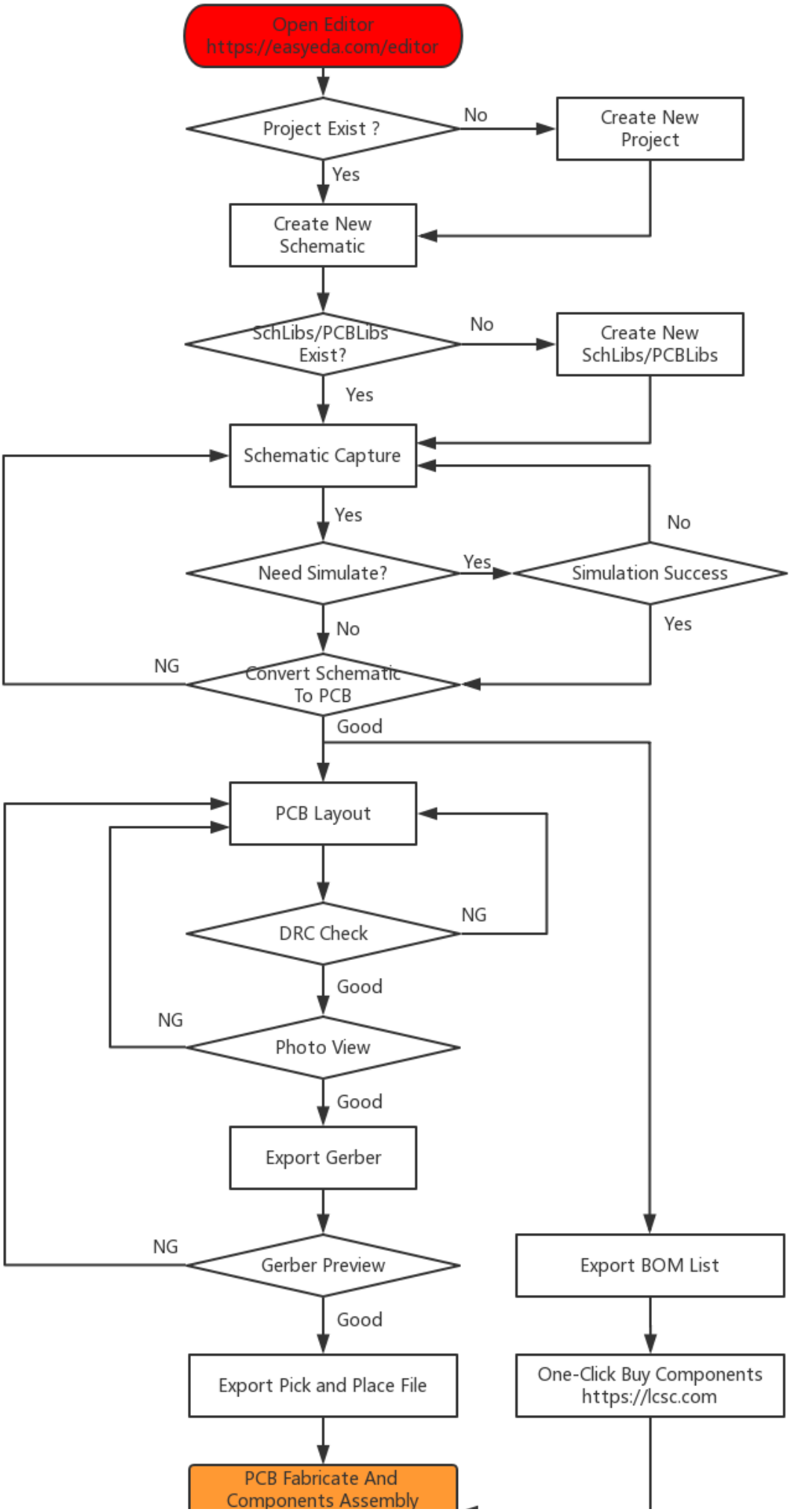
EasyEDA Provides:

