

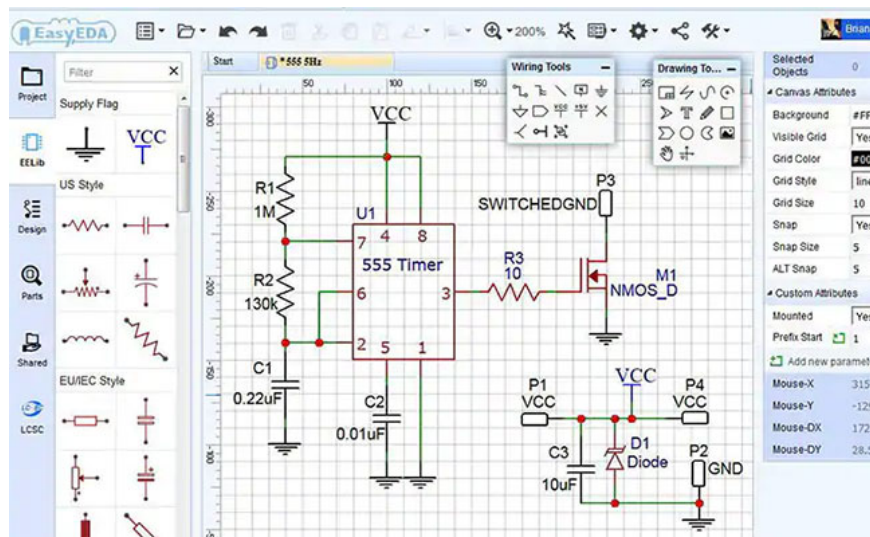
Top best electronic circuit drawing software

This article lists all the software available for drawing circuits. There are both free and paid software for you to choose from according to your needs. Let's find out more details through the following article!

List of free circuit drawing software

First let's list some free electrical schematic drawing software.

1. EasyEda



As the world around us moves to the cloud, so too do some powerful EDA tools available on the cloud. EasyEda is one such tool, facilitating schematic drawing, spice simulation (an open source, general-purpose analog circuit simulator), and PCB layout.

EasyEDA is an easy and powerful online PCB design tool that allows electronics engineers, educators, students, makers or enthusiasts to design and share their own projects. It is a design tool that integrates LCSC component catalog and JLCPCB PCB service to help users save time turning their ideas into real products.

The program has more than 70,000 diagrams available in its web database of more than 15,000 Pspice libraries. Since the application is built on the cloud, it offers the convenience of portability. Another advantage that comes with cloud applications is cross-device compatibility.

EasyEDA's outstanding features include:

1. Simple, easy, user-friendly and powerful general drawing capabilities

2. PCB layout
3. Work anywhere, anytime, any device
4. Real-time team collaboration
5. Share online
6. Thousands of open source projects
7. Direct links to LCSC components for selection
8. Integrated PCB Manufacturing
9. Import Altium/Kicad/Eagle, PNG, DXF
10. Over 1 million free libraries
11. Provide API
12. Script support
13. Schematic capture
 1. LTSpice based simulation
 2. Waveform viewer and data export (CSV)
 3. Export netlist (Spice, Protel/Altium Designer, Pads, FreePCB)
 4. Export documents (PDF, PNG, SVG)
 5. Export EasyEDA source file (json)
 6. Export to Altium Designer format
 7. Export BOM
 8. Multi-page sitemap
 9. Module diagram
 10. Install theme
 11. Restore Document
14. PCB layout
 1. Design Rule Check (DRC)
 2. Multi-layer, support 6 copper layers
 3. Export documents (PDF, PNG, SVG)
 4. Export EasyEDA source file (json)
 5. Export to Altium Designer format
 6. DXF Export
 7. View photo
 8. 3D View
 9. Generate fabrication file (Gerber)
 10. Export Pick and Place file
 11. Automatic router
 12. PCB Module
15. Enter
 1. Altium/ProtelDXP ASCII/PCB Schematic
 2. Eagle Schematic/PCB/Libraries
 3. KiCAD/PCB Schematic/Library
 4. DXF
16. Library
 1. More than 1,000,000 public libraries
 2. Library Management
 3. Create and edit icons/subsections
 4. Create and edit models
 5. Library Management
 6. Create and edit footprints

2. PCB123

This is Sunstone's free circuit drawing software. You can do almost all your needs from design to construction using PCB 123. PCB123 offers 500,000 parts with a great search function to access them all while drawing. Datasheets of selected parts are available. The cool feature of this software is the "real-time" search to select available parts from Digi-Key. After all, this is free software from a reputable company. That's Sunstone.

