The PC as Breadboard

Simulating electronic circuits with SPICE

Kees de Groot

Electronics designers these days spend more time behind their PCs than holding their soldering irons. Thanks to clever software, entire circuits can be simulated without making even a single solder joint. The basis for such simulation software is SPICE, which was developed back in 1972. Here we give a concise description of the way SPICE achieves a realistic simulation of components and circuits.

SPICE is a program you can use to simulate electronic circuits. All voltages and currents can be examined before the circuit has been physically built. There you have it: soldering and experimenting using the PC!

The circuit can be made from all currently known components. That means resistors, capacitors and inductors, but also diodes, transistors and FETs. Many ICs are now also available via libraries. You can define new components yourself, buy them or pick them from the Internet. The results from the simulation generally correspond very well with the real world, even up to very high frequencies.

In addition to analogue circuits, modern simulation software can also simulate digital circuits, such as microcontrollers, RAM and circuits with digital ports, as well as antennas and transmission lines.

Now why would I use SPICE?

For the hobbyist, SPICE offers a tremendous opportunity to experiment with new or (still) unknown components. It is interesting to spend a rainy Sunday afternoon to put together a valve circuit and then drastically lower the supply voltage and examine where problems occur in the circuit. A push-pull audio amplifier power stage with a class-E RF final stage is quite easily slapped up from valves, transistors or FETs. And there is no risk of the accidental demise of expensive components!

The professional electronics designer, too, can benefit a lot from such software. Circuits and changes can be tested without building a new prototype every time. It is also possible to take into account the tolerances and temperature dependency of the components used. In this way it is quick to check if a circuit is reproducible in practice.

How it started

The development of SPICE (Simulation Program with Integrated Circuits Emphasis) dates back to 1972 when Larry Nagel and Donald Pederson of the University of

Berkeley in California wrote the very first version in Fortran. The early versions did not have a graphical user interface, because the programs were carried out on a mainframe computer. This is part of the reason why a rather Spartan description of the circuit is used. This method of description is still used in modern SPICE-models and sub-circuits (**Figure 1**).
Later versions of SPICE, we have now moved forward to 1985, were written in C. The first PC version, PSPICE, was marketed by MicroSim.

These days there are dozens of simulation tools that are more or less based on SPICE. In addition to the commercial versions there are also open-source versions. For educational purposes there are versions which are limited in the size of the circuit that can be simulated or versions with a time limit. Many simulation programs offer the possibility to enter the schematic with a graphical user interface (GUI) and display the simulation results in graphical form on a virtual oscilloscope. It is often also possible to seamlessly convert a virtual circuit into a PCB design. In addition to the simulation of standard electronic circuits, there are simulators for specialist fields. There are simulators of integrated circuits, microwave circuits and filters, but also radio antennas and even electromagnetic fields. The input can be done in the old-fashioned, numeric way, by naming all the inputs, outputs, nodes, voltages, currents and components. However, many modern simulation programs fortunately make use of graphical input, in which the mouse is used to place components and connect the parts together. For digital circuits a hardware description language like VHDL or Verilog is often used, extended if need be, with an analogue description language.

A completely different area is the simulation of mechanical systems. And what do you think of a simulation program that allows you to build LEGO designs? However, we won't pursue those ones any further here.

How does SPICE work?

SPICE makes use of the Laws of Ohm and Kirchhoff in a clever way.
Ohm's Law gives the relationship between the voltage across a resistor and the current that flows through that resistor. If at a voltage of $U = 12$ V there is a current of $I = 0.5$ A through the resistor, then the value of that resistor is 24 Ω ($R = U/I$).
Kirchhoff's Current Law states that at any node the current entering the node is equal to the current leaving the node. For example, connect a few water hoses to a T-coupling. All the water that flows into a hose is certain to leave via the other connected hoses. Not more and not less.

Kirchhoff's Voltage Law states that around a net (a loop through the circuit such that you end at the starting point) the voltages sum to zero. This is analogous to a cycle route through hilly terrain. Whichever route you choose, you can never go only downhill from the camping ground to the pub and then downhill back to the camping ground. You will later go up by the exact same amount as you went down. And you'll know it.

A small example: Assume that we want to charge a penlight (AA) battery of 1.2 V via a resistor of 10 Ω from a voltage source of 12 V. Across the battery there is a moving coil meter with an internal resistance of 100 Ω (**Figure 2**).

