

- <u>Labs</u>
- Q&A
- quizzes
- Comments
- Contact
- Participate!

Register

Login

- Arduino
- 555
- Alarms
- Audio
- DIY
- Hobby
- Lights
- Power supply
- Radio
- Solar
- Tested
- Tools »
- Tutorials »
- Various »



MOUSER

ELECTRONICS Search 4 million + Products

▼

<u>View Product Index</u> | <u>Close</u> ❖



Browse Products

- Capacitors
- Resistors
- Input Devices
- <u>Transformers</u>
- Temperature Sensors
- Encoders
- Motors & Drives
- Liquid Sensors
- Current Sensors
- Timers
- Potentiometers & Trimmers
- Batteries
- Proximity Sensors
- 7 Segment Displays
- Relays
- Connectors
- Pressure Sensors

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish. Accept Read More

- Integrated Circuits ICs
- DC/DC Converters
- AC Powers Cords
- LED Lightings
- Multimeters & Voltmeters
- RF Semiconductors
- Processors & Controllers
- Oscilloscopes
- Counters & Techometers
- Wire & Cable



Best of Free PCB Design Software

Posted by P. Marian in Useful with 73 comments

Tagged with: electronics software pcb

G+1 9 Tweet Ask a question Like Share 683

+ this message might be for you!

Do you need a free PCB design software or tool to put in practice the new electronic project you've just designed? So, we present you 10 of the best PCB software tools available on the internet that will help you to develop your printed circuit board faster and with ease. See the list under these ads.



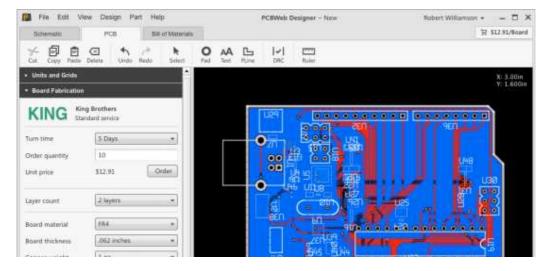
0 : PCBWeb Designer

PCBWeb is a free CAD application for designing and manufacturing electronics hardware. Design multi-sheet schematics with our fast and easy-to-use wiring tool. Route multi-layer boards with support for copper pours and DRC checking. Integrated Digi-Key Parts Catalog with Bill of Materials Manager.

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

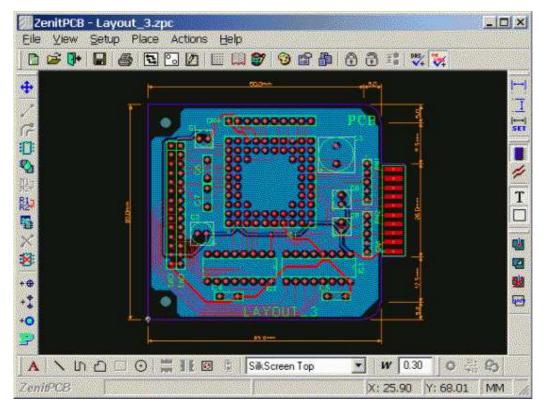
Read More

Read More



1 : ZenitPCB

Is an excellent pcb layout design software tool to create professional printed circuit board (PCB). It is a flexible easy to use CAD program, which allow you to realize your projects in a short time. With ZenitPCB Layout is possible to create the project starting both from the schematic capture or by the layout itself.



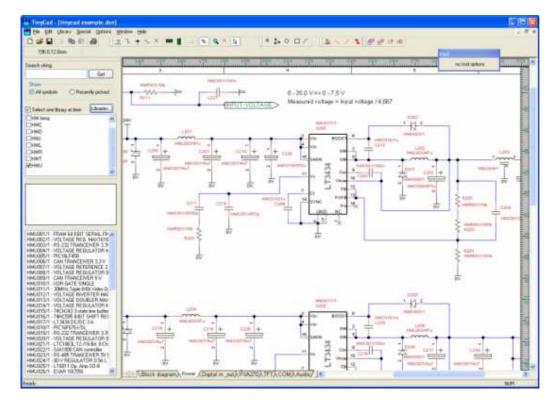
2 : TinyCAD

Is a program to help you draw circuit diagrams. It comes complete with symbol libraries to get you started straight away. As well as being able to simply print your designs, you can use TinyCAD to publish your drawings by copying and pasting into a Word document or saving as a PNG bitmap for the web.

