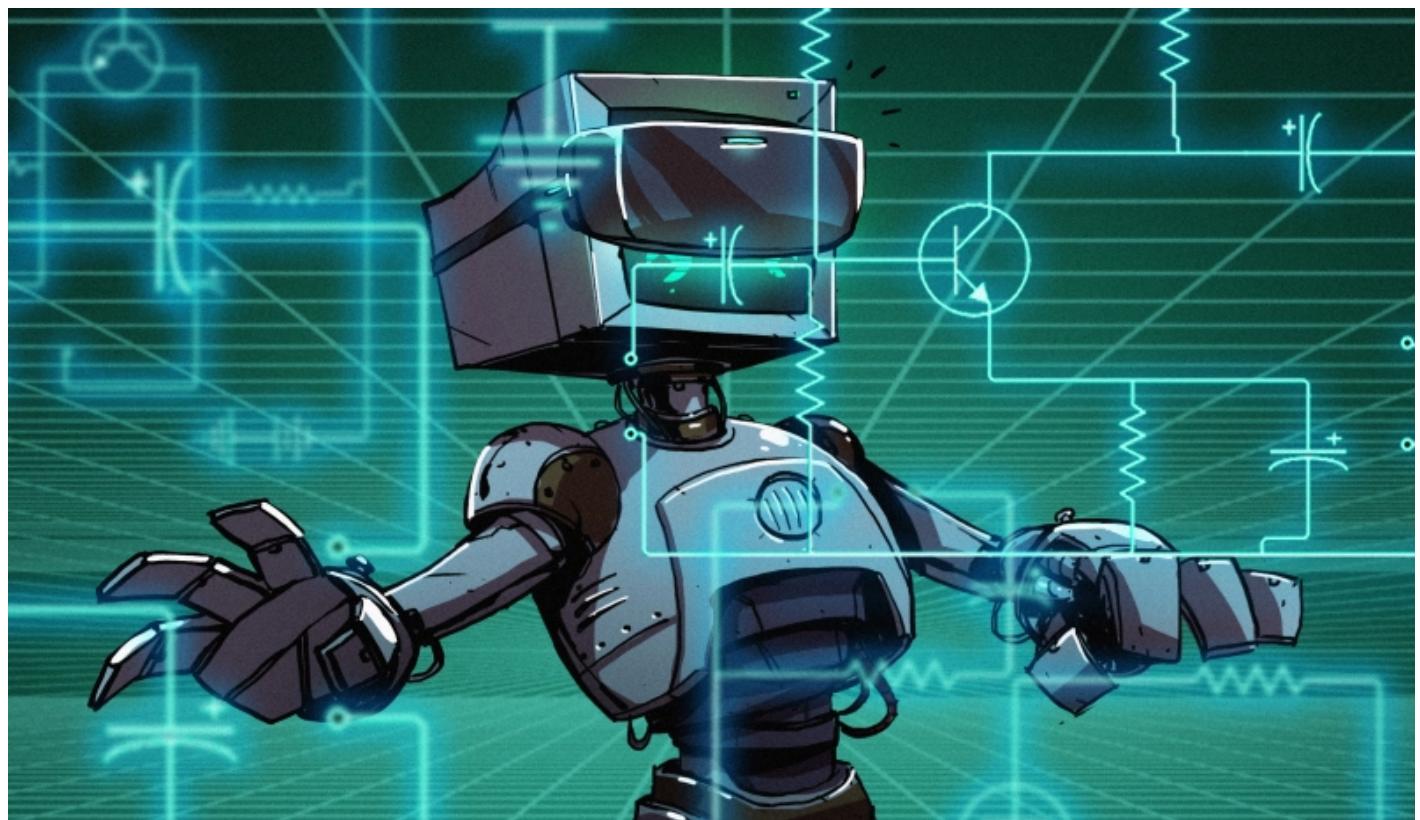




30 FREE CIRCUIT SIMULATORS LIGHTLY REVIEWED

by: **AI Williams****17 Comments**

August 4, 2022



We live in a time where great software is available with the click of a mouse, often for free or — at least — low cost. But there's a problem: how do you select from so many alternatives? We were interested in [Lee Teschler]'s review earlier this year of [30 free circuit simulators](#). If you are selecting one or don't like the one you are currently using, it is well worth the time to review.