Kirchhoff's Voltage Law states

$$U_1 - I_1 \times R_1 + U_2 = 0 \qquad [1]$$
$$U_2 + (I_1 - I_2) \times R_2 = 0 \qquad [2]$$

These two equations can be solved for I_1 and I_2 using some simple algebra. It turns out that $I_1 = 1.08$ A and $I_2 = 1.068$ A. The battery will therefore be charged at a current of 1.068 A.

SPICE does it in the same way. At each node Kirchhoff's Current Law is applied and Kirchhoff's Voltage Law for each net. In this way you get a number of equations that are ultimately stored in memory in the form of a matrix of numbers. This matrix is inverted and in this way the set of equations is solved. Any arbitrary number of resistors, voltages sources and current sources can be connected, as long as we tell the computer what is connected
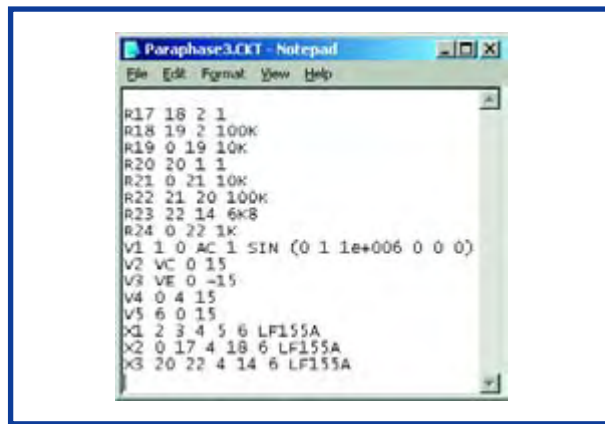

Figure 1.
Part of a SPICE netlist. All components are listed with the node numbers they are connected to, followed by other specific characteristics.
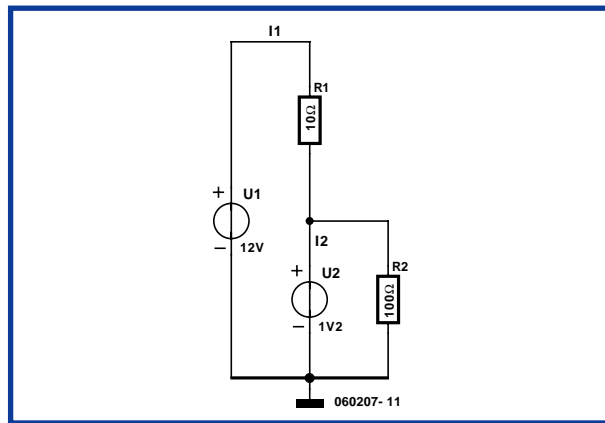

Figure 2.
Replacement schematic of a 12-V power supply which charges a NiCd cell via a 10-Ω resistor. A moving coil meter is in parallel with the battery.
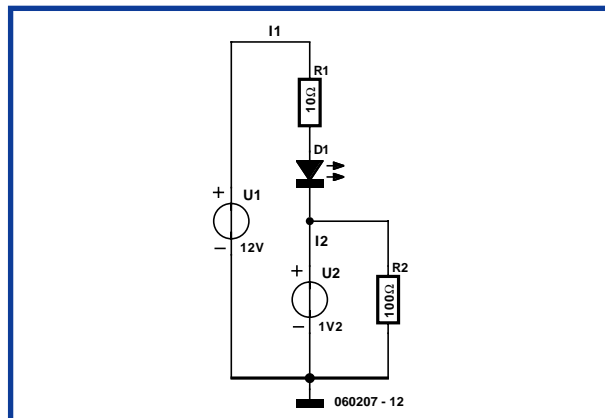

Figure 3.
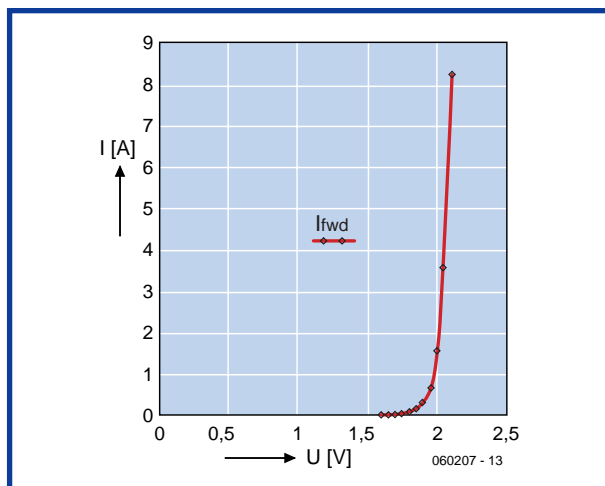**An indicator LED is added in series with the charging resistor.**

between the various nodes.