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

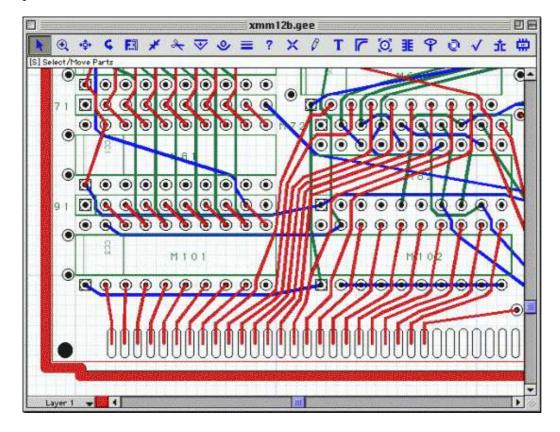
Read More

Accept



3 ∴ Osmond PCB

Is a flexible tool for designing printed circuit boards. It runs on Macintosh. Its many features include: virtually unlimited board sizes, number of board layers, number of parts, support for both through-hole and surface mount parts and more.



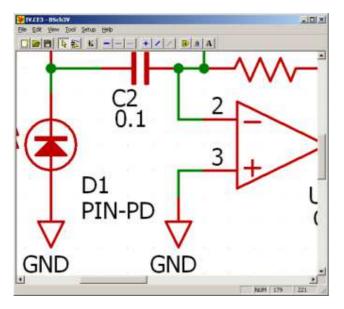
4 ∴ **BSch3V**

Is a schematic capture program for Windows. The name "BSch" is an abbreviation of "Basic Schematic". It has only basic functions, in order to simplify operation.

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

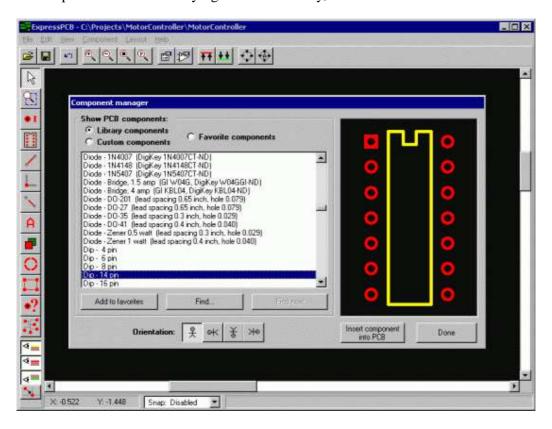
Read More

Read More



5 ∴ ExpressPCB

Is a snap to learn and use. Laying out PCBs is easy, even for the first time user.



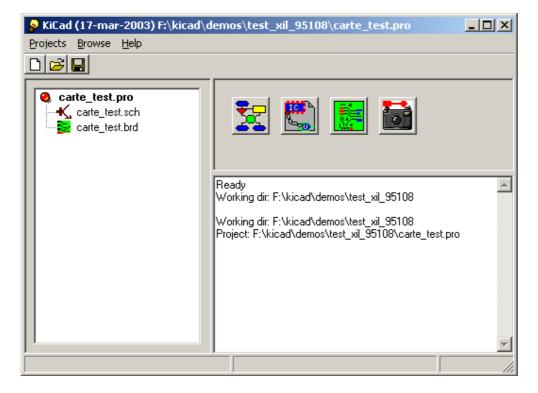
6 ∴ **Kicad**

Is an open source (GPL) software for the creation of electronic schematic diagrams and printed circuit board artwork. It is useful for everybody working in electronic design (schematic diagrams and Printed Board up to 16 layers).

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

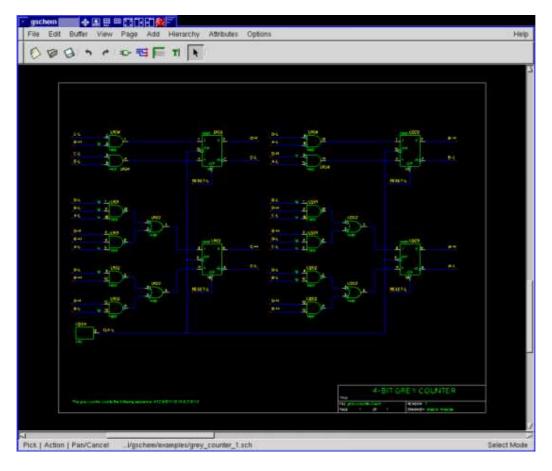
Read More

Accept



7 ∴ **gEDA**

Runs on Linux and has produced tools which are used for electrical circuit design, schematic capture, simulation, prototyping, and production. Currently, the gEDA project offers a mature suite of free software applications for electronics design, including schematic capture, attribute management, bill of materials (BOM) generation, netlisting into over 20 netlist formats, analog and digital simulation, and printed circuit board (PCB) design layout.