There are several on the list that you've probably heard of before like GnuCap and LTspice. There are also some lesser-known products. Some of those are just trial or student versions of paid products. Some are branded versions of commercial products (like Tina) or were made free after selling for higher price tags (like [MicroCap 12](#)).

Old favorites like Falstad (which is apparently known as Circuit Sims) and TinkerCAD made the list. Many of the trial versions were very limited. For example, DCACLab only provides an NPN bipolar transistor model. Proteus doesn't let you save or print unless you pay. While the list includes TI's Tina, it doesn't seem to mention that TI also provides a **free version of PSpice** which is a very popular professional product.

While the capsule descriptions are nice, you may want to dig in a little on the ones you are most interested in. For example, Falstad has a great mixed mode that can even **include an AVR microprocessor**. But there were a few on the list we had not heard of and maybe you'll find something new there, too.

Posted in **Software Hacks**

Tagged **simulation, SPICE**

← REDUCED SULFUR EMISSIONS COULD CAUSE CLIMATE SHOCK

CONVERTING AN 80S TYPEWRITER INTO A LINUX TERMINAL →

17 THOUGHTS ON “30 FREE CIRCUIT SIMULATORS LIGHTLY REVIEWED”

Pat says:

August 4, 2022 at 12:43 pm

Qucs and QucsStudio (not mentioned) are actually different programs – the backend simulation stuff for QucsStudio is entirely redone. It's one of the only transmission line calculators I've found that has coupled CPW-G as an option and actually tends to match direct finite-difference results (like atlc).

Reply

Report comment

Drone says:

August 4, 2022 at 2:57 pm

@Pat said: “Qucs and QucsStudio (not mentioned) are actually different programs...”

Actually there is a bit more to the story. The Qucs-based eco-sphere is rather fragmented and somewhat confusing in my opinion. To my knowledge there are three main facets, each with their own strengths, Qucs, QucsStudio, and Qucs-S, with not a lot of interoperability between them:

1. Qucs

Qucs (Quite Universal Circuit Simulator) is the original application. It enables you to setup a circuit with a graphical user interface (GUI) and simulate the large-signal, small-signal and noise behavior of the circuit. After the simulation has finished you can add and view the simulation results along with the circuit schematic or on a presentation page. Qucs has some nice RF capabilities. Qucs offers Windows, macOS, and Linux binaries. The Windows version is stand-alone (no installer).

<http://qucs.sourceforge.net/>

https://en.wikipedia.org/wiki/Quite_Universal_Circuit_Simulator

2. QucsStudio

QucsStudio a free, non-commercial, powerful circuit simulator. QucsStudio is free but does not seem to be open source. QucsStudio is mainly a circuit simulator that has evolved out of the project Qucs, but isn't compatible with it. The simulation engine is even a complete new creation. It's meant to be a test project to create a complete development environment for electrical engineers. (graphical user interface, circuit simulator, PCB layout, and numerical data processing etc.) Currently QucsStudio supports Windows only. The Windows binary is stand-alone (no installer).

<http://qucsstudio.de/>

3. Qucs-S

Qucs-S is a spin-off of the Qucs cross-platform circuit simulator. The “S” letter indicates SPICE. The purpose of the Qucs-S subproject is to use free SPICE circuit simulation kernels with the Qucs GUI. It merges the power of SPICE and the simplicity of the Qucs GUI. Qucs-S offers Windows and Linux binaries. The Windows version is stand-alone (no installer).

<https://ra3xdh.github.io/>

Reply

Report comment

Simon Clifford says:

August 10, 2022 at 5:45 am

QucsStudio is more of a simulator for people looking for something like Microwave Office or HP's (sorry Keysight) ADS but short of 80-90k for a license. It has the same programming capability in a schematic as LT SPICE, and the array of optimisers – astonishing, oh and it'll do 2 1/2 D FDTD finite element Maxwell Equation simulation on the PCB you're designing too. It really is a remarkable package.