If we now connect an LED in series with R_1 (**Figure 3**), then finding the solution is not quite so easy any more. The LED will cause a reduction in voltage of about 2 V, causing U_1 to drop to about 10 V. Now we can, just as before, calculate the solution for I_1 and I_2, but with that value of 2 V, these will only be a rough approximation. If we want an exact solution, then we need some more maths, because with the LED we are dealing with a non-linear element.
For the current I_{LED} holds:

$$I_{LED} = I_s \times (e^{U_d / N \cdot U_t} - 1)$$

Where $I_s = 5.5 \times 10^{-15}$ A, U_d is the voltage across the diode, $U_t = 25 \times 10^{-3}$ V and $N = 2.3$.
Figure 4 shows what this looks like in the form of a graph. This is the forward voltage characteristic of a diode or LED, as supplied by the manufacturer. If the voltage across the LED is less than about 2 V, then the current is very small. When the voltage is greater than the threshold voltage the current increases quite quickly.

The LED can therefore be replaced by a voltage source U_{LED} with a voltage that depends on the current I_1 in the circuit (**Figure 5**). But we are not there yet. Unfortunately, Kirchhoff's Laws fail when there are non-linearities in the circuit, and a voltage source whose value depends on something else in the circuit cannot be made to fit in the equations. What we can do, is linearise the LED around an arbitrary point. The idea is that with small variations of the voltage or current we can consider the LED to be a linear element. Now we can apply the Laws of Ohm and Kirchhoff again and calculate all the voltages and currents in the circuit. This does not give us the right answer, but it does give us a better approximation. With this better approximation we adjust the parameters of the approximation and then calculate another, better, linear approximation of our diode at this new operating point. It turns out that after only a few iterations we already have a usable result.

We have now seen how SPICE deals with non-linear elements. These are replaced by circuits that are linear around a certain operating point. By repeating the calculations a number of times, SPICE can, in the end, also

find a solution for non-linear circuits. An operating point that 'fits' in the circuit.

Inductors and capacitors

If the circuit is powered from AC, then we can consider capacitors and inductors as complex impedances and simply apply Ohm's and Kirchhoff's Laws to determine the voltages and currents in the circuit. But it is not that easy when calculating the initial conditions.
Let us again take a simple circuit as an example: a voltage of 12 V, a resistor of 1 kΩ and a capacitor of 1 nF (**Figure 6**). When the voltage is switched on, there will be a current through R1. This current will charge C1, which causes the voltage across the capacitor to increase. The charging current will be reducing continuously. In the end the capacitor will be charged to 12 V. If we look at the state when the capacitor is charged to 4 V, then at that moment there will be 8 V across resistor R1. The current will be 8 mA. At this time the capacitor can be replaced with a voltage source. The change of voltage is written as

$$dU = dt \times i \, / \, C$$

So, if for 0.1 ms there is a current of 8 mA, the voltage across C will increase by

$$1 \times 10^{-7} \times 8 \times 10^{-3} \, / \, 1 \times 10^{-9} = 0.8 \text{ V}$$

After this 0.1 ms there is therefore a voltage of 4.8 V across C1 and 7.2 V across R1. We can now use these new values to make the calculations for the next 0.1 ms. In this way we can calculate the curve shown in **Figure 7**.
When using this simple iteration method the time steps have to be kept very small. This means that the PC has to calculate many steps, which could take quite some time. If we make the time step too small, then we could have additional rounding errors.
The time step is often variable. With large changes the steps are made smaller automatically and time steps increase again when changes are small. In practice the formulas in SPICE are a little different (trapezoidal integration is used), but the basic idea is the same.
In this example the capacitor was replaced by a voltage source. But it is also possible to work with a current source and a resistor as with the example of the diode. The principle remains the same: replace the part to be simulated with a combination of linear components and iterate to a solution.

Real-life components

Up to now we've been working with only ideal, theoretical components. Realistic components have a much more complex structure. A resistor is in reality a series circuit of a resistor with a parasitic inductance and both of these in parallel with a capacitance. With a transistor or opamp a large number of additional characteristics are added to that. In SPICE models there is a distinction between theoretical (virtual) and realistic components. The latter are usually sub-circuits where the total behaviour of the real component is replicated as good as possible. The user does not notice any of this, because with a transistor the same symbol always appears on the screen. Only when looking at the internal characteristics of the netlist will you

observe that there is a whole lot more than a theoretical, ideal transistor.