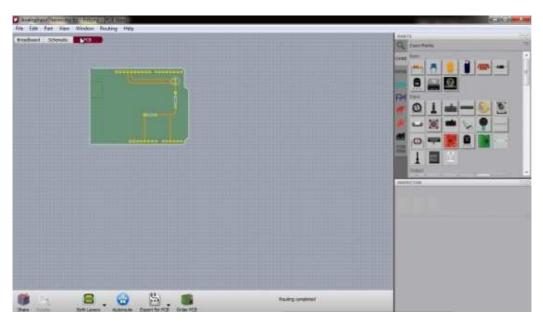
8 .: Fritzing

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

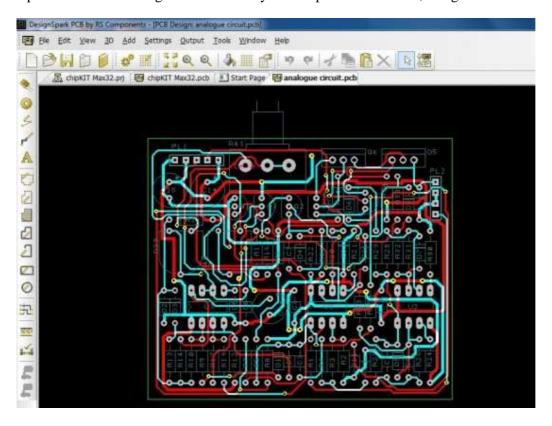
Read More

about electronic circuits, to document your projects and even let's you prepare them for production.



9 : DesignSpark PCB

DesignSpark PCB is the world's most accessible electronics design software. Easy to easy to learn and easy to use, it is designed to significantly reduce your concept-to-production time. At the core of this unique approach is a powerful software engine that enables you to capture Schematics, design PCB boards and layouts.



10 ∴ EasyEDA

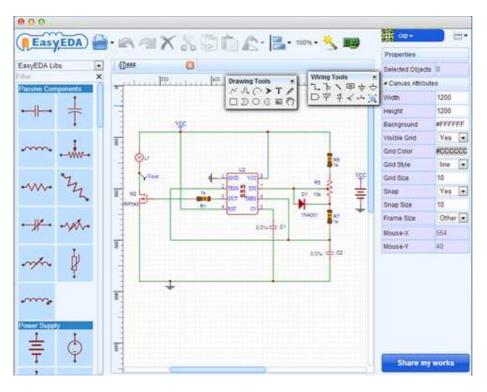
EasyEDA is a free, zero-install, Web and cloud-based EDA tool suite, integrating powerful schematic capture, mixed-mode circuit simulation and PCB layout in a seamless cross-platform browser environment, for electronic engineers, educators, students and hobbyists.

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

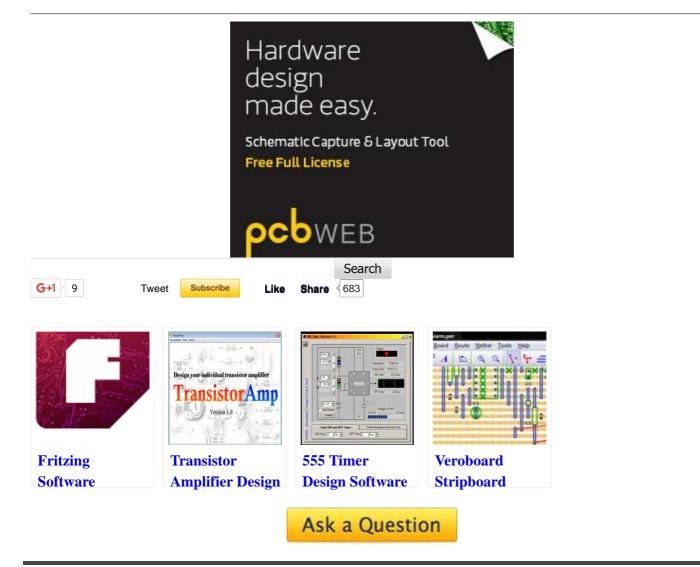
Read More

Read More

7 of 22



Please feel free to say your opinion about these applications or tell us what you are using and why do you think it is better than the rest of apps.



This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

1. Christo Candiotes says: on September 20, 2011 at 3:51 am
Log in to Reply

Another great PCB design program is KiCAD. Cross platform too.

Sal Christy says: on August 19, 2014 at 11:41 pm reply

Sadly it is very unstable on Linux. Constantly develops errors when designing PCBs. The designers just don't have it together.

qwerty says: on September 10, 2015 at 5:44 am

Not when I have used it on freebsd 6,7,8 and now 9. It was a glitch in some version but I have not found any in the later ones.

2. Circuito says: on March 5, 2014 at 2:34 am
Log in to Reply

I use DesignSpark PCB ver6.0 for my electronic design project. It's a free Schematic PCB software from RS Component.

P. Marian says: on March 5, 2014 at 9:05 am reply

Thank you for your suggestion!

