Free CAD

online design tools

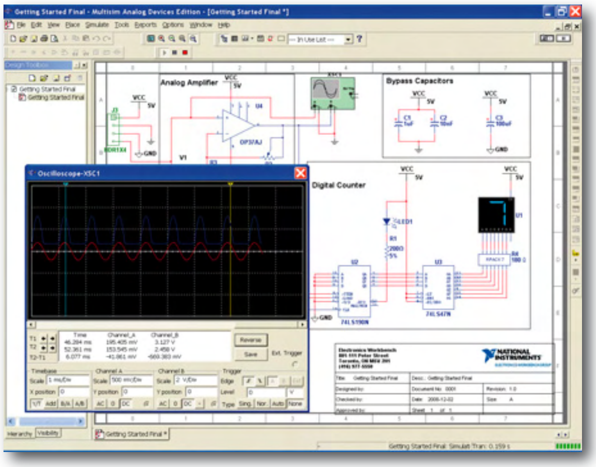
Clemens Valens

Clemens Valens (Elektor France Editorial)

Several electronic component manufacturers now offer design tools online or for free download. We registered – often an obligatory step – on a dozen or so websites to see just what's being offered and what it's like.

Analog Devices — www.analog.com

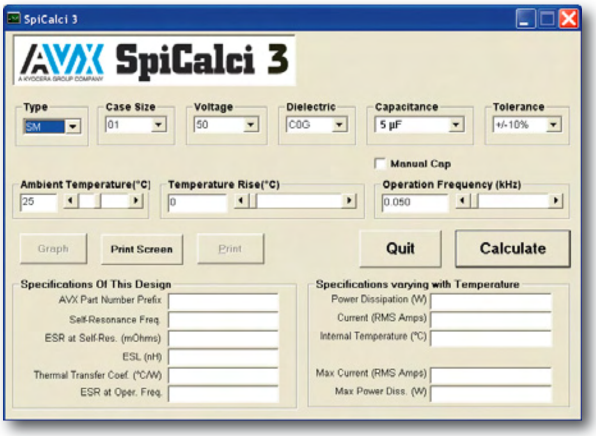
The Analog Devices website offers a whole range of interactive design tools, with names that all start with **ADIsim**. For example, there's **ADIsimPower**, a simulator for power supplies – or DC-DC convertors, to be more precise. Analog Devices also offers a resistance bridge calculator, an aid for dimensioning photodiode amplifiers, and a tool for calculating analogue filters.
There are also tools on free download, like **SRD Design Studio**, a tool for simulating short range radio systems, and **Multisim for Analog Devices**, a monster at 183 MB (v 10.0.0).
After running this program, we downloaded the *Getting Started* project, which is not actually mentioned anywhere, but can be found at the bottom of the directory where the program has been installed. This example shows a single-digit digital counter using the output of an amplifier amplifying a 1 kHz signal as its clock. Clicking the *Run* button at the top right of the screen starts the simulation. Double-clicking on the oscilloscope icon opens a window displaying a two-channel digital oscilloscope. It's all very pretty.
Apart from some models of certain common components,



Multisim only handles Analog Devices components, and it's not possible to modify them or create new ones.

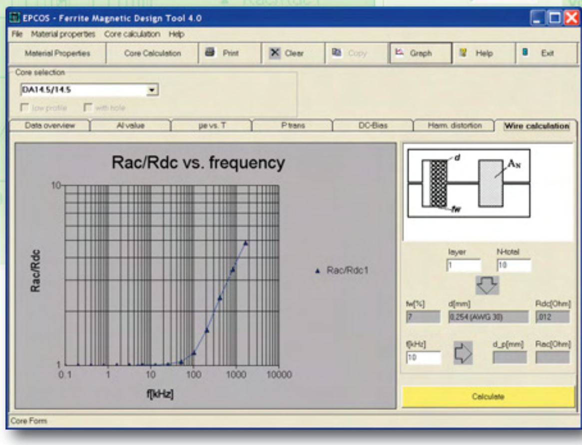
AVX — www.avx.com

AVX offers several tools for selecting its capacitors. We tried **SpiCALCI 3.0**, a tool for dimensioning capacitors for switch-mode power supplies. Unfortunately the program does not offer a help function so a good awareness is required of the various capacitors in the AVX product range. After selecting a capacitor type, working voltage, dielectric, etc. the program displays the capacitor's resonance frequency and dissipation at a user selected temperature. A graph can be produced showing impedance response and ESR.