PCB123 is an application designed to help engineers and those interested in designing printed circuit boards (aka PCBs). It is a CAD utility in its true form not only because of its clear and well-structured interface but also because of the practical tools it provides you.

During setup, you can include or exclude additional components, namely a software development kit for writing plugins that integrate with PCB123, as well as a plugin with an application guide template for creating PCB123 plugins in Visual Studio.

Create boards and customize PCB settings

You are greeted by a user-friendly interface on startup that can be changed. With the help of a wizard, you can assemble a new board by specifying the board name and an optional Netlist file. Multi-page Netlist files can be added, along with Protel, Tango, OrCAD and native PCB123 Netlist files.

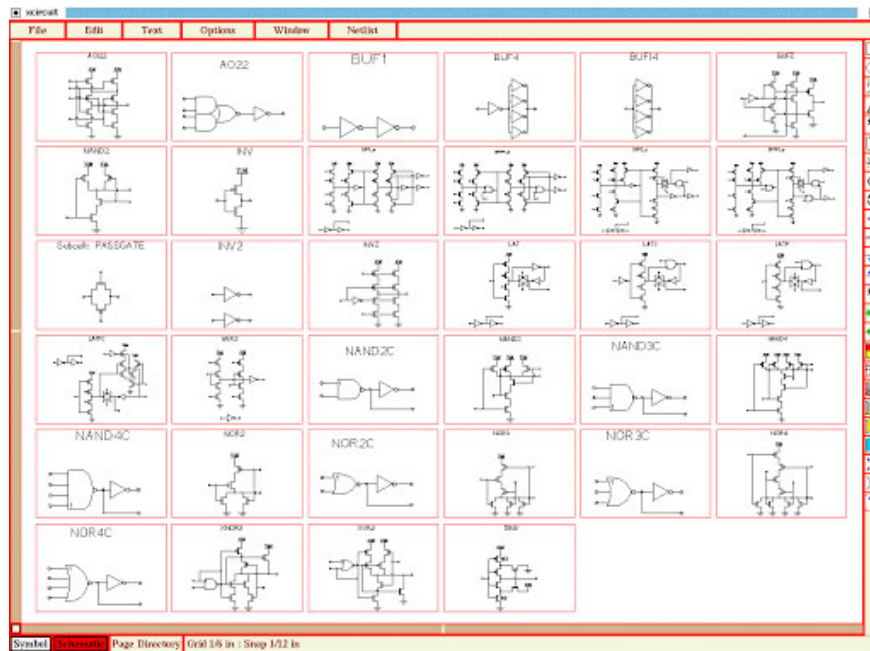
In the following steps, you can define the width and height of the board or use a template from the list, as well as set the number of layers and the type of inner layer. Additional board features focus on solder and silkscreen material, PCB thickness, copper weight, and lead-free.

Design CAD diagrams using professional-grade functions

Using PCB123, you can design multi-page schematics complete with components wiring between them and symbols. The working environment is well organized and gives you quick access to all the tools you need to work efficiently. You can view the overall layout of the board or component you are designing, explore 3D renderings, and view layers containing schematics for each component.

PCB designers can automate some tasks. For example, connecting pins from different parts will automatically create wires in between. You can also place application components on the board, minimizing connection distances while maintaining aesthetics.

3. Xcircuit



Xcircuit is a free schematic drawing software from OpenCircuit designs, created for Unix/Linux environments. You can use it on Windows, if you are running an X-server or Windows API. This open source project aims to develop high quality output schematics, ready for immediate publication. There are several versions of the software available. You should read the manual carefully before starting to use it.

4. TinyCad

TinyCad is a schematic drawing software for Windows from SourceForge. It supports circuit drawing, layout development, and circuit simulation. TinyCad is available for free download.