Usman Ul Haq says: on June 3, 2014 at 11:07 pm reply

can you send me the link from where i could download it!!!!

PrashanthSuvarna says: on August 13, 2015 at 6:50 pm reply

Here is one more software that i happene to pick up (one transistor.blogspot.in) somewhere. Figured it may be useful to some of you out there you can follow this link... http://onetransistor.blogspot.in/2014/10/draw-electronic-schematics-using.html

Christian Ressel says: on March 26, 2014 at 6:52 am
Log in to Reply

How do you rate SoloPCB?

It can be seen at

http://www.fabstream.com/solopcb-design-software.php

tp.//www.itaostream.com/solopeo design soltware.

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Read More

hellow i will help to pcb edit mobile phone

4. Eng. Ahmad AL-FaiFy "Makkah" says: on April 21, 2014 at 3:36 pm
Log in to Reply

Fritzing software is an interesting open-source "GOOD"

and Proteus its good >>> and i will try DesignSpark PCB.

thx a lot

5. ericaleo says: on April 26, 2014 at 2:23 pm
Log in to Reply

You may try pcbmaking.com to get your job done quickly and effectively. It gives big discount on various PCB and also FREE shipping.

6. AJ says: on May 9, 2014 at 8:29 pm
Log in to Reply

This is some great info that i would like to share with our customers who are looking for good source for PCB Design tools.

If you need a source for Quick turn prototype PCBs or assembly, check this website. http://www.Pcb4assembly.com

7. Hawa Singh Sharma says: on May 14, 2014 at 6:44 pm
Log in to Reply

Please tell me that "How can be downloaded these PCB designing softwares?"

8. ** rasoul_najafkhani_iran_qazvin says: on May 16, 2014 at 5:16 pm

Log in to Reply

only altium designer && ads

o Naqash Ahmad says: on November 8, 2015 at 1:10 pm reply

yes boss altium desiner is the best pcb desing tool

9. Darshak says: on May 17, 2014 at 2:50 pm
Log in to Reply

Please let me know which programming language is used in these ICs

10. Review says: on May 23, 2014 at 8:18 pm
Log in to Reply

There is no such thing as a free dinner

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Read More

10 of 22

work. Support couldn't solve it. Later I realised it required constant connection to the net with cookies enabled. I believe it is a spyware.

Fritzing is a toy.

Kickad I found to be full of bugs to the extent of uselessness. I deleted it when I found myself fighting the software rather than designing a PCB.

11. Review says: on May 23, 2014 at 9:13 pm
Log in to Reply

FreePCB wasn't maintained for the last three years – since 2011. This tells the story.

TinyCad is what the name says: tiny. The library is extremely limited. For example there are only 11 microcontrollers.

ExpressPCB is anything but express. Very difficult to use it. If you re-position a part, you have to re-align every-single-connection going to it. This means, every corner of every lines. This is perhaps the most cumbersome to use among all of these. Badly limited library.

Sadly, all of these are totally useless but for the most simple, few-pads jobs. But then one could just get a piece of vero-board.

Most of these require one to create custom components almost for everything.

Might as well use inkscape ... lot easier to make that few small boards.

There is no such thing as a free lunch ...

Good luck!

O AJ says: on June 4, 2014 at 12:46 am reply

That's absolutely true. Either these free design software are full of bugs or tie you into buying boards at higher prices by not allowing you to export gerber data files that is a standard for pcb design, if you want a choose your on pcb maker.

There is a option for having somebody else do the pcb layout for you..here http://www.pcb4assembly.com/pcb-design-and-layout/

12. prashanth suvarna says: on May 26, 2014 at 12:09 am
Log in to Reply

Yo! how come you guys havnt mentioned Eagle and solo PCB amongst others...? there may be others 2!(:-[

cavejug says: on July 21, 2015 at 1:15 pm reply

Because this article refers to "free" software. Eagle is not free.

qwerty says: on September 10, 2015 at 5:41 am

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Read More



The best for me in PCB design is Eagle Cadsoft, but there is not simulation & too much libraries. Circuit Wizard is very intuitive to check simple problems in simulation. Proteus is excellent. I like WinQcad of MicroCad, but not so practical as Circuit Wizard, PCB Wizard & Live wire. Multisim is very good. I've never used Altium Designer. I think that each one of us have to start walking the first step trying at less one of these PCB software and then we will know with wich one of them we fell ok according our requeriments.

```
14. Tadesse says: on August 6, 2014 at 2:30 pm
Log in to Reply
```

how can i find faulty component identifier

15. Eda says: on August 9, 2014 at 10:49 pm
Log in to Reply

I am working with Altium Designer 14 and find myself hating it. If you decide (for very good reasons) to use the metric system instead of that damn idiotic imperial system from the days of the Roman Empire, you are in hell with this software.