Epcos www.epcos.com

This passive components manufacturer offers several tools, including **AlCap** for dimensioning aluminium electrolytic capacitors, and the **Ferrite Magnetic Design Tool**, used for evaluating Epcos magnetic materials and for performing certain calculations pertaining to inductances and coils.

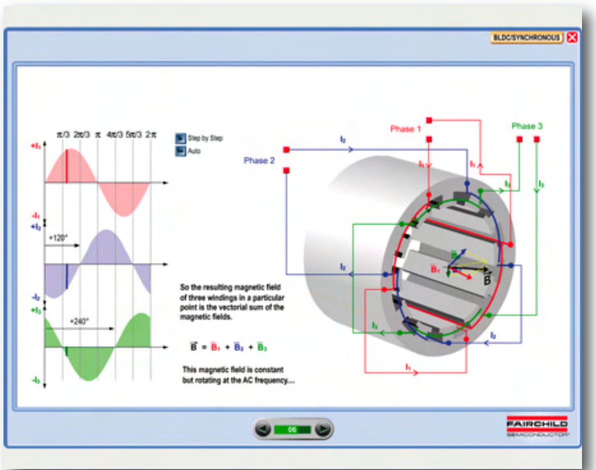


Fairchild www.fairchildsemi.com

FETBench is Fairchild's online design tool, and comprises three parts:

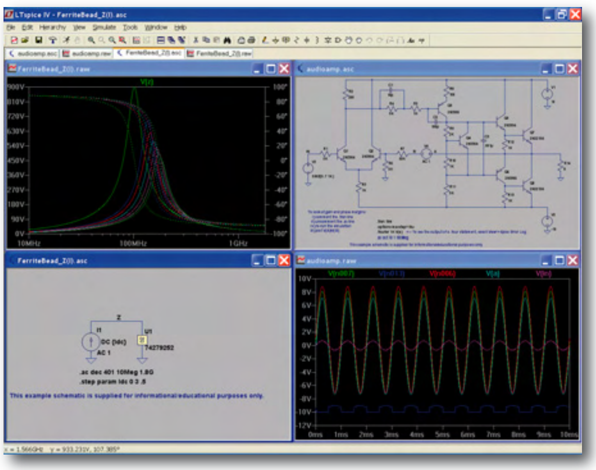
- **Device Analysis** lets you evaluate a component with customized graphs;
- **Application Analysis** is a switch-mode power supply simulator;
- **Thermal Analysis** lets you specify a board and position active components from Fairchild on it. Then the tool shows a coloured diagram with the temperature distribution across the surface of the board.

But there's even better from Fairchild: **SPM Tool**, a very fine piece of software that is not only able to simulate motor drivers, but also contains all the theory of motors in the form of interactive animations. Not to be missed! Before you can download the software, you are asked for a username and password. These are the same ones you used when you signed on.
The same goes for **PFC Toolkit**, a tool for exploring phase correction techniques.