5. Dia

Dia is a basic drawing software suitable for drawing block diagrams. It also provides access to some basic and important components. This software is only for beginners or those who are new to the field of electronic circuit drawing. The software is GPL licensed, created for Mac and Linux.

6. SmartDraw

This free software for drawing electronic schematics is from SmartDraw LLC, a company that develops high-end drawing and CAD software. The free version is a promotional tool for their paid versions. So, you can't expect any great/advanced features in the free version.

7. ProfiCAD

ProfiCAD is a basic, free tool for drawing.

8. Solve Elec

Solve Elec is a free software for Windows and Mac.

9. CircuitMaker

A free to use schematic and PCB design tool for the Open Source Hardware community.

10. Open Circuit Design

Reference link:

<http://opencircuitdesign.com/>

This site is a repository of open source EDA (Electronic Design Automation) tools including Magic, IRSIM, Netgen, PCB, and Xcircuit. All of these tools are freely available under the GNU General Public License (GPL) or a similar open source license.

11. KiCad EDA

Open source and cross-platform electronic design automation suite.

12. ADS circuit design software

Supported operating systems: Windows

ADS is an analysis tool, which allows to extract or verify the properties of an analog or digital system.

13. nagaEDA circuit design software

Supported operating systems: Windows

naga EDA is dedicated to providing useful electronics design tools in C++ and especially Python. The current release contains naga.Verilog, a Verilog parser

14. OpenSce circuit design software

Supported operating systems: Windows

OpenSce is an open source software with an attractive and efficient GUI that allows the design of linear electronic circuits and the description of existing electronic circuits.

15. QSapecNG circuit design software

Supported operating systems: Windows

QSapecNG is a Qt-based symbolic analyzer for linear analog circuits. It actually consists of two separate parts: the SapecNG framework tool and the QsapecNG gui application.

16. Simulide

Supported operating systems: Windows

Simulide is a circuit simulation software that allows you to test your circuit before prototyping it.

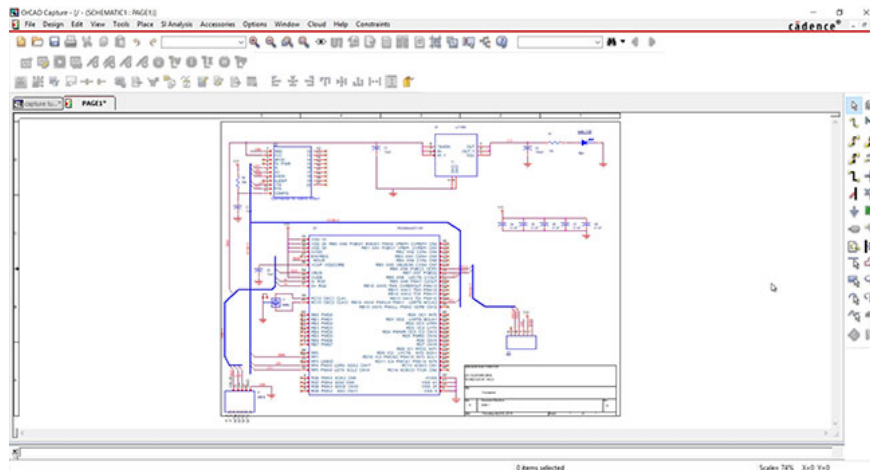
17. gerbv

gerbv is an open source Gerber file viewer for RS-274X only.

Paid circuit drawing software

Tina - Tina is a decent solution for small industries and freelancers. It facilitates circuit drawing, layout development, simulation and other features. Real-time circuit testing is another notable feature.

Orcad - A very popular software from Cadence for circuit drawing, layout development and simulation.



CadSoft Eagle - Another high quality PCB design software. EAGLE stands for Easily Applicable Graphical Layout Editor.

LTSpice - Is a simulation software by Linear. Schematic development, spice simulation, waveform viewer and many more features.

PCB Pool - This link is essentially a list of software for PCB design.

You finished reading the article "**Top best electronic circuit drawing software**" edited by the [TipsMake](#) team. We hope this article has provided you with many useful tech tips and tricks. You can search for similar articles on tips and guides. Thank you for reading and for following us regularly.