Jim Keith says: on August 10, 2014 at 1:57 am reply

Unless you grew up with the imperial system like I did here in the US-and we continue to use it. In engineering school and industry, we were forced to use the metric system.

That is one of the details that I like about expresspcb.com software—it uses either system with ease.

o Phil says: on October 22, 2014 at 4:27 am reply

In Altium, push the "Q" key to switch between Metric and Imperial. It really is that easy. \bigcirc

Every cad piece of software has issues. None of them are going to be everything you want. You are not the only cad person out there to use the tools and have ideas and suggestions on how to best make these tools. Why do you think there are so many small free cad tools. Lots of people want to make the software do what and how they want. Maybe someday, someone (maybe you) will come up with a very good tool that everyone stands behind. Until then, I wish you luck in finding / creating the cad software that fits your style.

Kind regards,



It sounds to me that you are not fimilar with measurement systems, and do not know how to convert metric to imperial and vice versa. learn to convert measurements before you condem this software,

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

Why bother to learn the Imperial system when only 3 countries in the world use it? The USA is officially Metric but the diehards still stick to a mishmash of a system along with such great countries as MyanMar (aka Burma) and Liberia, ah well c'est la vie!



Well said Phil

ron says: on September 10, 2015 at 5:00 pm reply

Not to mention the USA is the most productive, innovative, prosperous country that ever has existed!

Heinz says: on January 2, 2016 at 11:20 am reply

That imperial system was the basis of all electronics component lead spacing and was very easy to use. The metric system has to convert the standard imperial lead spacing into the metric equivalents, hence the difficulty. BTW, the imperial system is still being used in the USA and for very good reason, I grew up with it and it find relatively easy to use. Bear in mind the Digital System needs extensive conversion to communicate with a base 10 system.

16. Bob Wareham says: on August 29, 2014 at 2:23 pm
Log in to Reply

Well it is all rubbish really I have only found Target that works for me the rest of the so called free stuff is a wast of time I have used http://ibfriedrich.com/ and that is very good and free if you are not using it to make commercial products if you are then 49 dollars for 400 pins. has anyone tried any of the pcb software on line like EASYEDA or UPVERTER both look good and are free but you are on line all the time.

Bob

17. PCBMan says: on September 1, 2014 at 3:40 am
Log in to Reply

Upverter.com is really good. It's simple and easy to use but is able to design complex boards. The best part is that you can use it from anywhere on any OS since it runs in the browser. So you need to be online to use it but I'm online 99% of the time anyways...I'm rarely working on a plane or something. It's free for open source designs and I think it's pretty cheap for private designs too.

18. Kaiser says: on October 10, 2014 at 4:06 pm
Log in to Reply

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

19. Kaiser says: on October 10, 2014 at 4:09 pm
Log in to Reply

Design complex software iz not applicable for windows_7.

20. Drew robertson says: on October 26, 2014 at 6:12 pm
Log in to Reply

DesignSpark is pretty horrible. The library structure is unfathomable and it's difficult to construct your own completely independent of the RS agenda of directing you toward their own products. That would be fine if all RS components were modelled but most (that I need) aren't. There are extensive tutorials but intuitive apps don't require tutorials. It's not intuitive, plus the talk-over is a ghastly estuary accent which grates – 'model' for instance pronounced 'modo-oww', aghhhh. The help files don't match the program and it's difficult or impossible to track down what many of the click boxes mean. And it's flaky, frequent crashes.

Problem is, in the UK there isn't a lot of choice and we don't have the convenience of progs linked to manufacturers. So if we want an export, DesignSpark is about the best (or least bad) there is. The alternative is to fork out on a decent package.

21. *Jim Rattray* says: on <u>December 1, 2014 at 1:29 pm</u>
<u>Log in to Reply</u>

Thanks Drew for letting me know about how rubbish DesignSpark is.

I downloaded the app and ran it. I was then asked to register and that's where the trouble started. There were numerous problems to many to explain. So now I know it's not worth bothering. Thanks again Drew.

Drew Robertson says: on February 22, 2015 at 7:33 pm reply

Hi Jim, just picked up your reply. One trouble with DesignSpark is RS tell you how wonderful it is and what a great favour they are bestowing upon punters. So when people get into a fankle with it they are inclined to blame themselves and think they are being stooopid or thick. Well I can assure all frustrated users that they are not to blame, it is the most deplorable piece of **** I've ever encountered. Never have I sworn so vigorously at a prog, indeed I've smashed my desk banging fists against it. Pages could be filled defining the faults, suffice to say it doesn't do *anything* properly, spend all ones time doing work-arounds. The latest version I cannot even get to load and the designers are incapable of fixing it. Incompetence personified. Unfortunately there is a dearth of decent free pcb software, even many of the paid for ones are dire. I'll keep using it 'cos my will is greater than its wilfulness and I can't justify the £wads for a professional package. All we can do is complain to RS, that'll make a big difference. Just like my incessant complaining about their atrocious web site. Civilisation is doomed.