Linear Technology www.linear.com

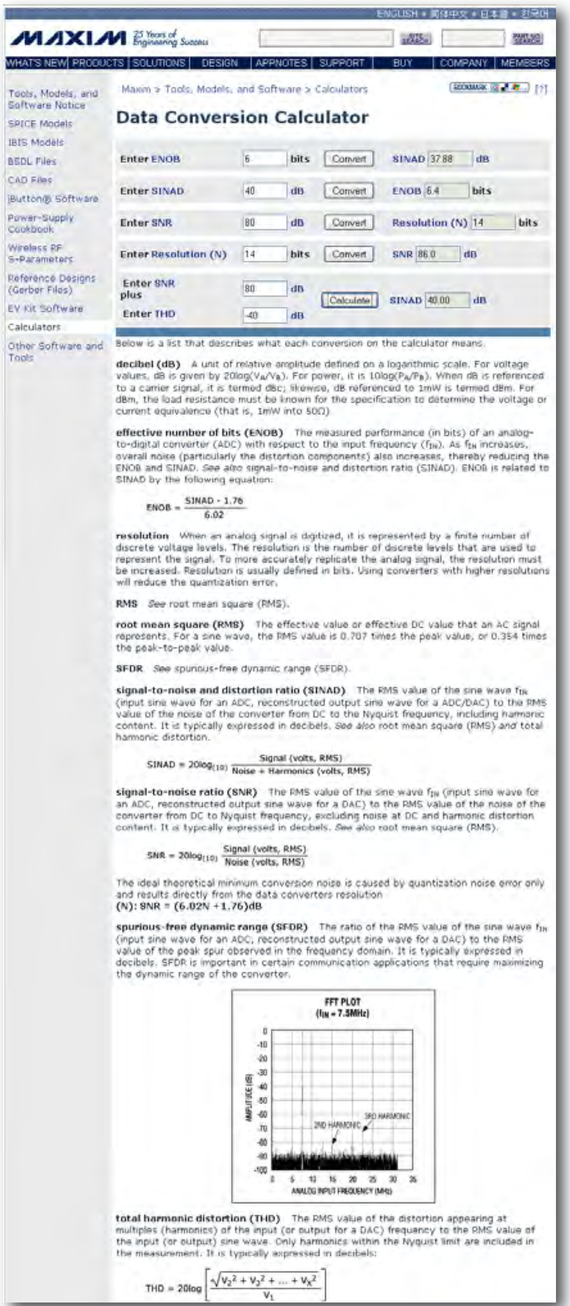
LTSpice IV is the current name of the SPICE simulator from Linear Technology, originally launched a few years ago as **SwitcherCAD**, principally for simulating switch-mode power supplies. Users soon found that the tool is in fact a souped-up SPICE that lends itself perfectly to other simulations. LTSpice is fairly easy and intuitive to use. You can quickly draw a circuit diagram and simulate it equally swiftly. OK, so maybe the graphics aren't the best in the world – but this is a very powerful tool and is regularly updated.
Apart from LTSpice, the site also offers **FilterCAD** and **BodeCAD** and a number of tools for evaluating certain Linear Technology components.



Maxim www.maxim-ic.com

Maxim does not offer any very sophisticated online design tools, but there are twenty or so useful calculators available. Some of them are dedicated to a specific Maxim component, but not all of them, and there are several you may find useful. The interesting thing is that Maxim gives references and detailed explanations that let you delve into the theory applied practically in the calculator.

The **Power-Supply Cookbook** lets you download, for each regulator in the Maxim catalogue, a PDF file containing a circuit diagram and component list for several configurations of the chip. It is also possible to download all the files in one go, in the form of a 9 MB PDF, which does save time. It's a bit like the old databooks, with loads of ideas for circuits.



Microchip www.microchip.com

Mindi is the analogue simulator from Microchip based around (X)SPICE and developed by SIMetrix. Mindi is supplied with lots of component models – we counted around 4,500 – which makes entering the circuit very convenient. After selecting a component, you can modify its parameters, which in most cases comes down to choosing one of the models provided.