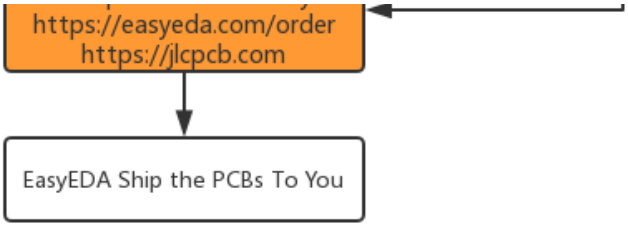
- Simple, Easy, Friendly, and Powerful drawing capabilities
- Works Anywhere, Anytime, on Any Device
- Real-time Team Cooperation
- Sharing Online
- Thousands of open-source projects
- Integrated PCB fabrication and Components purchase workflow
- API provided
- Script support
- Schematic Capture
    - LTSpice-based Simulation
    - Spice models and subcircuits create
    - Waveform viewer and data export(CSV)
    - Netlist export(Spice, Protel/Altium Designer, Pads, FreePCB)
    - Document export(PDF, PNG, SVG)
    - EasyEDA source file export(json)
    - Altium Designer format export
    - BOM export
    - Multi-sheet schematics

    - Schematic module
    - Theme setting
    - Document recovery
- PCB Layout
    - Design Rules Checking(DRC)
    - Multi-Layer, 6 copper layer supported
    - Document export(PDF, PNG, SVG)
    - EasyEDA source file export(json)
    - Altium Designer format export
    - BOM export
    - DXF export
    - Photo view
    - 3D View
    - Generate fabrication file(Gerber)
    - Export Pick and Place file
    - Auto Router
    - PCB module
    - Document recovery
- Import
    - Altium/ProtelDXP ASCII Schematic/PCB
    - Eagle Schematic/PCB/Libraries
    - KiCAD Schematic/PCB/Libraries
    - DXF
- Libraries
    - More than 1,000,000 public Libraries(Symbol and Footprint)
    - Library management
    - Symbol/Subpart create and edit
    - Spice symbol/model create and edit
    - Libraries management
    - Footprint create and edit

Design Flow by Using EasyEDA

You can design circuits easily using EasyEDA. The design flow is as shown here:


Open Editor
https://easyeda.com/editor
Project Exist ?
No
Create New Project
Yes
Create New Schematic
SchLibs/PCBLibs Exist?
No
Create New SchLibs/PCBLibs
Yes
Schematic Capture
Yes
No
Need Simulate?
Yes
Simulation Success
No
Yes
NG
Convert Schematic To PCB
Good
PCB Layout
DRC Check
NG
Good
NG
Photo View
Good
Export Gerber
NG
Gerber Preview
Good
Export Pick and Place File
PCB Fabricate And Components Assembly
Export BOM List
One-Click Buy Components
https://lcsc.com

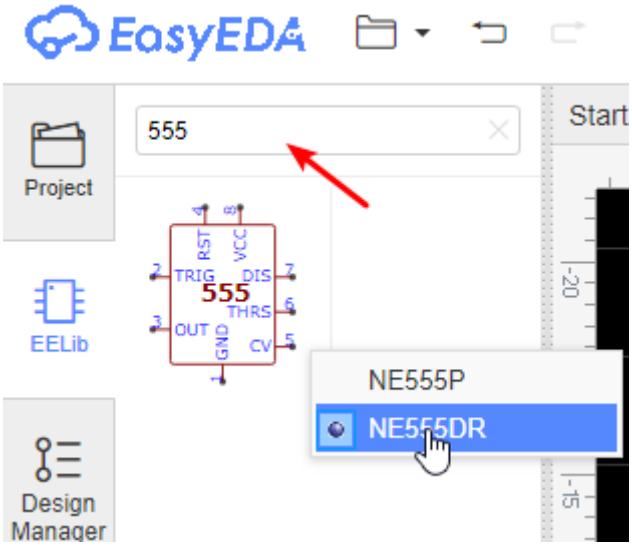


UI Introduction

EasyEDA Editor has a clear and friendly user interface. It has a short learning curve, and you will be productive in a short time.