22. Jyothi says: on December 16, 2014 at 11:47 am
Log in to Reply

Amoung all these above PCB designing software which one is the best for the schematic as well as Layout design...

o Brick says: on April 20, 2015 at 12:26 am

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Read More

/wiki/ibfwikien/index.php?title=First_steps

Grossen says: on December 3, 2015 at 7:03 pm

I used Target 3001 and I found it incredibly hard to use. The library was quite large but it included almost none of the components I regularly use and making my own components seemed rather difficult. If its better than most of the software presented here then I would rather stick to doing my PCB design in CorelDraw. It is not a CAD program so its suitable only for prototyping but I really can't put up with such hard to use programs.

23. David says: on December 23, 2014 at 4:45 pm

Log in to Reply

Yes, I too hated Design Spark, although I did not encounter problems, far too clumsy and OTT for my liking – and seems intrusive to me.

I use Proteus Lite by Labcenter at least it allows me to make my own components, it's not overpowering and lets me make my own component library should I need to.

There are many, many programs out there, the free versions are usually limited in board size, number of components or number of pins.... or riddled with bugs.

I doubt the average 'home user' would not go beyond a two PCB layer design.

o Bob Wareham says: on December 23, 2014 at 6:20 pm reply

I have tried pcb web this is now a download to your computer the database is Digi Key so not to bad the download is free to your desktop and will generate free Gerber files so you can go anywhere for boards.

24. John says: on January 5, 2015 at 4:21 pm
Log in to Reply

What do you all recommend for designing flexible printed circuits (flex) instead? I need to design a flex with some contacts on it and don't know which program is good at it. I am trying eagle but is more pcb oriented. Any suggestion is appreciated.

25. Andy Fierman says: on March 24, 2015 at 2:07 pm
Log in to Reply

I have been using EasyEDA for over a year now with great results.

There's no subscription cost to use it and in Anonymous mode you don't even have to register.

It has some rough edges but is developing very quickly with a Sandbox version for users to try out new features.

(There's a major update due from the Sandbox in a week or so that will introduce live DRC in PCB layout, the ability to add extensions and scripting and add a raft of new parts.)

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

you are not tied into buying the PCBs from EasyEDA: you can download Gerbers and send them off to any PCB fab house at no charge from EasyEDA.

It has some pretty impressive import (and export) options. For instance it can import schematic and PCB designs from Altium and Eagle. Library import from KiCad is also provided.

It can even import schematics from LTspice which gives LTspice users a really easy route into doing PCBs straight from LTspice schematics.

Although maybe not important to this discussion, EasyEDA also has good simulation support using ngspice. Many LTspice simulations can be run in ngspice with little editing effort.

Symbol and footprint library searching is simple (although searching for spice models is still a bit clunky but all devices in the default library have models already associated with them).

I've found the ability to share projects privately or publically a useful feature for collaboration.

If people are interested to try it, I'd recommend looking through the Tutorial and the Simulation eBook before diving in.

26. Deane says: on March 27, 2015 at 7:43 pm
Log in to Reply

I just got through attempting to use two programs. The first is SchemeIt from DigiKey. I created my schematic, this is a simple one with only 10 components. There is a tab at the top that says PCB but when I pressed, it just links you to a new company, PCBWeb.

So I downloaded PCBWeb software (free) and found there is NO WAY to import the SchemeIt schematic... so don't bother with that program.

Then I used PCBWeb to redraw the schematic and attempted to design the board. The program is quite new and the designer got back to me with answers when I had a problem... until I started getting dozens of strange messages in the review phase. Since then he has not got back to me so I would not bother using this program either!

So I am reading every review I can to find something that might work. So far some people like Fritzer and others hate it. Some like Design Spark and others hate it. And on and on it goes.

27. Bob Wareham says: on March 27, 2015 at 10:00 pm
Log in to Reply

I have tried so many now and not found any free software that is any good If there was one that made it easy to find parts that would help as I find this is the hardest part and so slow to find the part you want even in paid software.

Bob

28. Zvika says: on April 5, 2015 at 2:06 pm
Log in to Reply

I am recommending a toolset called "Proteus" from Labcenter Ltd.

It is not free but not expensive at all. It is a whole design environment where you can design, capture your schematic, run simulation of hw and sw together, has a very good sw IDE and professional PCB design.

-

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

16 of 22

Proteus has a nice user interface – easy to use.