Reply

Report comment

Tony says:

August 4, 2022 at 1:27 pm

Target3001 ! great software, from Germany I think, I used to work with it when using Windows, I went to the dark side many years ago but I still miss it.

https://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Main_Page

Reply

Report comment

Drone says:

August 4, 2022 at 1:53 pm

Target3001 "free" is quite limited and gets real expensive real fast as you remove restrictions:

<https://ibfriedrich.com/en/index.html#products>

Other than that it looks pretty good. LTspice and KiCad is where I am at right now, and quite comfortable.

Reply

Report comment

Chris says:

August 4, 2022 at 10:46 pm

The non commercial, light version is okay for “normal” hackers. And not expensive. I use it for years now and I am quite happy with it.

[Reply](#)

[Report comment](#)

smellsofbikes says:

August 4, 2022 at 1:30 pm

I've really enjoyed the fairly good integration that ngspice has with KiCad schematic, and that it can, with caveats, use pspice and ltspice directives and workflow.

[Reply](#)

[Report comment](#)

jpa says:

August 4, 2022 at 10:50 pm

Yeah. Combined with hierarchical schematics, it is quite reasonable to use the same schematic for both PCB design and for simulation, which is often not possible with separate simulators.

[Reply](#)

[Report comment](#)

Antron Argaiv says:

August 4, 2022 at 2:49 pm

I'm a fan of LTSpice, and it runs on Linux under WINE. The author is a longtime LT employee and a bit of a curmudgeon, but he's dedicated to making LTSpice aka SwitcherCAD the best he can. I heard him speak once, and he confirmed a commitment to keep LTSpice runnable on Linux using WINE

[Reply](#)

[Report comment](#)

Greg says:

August 5, 2022 at 9:09 am

He is definitely a curmudgeon but he does know the ins and outs of his program. He will tell you the limits of LTSpice and Spice in general as well as where LTSpice excels. We use LTSpice at work almost exclusively for discrete component simulation. So far, it's only limitation is the parts library favoring ADI components.

[Reply](#)

[Report comment](#)

Mark Walter says:

August 4, 2022 at 4:26 pm

SPICE Opus is a good non-graphical (netlist input) simulator very similar to the original Berkley SPICE. Opus has some additional features geared to circuit optimization. SPICE Opus is free.

[Reply](#)

[Report comment](#)

GenTooMan says:

August 4, 2022 at 8:28 pm

FYI if you frequent librachat the “unofficial support channel” #kicad has been having discussions on updates in the last several months regarding ngspice so look forward to further integration and interesting (if not actually useful changes to) ngspice and its KiCAD integration.

Currently the most frustrating issue in ngspice (in KiCAD or independent usage) is a lack of spice models for common existing devices.

[Reply](#)

[Report comment](#)

Steven says:

August 5, 2022 at 1:47 pm

It wasn't mentioned yet, so I'll share CircuitJS.

<https://www.falstad.com/circuit/circuitjs.html>

It's a Java applet that's been converted to JavaScript and is open source. It does live simulation with reasonable accuracy and has a nice variety of generic component models as well as subcircuit support. It isn't something you'd use for complex precision circuits, but it is great for simple circuits, prototyping, and more importantly I use it for illustrating circuits and sharing them as links.

[Reply](#)[Report comment](#)**Steve** says:[August 5, 2022 at 11:35 pm](#)

I'll agree, CircuitJS is great when you need to share or show others a small schematic or circuit idea, without a lot of technical detail or a high component count. It's quick and easy, and sometimes that's all you really need.

That said, I wish there was a 'modernized' LTSpice out there, one that was truly cross-platform, allowed plug-ins, etc. But for what it is, it's still a good tool, among the free-to-use choices.