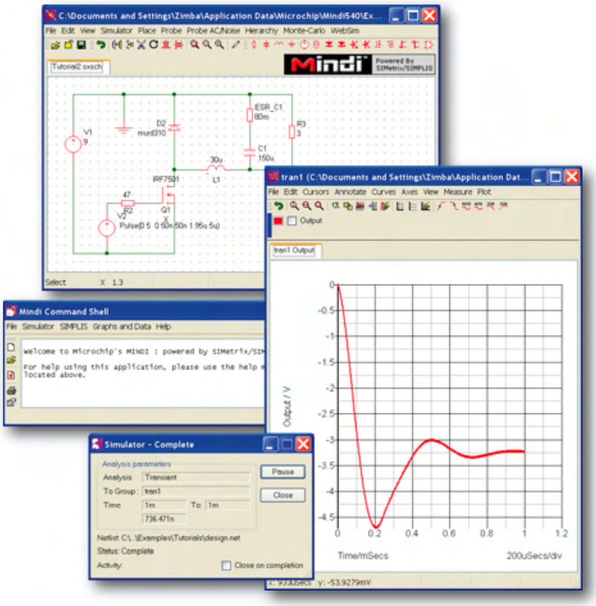
Another free design tool from Microchip is **FilterLab2**, which lets you simulate Chebychev, Butterworth, and Bessel type filters. In a quite user-friendly environment, you can choose your filter, and the software plots the characteristic curves. It also offers an implementation of the filter, based on Sallen & Key or MFB (multiple feedback) sections. Several ways of designing the filter are offered. Once you are satisfied with the theoretical performance of the filter, you have immediately available a circuit diagram and a SPICE model.

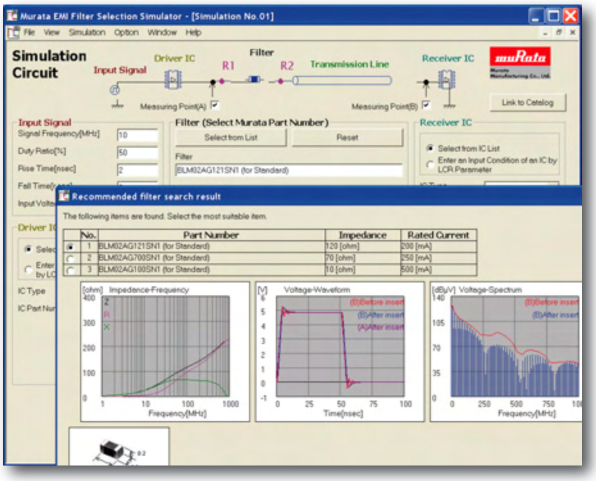
Murata www.murata.com

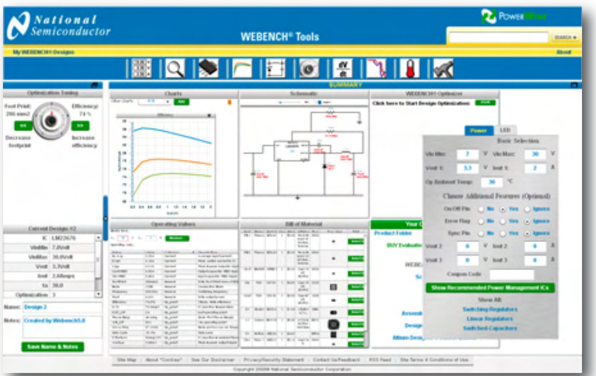
Five design tools are available for download from Murata, for capacitors, EMC filters, and thermistors. We tried out the **Murata EMI Filter Selection Simulator** (version 3.6.0). This tool lets you dimension EMC filters for transmission lines. Several configurations are supported and users can set the circuit parameters in detail. Once these parameters have been entered, a click on the *Start Simulation* button is all it takes to have access to the results in the form of diagrams. In short, a fine, easy-to-use tool.

National Semiconductor www.national.com

National Semiconductor has been offering its **WEBENCH** simulator for some years now, originally for simulating power supplies, but now the tool also helps with designing filters, phase-locked loops, LED drivers and even audio amplifiers. In less than ten minutes, for example, we were able to design and simulate an ADC with anti-aliasing filter. Remember to enable pop-ops, otherwise you won't see all the results of the simulations.