But:

- 1> It crashes all the time. Whoever coded the software does not know how to write exception handlers and when and how to use them. Definitely bad coding practices re errors.
- 2> The PCB side throws away your tracks and mark them void.
- 3> They are under the impression that you can have different voltages on the same wire. 12V Supply with 555 chip no voltage regulator and now you supply line has changed from 12V to 5v just by adding a 555 chip. mmmm.
- 4. Spice for simulation Pure Current Source? Show me a current without a voltage. Ohms law incorrect? mmm.

Wish they want to get it right.

29. BEMNET says: on April 27, 2015 at 6:48 pm
Log in to Reply

FOR ME IF KICAD DEVELOPES A PLATFORM LIKE ORCAD DESIGN TOOL IN ITS SIMPLICITY I THINK IT IS GOOD FOR WHO USE THE TOOL TO DEVELOP BEST PCB/PWB DESIGN IN SIMPLIFIED WAYFOR MORE COMPLEXITY SIMPLIFICATIONIS CADSTAR DESIGNER THANK YOU

30. BEMNET says: on April 27, 2015 at 6:57 pm
Log in to Reply

I DONT RECOMMEND ANY EDA TOOLS THAT IS REGARDED TO THE DURATION AND STANDARD OF THE USER FOR ME I TRY TO FIND COMPACTIABLE EDA TO DESIGN HDI AND HIGN DENSITY BOARDS FOR PURPOSE OF COMPLEX POWER SUPPLIE BOARDS INSTED I DONT FIND ANY SO WHAT KIND OF

31. Ajit says: on May 2, 2015 at 5:04 pm
Log in to Reply

I use diptrace. very easy.try it.

32. stephen van buskirk says: on May 18, 2015 at 2:07 am
Log in to Reply

I just downloaded DesignSpark PCP, twice, and both times I tried to run it and got an error: Authorization Service Error. Go figure.

I have been designing pcbs since you used tape and decals on clear mylar sheets to make artwork. I used pads and liked version 1.2.It had the BEST autorouter. A company I work for bought v7 something a few years ago for thousanbds. It was a BIG step back. The autorouter was poor and the nifty pull down menus meant that it took 7 clicks to change the layer on each segment of a route.

The only problem I had with v1.2 was it didn't support single combined gerber files.

My computer died and my v1.2 disk is bad so I've been trying free layout programs and so far they ALL suck. (apologies to Leo Tolstoy, but they all suck in their own way).

By any chance is there anyone who still has pads v1.2. If I could get a copy I'd be so happy I could wet myself.

Thanks – stevejvb-at-gmail-dot-com.

Now my computer

-

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

Pentru Romania, realizare de cablaje imprimate pe

cablajepcbpret.blogspot.ro

la doar 9 lei / dm2 fara alte costuri de pornire si gaurire inclusa pentru 300 gauri

34. Shalin says: on June 3, 2015 at 11:22 am
Log in to Reply

Schematics can be drawn from http://creately.com diagram software as well. Its limited to real-time logic gates for now. However you can make very pretty circuit diagrams with creately.

35. NottsGuy says: on June 15, 2015 at 10:21 pm
Log in to Reply

Installed DesignSpark and it failed at the first hurdle. IE most of the components I wanted to use from the RS catalogue were not in the library. EG any of the DIN-41612 a+b 1/2B type 32 way.

36. SAM says: on June 28, 2015 at 9:32 am
Log in to Reply

Which pcb layout software is best to design double side/sigle side PCB layout

o signality says: on June 29, 2015 at 12:46 pm reply

@Sam,

Try https://EasyEDA.com

It's a free, web-based EDA suite that allows you to download the Gerber's you generate in their s/w so you can buy PCBs from anywhere or you can buy low cost PCBs direct from EasyEDA.

37. Bobby says: on July 15, 2015 at 1:24 am
Log in to Reply

Hey guys,

I've drawn up exact size images with my ACAD program and printed the images onto plain printer paper then re-inserted the paper back into my laser printer with the same exact orientation as the first printing with clean smooth aluminum foil taped (with scotch magic tape) at it's edges, on top of the image of the first printing. Then I've printed the image onto the aluminum foil with the acid resistant ink that laser printers use. This can be ironed onto circuit boards to be etched. Some people have done this with just the plain paper but the paper fibers get stuck into the ink and cause problems in etching and/or while trying to remove the paper stuck to the board. I then immerse the to be etched metal into a shallow dish of muriatic acid which one can get from almost any hardware store and it will totally dissolve the aluminum foil leaving the acid resistant ink on the metal being etched, clean and undisturbed. I've done this to make fine detailed stamps in harden steel with amazing results. I haven't tried etching copper clad boards with the muriatic acid but if it doesn't etch, I'll just rinse the board and etch it in the usual way. That is what I want to try and that is why I came to this site to find a program that I might be able to use to design my circuits and make my circuit board printouts. Disclaimer: I am telling you not to do this, but if you do it anyway

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

18/05/2016 14:07



Just buy a laserfilm called "Polimat" for lasers and the spray to get the toner darker. No messing with tonertransfers and such. There are two sizes A4 and A3.