Filter

To use the filter first select what module you need in the left navigation panel, and then you can find projects, files, parts, and footprints quickly and easily just by typing a few letters. For example, if you want to find all files containing “NE555” in the title, just type “555”. The filter is non-case sensitive.



The Filter can only find projects, files, part titles, and names. It does not search the Descriptions and Content fields.

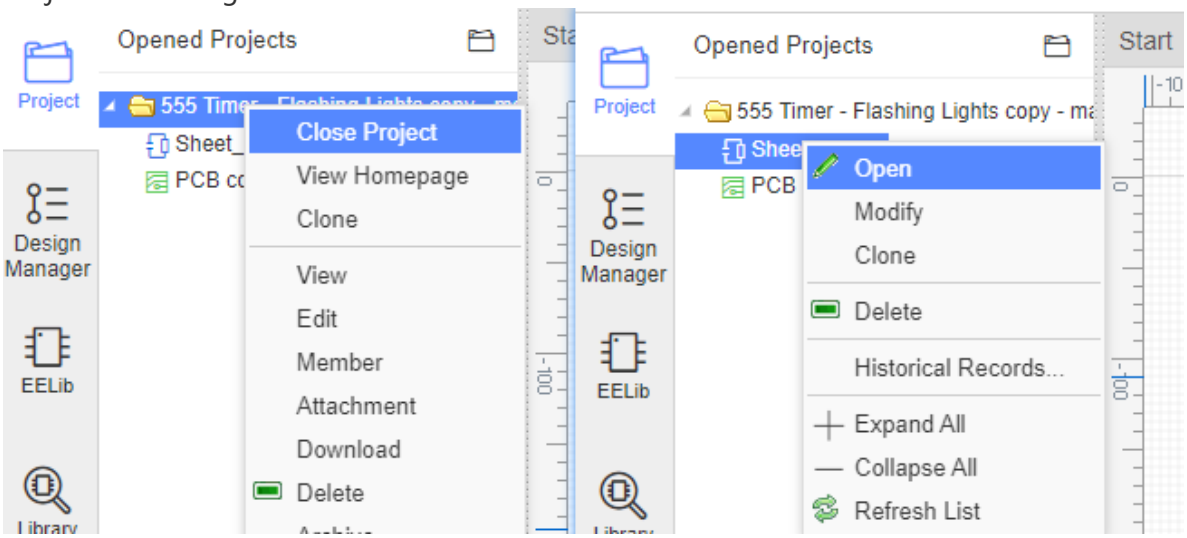
Click the X to clear the filter.

Navigation Panel

The Navigation panel is especially important for EasyEDA: This is where you can find all of your projects, files, parts and footprints.

Project

Here you can find all your projects, both private or shared with the public, or fork them from someone else's. These options have a context menu found by right clicking and selecting Projects. You will get a menu tree like:

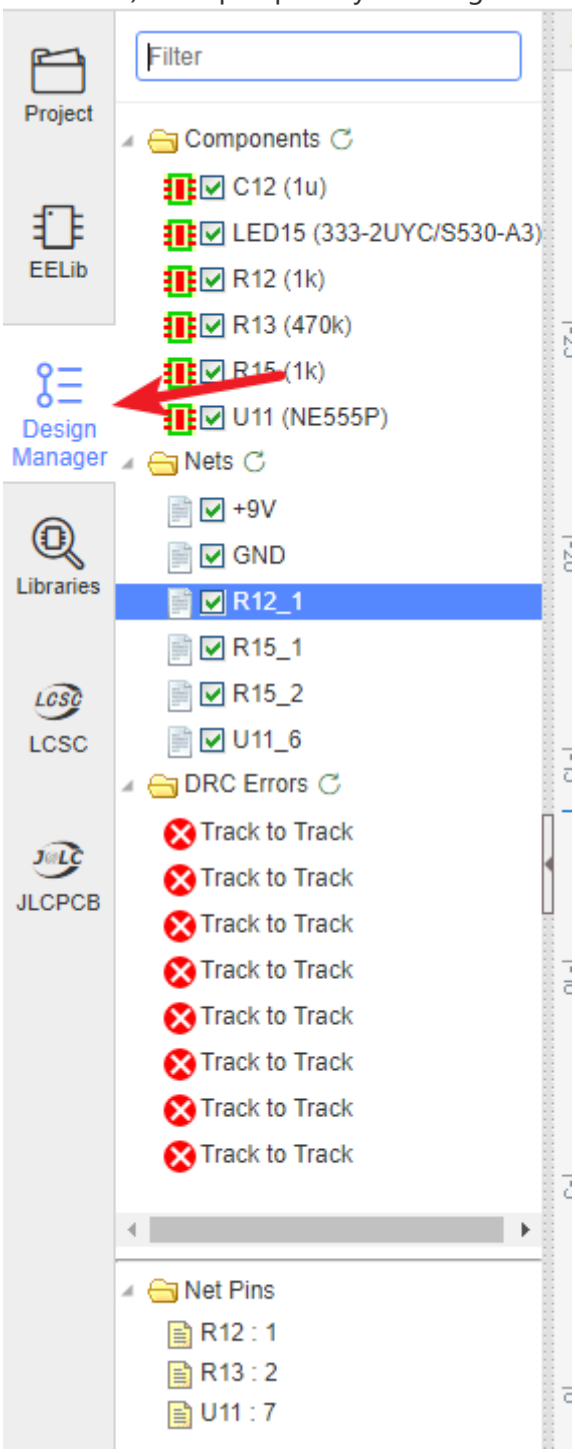


EELib

EElib means EasyEDA Libraries, it provides lots of components complete with simulation models, many of which have been developed for EasyEDA to make your simulation experience easier.