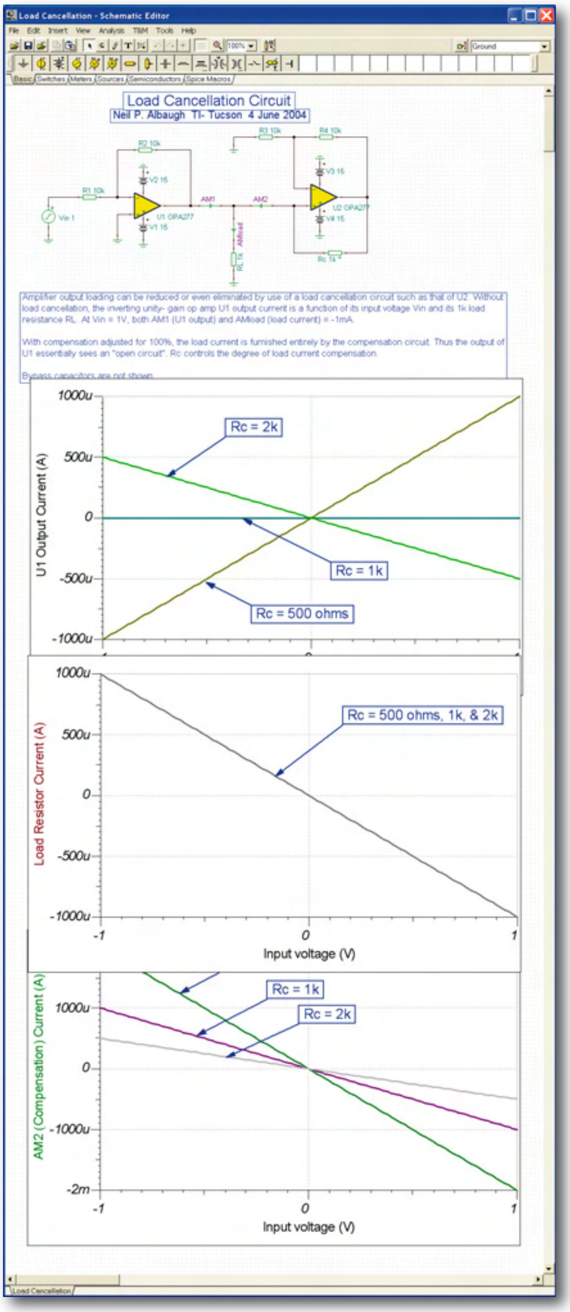




NXP

www.nxp.com

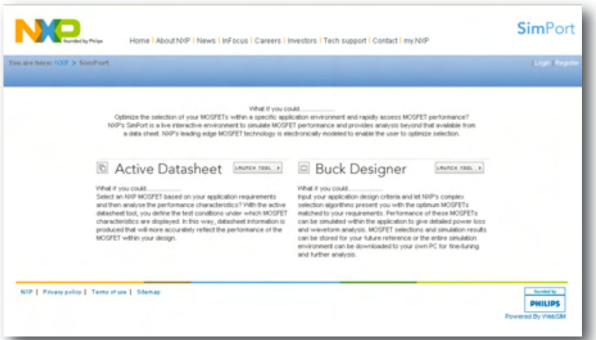
SimPort brings together the online simulators from NXP, developed, like Mindi from Microchip, by SIMetrix/SIMPLIS. In fact, there are just two: **Active Datasheet** and **Buck Designer**. Before it is possible to use them, you need to register, just like everywhere else. Nothing so special about that, you may say – but you do need to have good eyesight! If you're not yet registered, you need



to follow the link at the top right of the page which lets you choose between the two simulators. In *Active Datasheet*, you can select a MOSFET from NXP and generate for yourself the curves that are usually found on a data sheet. The difference from a data sheet is that here it's up to you to specify the values for which the curves are generated. *Buck Designer* is a tool for simulating a step-down converter The tool offers you a block diagram and a detailed circuit diagram, and generates diagrams where you can enlarge certain zones and measure certain parameters.

STMicroelectronics www.st.com

Tools from ST, online and to download. For the online tools, an audio amplifier simulator and a switch-mode PSU simulator, you need to install the Java runtime environment (JRE) first. Then the tools will work… more or less, at least, on my test computer.
On the other hand, **ST Lighting Designer** is a Flash tool for dimensioning fluorescent lighting that is worth going out of your way for. A designer must have spent a lot of time on the graphics interface – you'll either like it or you won't – and the tool works wonderfully well. Once you have realized that it's not possible to have a solution with all the options enabled, the tool outputs the circuit, the components list, and some graphs. You can then fine-tune loads of parameters.