38. *cavejug* says: on <u>July 21, 2015 at 1:34 pm</u>
Log in to Reply

It started out as a list of free PCB software, however, many of the posted comments were about paid software, personal likes and dislikes and so on. The forum lost the plot and very few staid on the subject.

39. Daniel.R says: on August 13, 2015 at 9:22 am
Log in to Reply

"PCB Artist" is an easy-to-use, intuitive printed circuit board design software; It is a free design software easy to get for PCB industry manufacturers, layout designers and electrical engineers.

"PCB Artist" contains schematic symbols, footprints and parts generation wizard to improve PCB design process, shorten the time of generating components to only five minutes. In addition, customers can take advantage of a wide range of component library as a reference. In addition to the functions currently provided, "PCB Artist" also:

- Provides Gerber format files when needed;
- automatic components mounting and tracing simulation
- -Well integrated, convenient for user to change the schematic, schematic diagram can be automatically converted to the PCB design.

Download it from here: http://download.cnet.com/PCB-Artist/3000-18496_4-10712335.html If you need source of PCB or PCBA, here is a recommendation: http://www.syspcb.com

40. Andy Fierman says: on August 14, 2015 at 1:38 am
Log in to Reply

PCB Artist is free in the sense that the software is free to use but it appears to tie the user to having their PCBs made by only the company that provides the software:

http://www.4pcb.com/free-pcb-layout-software/

Unlike for example, https://EasyEDA.com, it seems that the user is not free to download the Gerbers and have their boards made by any PCB supplier of their choice.

It is also Windows only although it may run on Linux and Mac using WINE.

41. anand says: on September 26, 2015 at 4:38 pm
Log in to Reply

hi i want to do 300*300 mm pcb please send me the best software plase suggest me

42. anand says: on September 26, 2015 at 4:39 pm
Log in to Reply

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Read More



I need pcb design software of having amplifier ic TDA2822M. Will u tell me names of software???

• MaxBlack says: on October 30, 2015 at 5:01 am reply

https://easyeda.com/

EasyEda is a tool based on the cloud platform which facilitates schematic drawing, spice simulation and PCB layout. They have over 70,000+ readily available schematic in their web database along with 15,000+ Pspice libraries.

You can find your IC here.

44. snowabbit says: on November 7, 2015 at 2:20 am
Log in to Reply

I downloaded DesignSpark and tried running it. But no way could I get libraries to load. Gave up and after reading most of the comments have now uninstalled it.

45. Max says: on December 8, 2015 at 6:50 pm
Log in to Reply

I have made the same experience with DesignSpark. Now I use CAD SoftEagle and im quite happy. You can get started with a free trial

 $\underline{http://www.cadsoftusa.com/download-eagle/}$

46. *MikeM* says: on <u>January 9, 2016 at 6:44 pm</u> Log in to Reply

After a week working on PCBweb, it will not open the project files anymore. Also it has a very limited library. Works with DigiKey for some parts but even that is not complete and you'll have to create symbols for basic parts like zeners, trimmers or tubes. Limited and buggy still.

47. *MikeM* says: on <u>January 9, 2016 at 8:05 pm</u>
Log in to Reply

Also be aware of Mouser's MultiSIM Blue. Schematics are limited to 50 parts in FREE mode, and Utiliboard (to design pcb) is a trial. No print or export, something they will not tell you.

48. *namehta39hotmail-com* says: on <u>February 13, 2016 at 8:17 am</u>
Log in to Reply

Good articles. most of them are useful. I have asked about power supply design for LED lamp. How I will get the reply and how i will know, if I have got some replies? will I get email or will I have to surf answers everyday?

Thanks

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

account.



Login Email	
Password	
Remember Me	
Log In	

Register | Lost your password?

Join all of us on our group



Recent Questions

- inverter power consump in watt at charging time
- Here is more info hope it helps
- what ic can be used to start and stop a buzzer in 3 minutes
- In the schematic universal-triac-control-with-optocoupler, the PNP transistor 2SB328 is shown as NPN. Please clarify.
- <u>In the schematic universal-triac-control-with-optocoupler, the PNP transistor 2SB328 is shown as NPN. Please clarify.</u>

recently added electronic circuits

- How to Play with Thermocouples
- Layman's Camera Remote Control
- Selecting Semiconductor Components For Prototype Design
- Filters Tutorial
- Webcam Sight Illuminator Circuit

Links - Policy - Disclaimer - © Copyright 2008 - 2015

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

This website uses cookies to improve your experience. We'll assume you're ok with this, but you can opt-out if you wish.

Read More

Accept

22 of 22