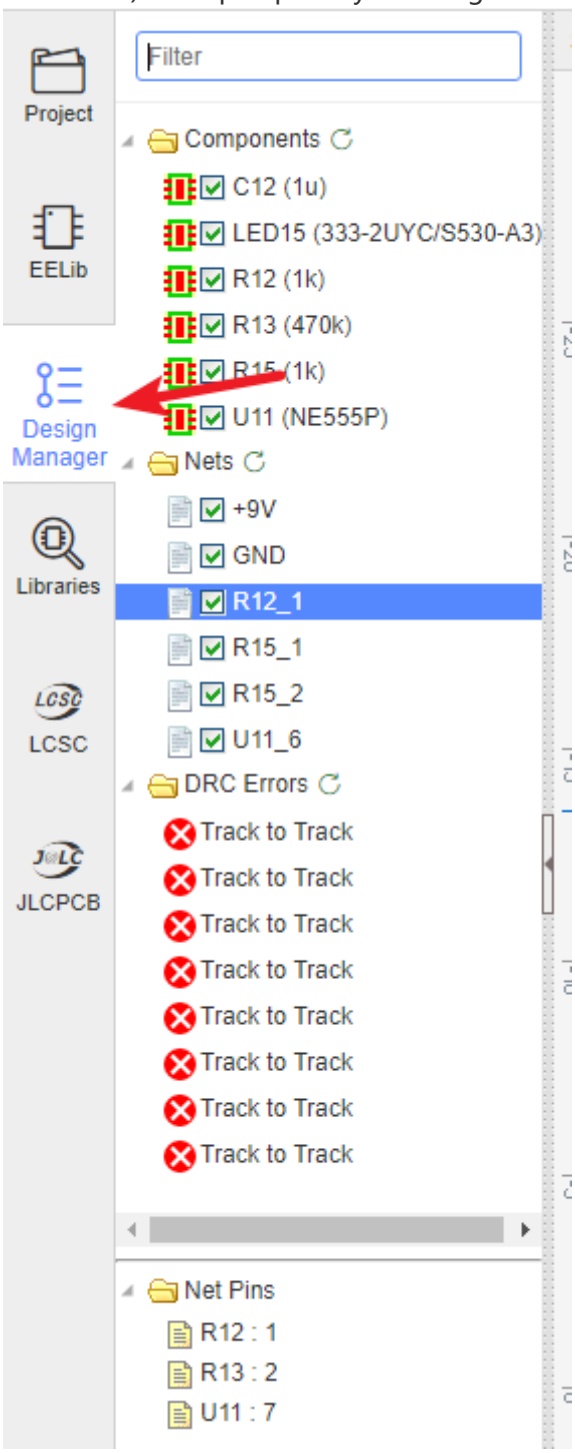
Design Manager

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



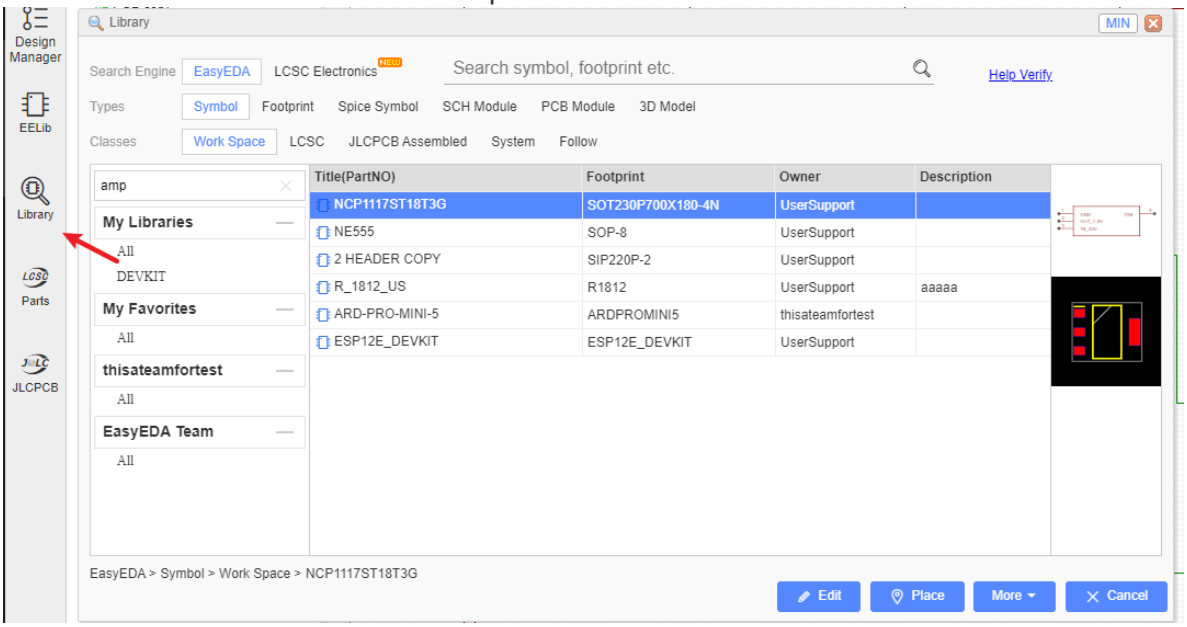
Library

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



Library

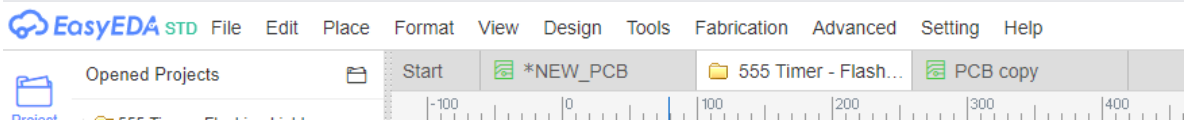
Contains schematic symbols and PCB footprints for many available components and projects. Your own libs and modules will show up here.