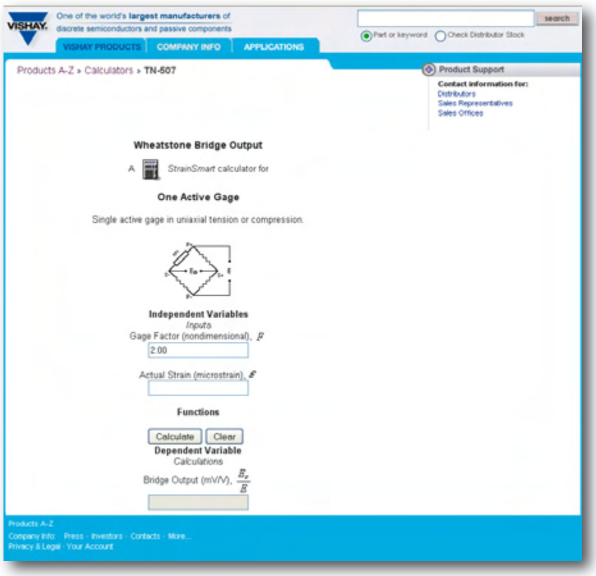


Texas Instruments www.ti.com

TINA-TI is the free simulator for downloading from Texas Instruments and is able to use SPICE models. TINA-TI is one part of TINA, a complete CAD tool with circuit entry, PCB design, and simulator.
SwitcherPro is Texas Instruments' switch-mode PSU simulator. For some months now, this simulator has also been available online. As usual, you have to register before you can use it. Once you are connected to the simulator, it can be run in two ways: from a component or from specifications.

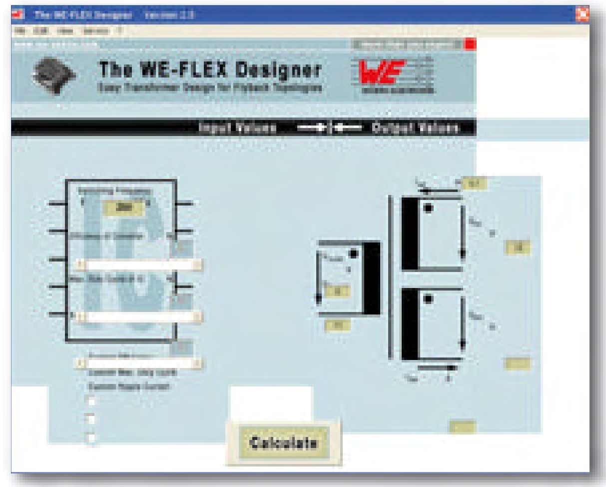
Vishay www.vishay.com

ThermaSim is a thermal simulator for PCBs. It's not terribly intuitive to use, but after a bit of effort you can manage to get the tool to accept your parameters, and it will then send the results by e-mail – rather original, but not very quick. And then when the message does eventually arrive, you find that there was a problem and the simulation has failed. "Please correct and start again", the e-mail reads. Sure, of course…
Apart from ThermaSim, Vishay also offers several basic calculators, like for example the TN-507 calculator for strain gauges.



Würth Elektronik www.we-online.com

Two tools are available from the Würth website: **WE Inductor Selector 1.0** and **WE Flex Designer 2.0**. The latter is used to select a transformer for flyback-type switch-mode power supplies. The tool isn't exactly pretty to look at, at least on the test PC, since all the buttons and boxes were displaced, however a single click on the *Calculate* button is all it takes to generate a transformer reference number.



And finally…

It's 2009, and all the component manufacturers are offering free design tools… All of them? Well, not quite… some manufacturers are still holding out against this trend. On the Infineon, Hitachi, Freescale, Toshiba, and NEC websites, for example, we found nothing at all that was worth mentioning here.

Despite the number of design tools available on the Web, we may well be asking ourselves just how useful they are. Sure, some of them are well done, some of them are good to look at — but is there really any need for so many switch-mode PSU simulators?

(080874-I)