[Reply](#)[Report comment](#)**stts** says:[August 8, 2022 at 7:24 pm](#)

Yup, I just tried the PC version CircuitJS last weekend. It was really easy. I tried PartSim first. It was kinda a pain to enter in my circuit. Wires didnt connect together as expected. Causing trouble. Finally figured out how to fix the circuit, but I never could get the Mosfet to turn on. No matter how I hacked on the gate voltage. So I gave up and went to CJS. Today I went back to PartSim and it wont even let me add parts any more. PartSim is just a broken waste of time. CJS was 3 times quicker at building circuit, and it worked first time in Sim. But its Generic parts. I need to find a sim that will let me enter real world parts.

[Reply](#)[Report comment](#)**Conor Stewart** says:[August 6, 2022 at 8:48 am](#)

For digital circuit simulation, logisim is the most well known one but there is a better alternative called “Digital”, it is on GitHub and is open source. If you search “digital logic simulator GitHub” it is one of the first options, it is from hneemann.

It is faster than logisim, has blocks to allow you to write and simulate verilog and vhdl and allows you to convert your digital circuits into verilog or vhdl. It also has a graphics ram block that allows you to display the contents of the ram in a window.

Overall it is better than logisim for most digital design tasks and would recommend anyone to give it a go.

[Reply](#)

[Report comment](#)

Micha says:

August 7, 2022 at 10:24 am

I use Falstad simulator for my simple projects. It is accurate enough for me – everything works as simulated but i think MOSFETs had a bug where they never saturated and had huge voltage drop. It can even run Chua chaos oscillator with a little component tuning. Never tried any serious RF work and you probably cannot make filters above LF in it.

[Reply](#)

[Report comment](#)

Leave a Reply



Email (required)

(Address never made public)

Name (required)

Website

Save my name, email, and website in this browser for the next time I comment.

Notify me of new comments via email.

Notify me of new posts via email.

Post Comment

Please be kind and respectful to help make the comments section excellent.

([Comment Policy](#))

This site uses Akismet to reduce spam. [Learn how your comment data is processed.](#)

SEARCH

Search ...

SEARCH

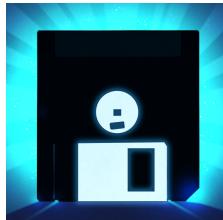
NEVER MISS A HACK

SUBSCRIBE

Enter Email Address

SUBSCRIBE

IF YOU MISSED IT



FLOPPY DISK SINGS: I'M BIG IN JAPAN

50 Comments



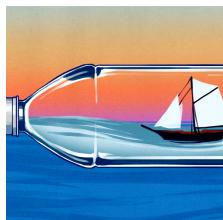
ROBOTIC SURGEONS ARE SHOWING HINTS OF ONE DAY OUTPERFORMING HUMANS

30 Comments



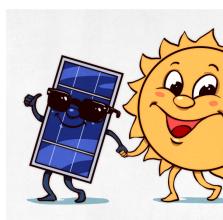
FRANK DRAKE'S LEGACY, OR: ARE WE ALL ALONE IN THE UNIVERSE?

76 Comments



DON'T BE SALTY: HOW TO MAKE DESALINATION WORK IN TOMORROW'S WORLD

54 Comments



AGRIVOLTAICS IS A LAND USAGE HACK FOR MAXIMUM PRODUCTIVITY

53 Comments

[More from this category](#)

OUR COLUMNS



GROUNDWATER: MANAGEMENT OF A MUCH NEGLECTED LIFELINE

9 Comments



HEAVY ENGINEERING HACK CHAT

1 Comment



GIT INTRO FOR HARDWARE HACKERS

19 Comments



HACKADAY LINKS: SEPTEMBER 11, 2022

9 Comments



WHO IS RESPONSIBLE FOR YOUR SAFETY?

181 Comments

[More from this category](#)



NEVER MISS A HACK

Copyright © 2022 | **Hackaday, Hack A Day, and the Skull and Wrenches Logo are Trademarks of Hackaday.com** | Privacy Policy | Terms of Service
Powered by WordPress VIP