- **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 ae 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:



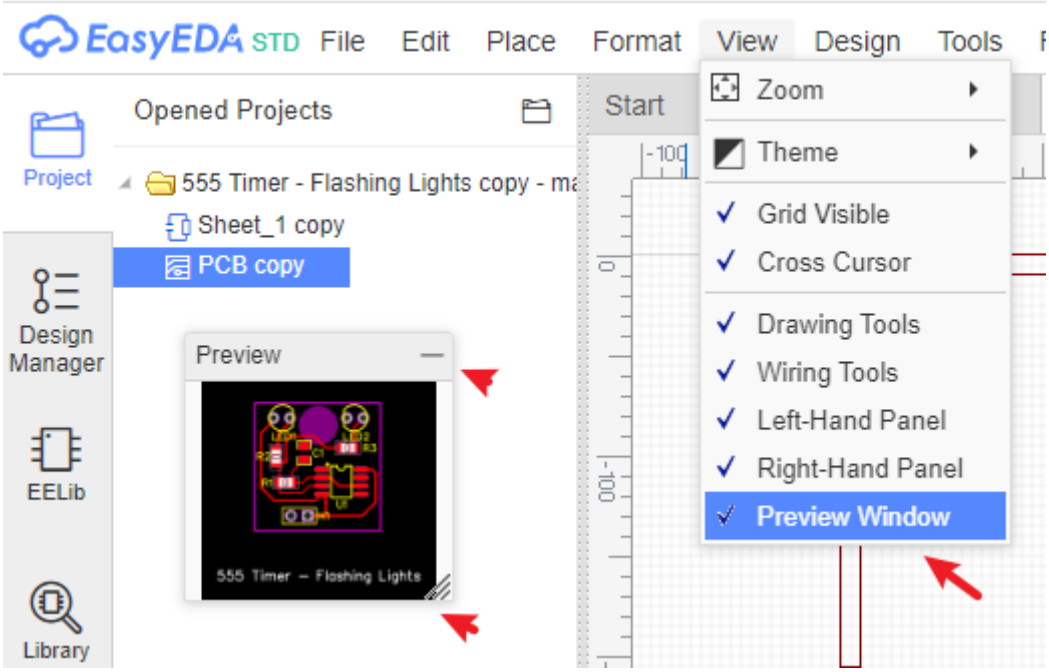
You can find what you need easily and clearly.

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:

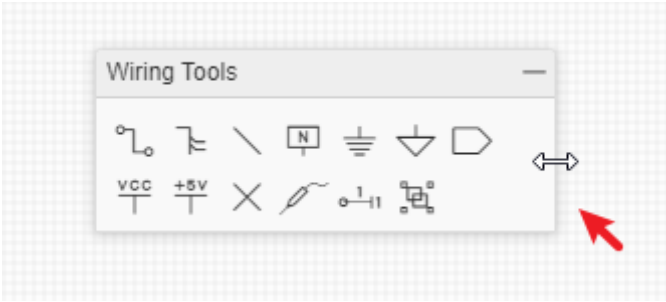
Top Menu > View > Preview Window.



- 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.

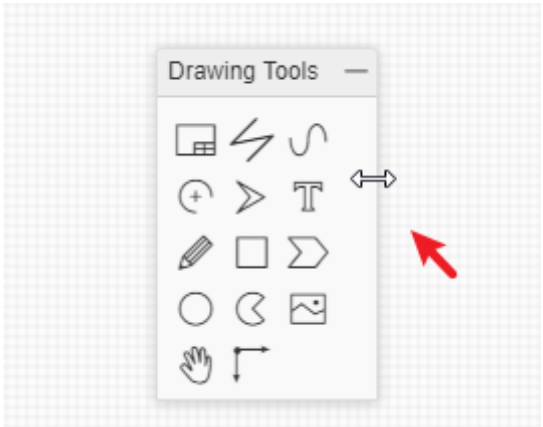
Wiring Tools

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



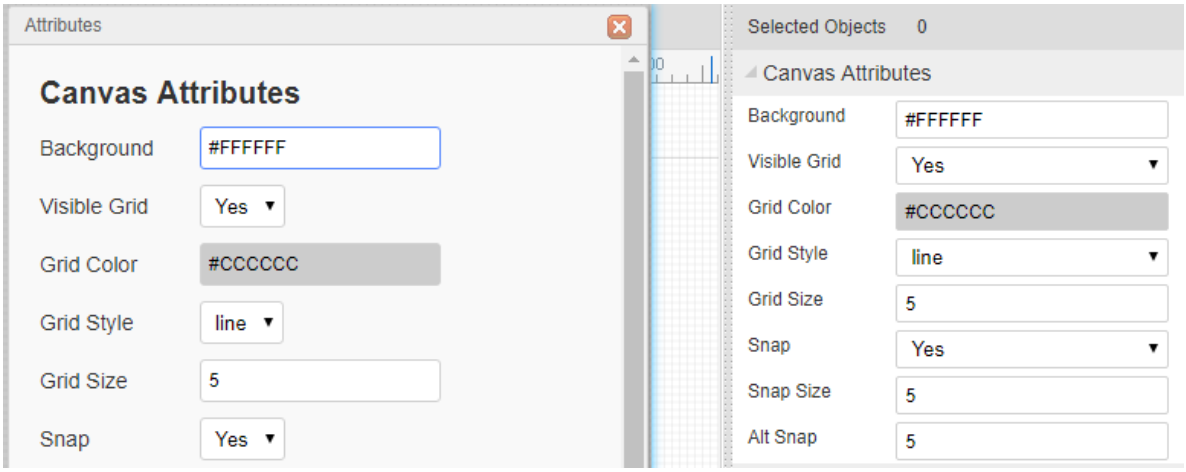
Drawing Tools

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.

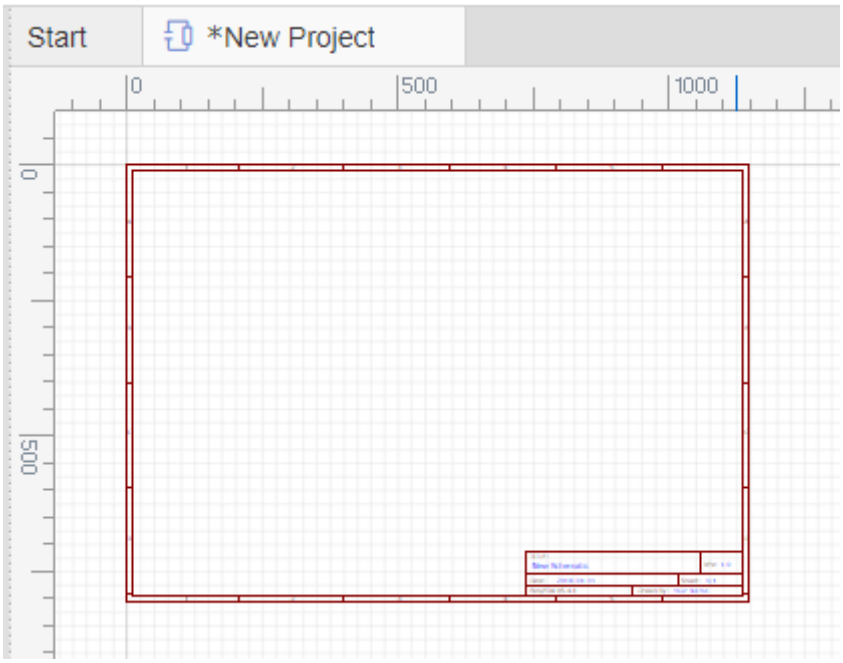


The background and grid colors and the style, size, visibility and snap attributes of the grid can all be configured.

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