Most semiconductor manufacturers supply SPICE-models for their components, which replicate the characteristics of specific components as best as they are able to. Only with such accurate models is it possible to carry out a simulation whose output corresponds with the behaviour of a real circuit.

Tips and tricks

In SPICE, one node invariably has to be connected to ground. This is the **reference** node. When simulating a circuit, there is always an analysis of the initial conditions first. Things can already go wrong here, for example when you connect three capacitors in series. The middle capacitor can in theory have any arbitrary DC voltage level. This depends only on the initial charge of the capacitors. Even if the initial charge is equal to zero, the middle capacitor can have any arbitrary charge at the end.

When strange errors appear it can often help to swap the location of parts or connect a resistor of a few MΩ from a few strategic locations to ground, provided that is not a problem for the operation of the circuit.

When working with virtual components it is possible that the simulation of the circuit gives an excellent result. If you then use standard parts it may be that the circuit behaves strangely. This could be because the values of the components are not ideal any more, because you had to make a choice from the E12 series, for example. Particularly with filters the transfer function can be significantly different than what was theoretically calculated. The simulation program can help here as well, because with a so-called Monte-Carlo simulation you can specify a tolerance for all the components. In addition you can also subject the circuit to a real heat wave. The circuit will then be simulated with various combinations of component values and temperatures. The final result is shown as a graph with a large number of lines that are hopefully nicely one on top of the other.

When simulating a circuit with 'real' components, interesting things can be observed at higher frequencies. It is then possible that the stray self-inductance of a resistor (the connecting leads) is more significant than the resistance itself. Also, various parasitic capacitances (often only a fraction of a picofarad) can spoil the functionality of the circuit. You can easily add these to the schematic in certain places and then examine what the consequences are on the output signal.

When working with virtual opamps it is possible that the output voltage increases to more than 1 kV when there is a problem with the circuit. 'Real' opamps saturate to one of the power supply rails.

SPICE often has a dislike of transformer outputs that 'float'. So connect one side of the output of the transformer to ground. If that is not possible, a 1 MΩ resistor can often do wonders.

Note carefully: some SPICE programs do not know the difference between m and M, but do understand MEG. A resistor of 1 mΩ has a very small resistance! When in doubt use 1000 kΩ or 1 MEG.

With SPICE it is also possible to build circuits that are not practicable in the real world. Try, for example, to make an inductor with a self-inductance of 100 H and a resistance of only 0.01 Ω. Even transformers with a flux density of 1000 T (tesla) don't pose a problem for SPICE. Finally: it is possible that an apparently simple circuit
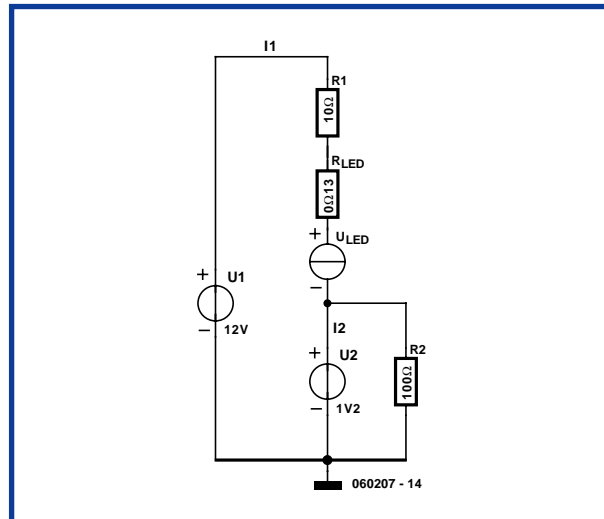

Figure 5.
The LED is replaced with a voltage source and internal resistance in order to calculate the voltages and currents in a circuit at a certain operating point.
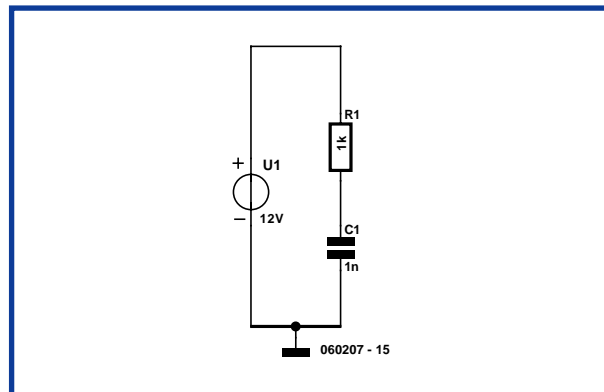

Figure 6.
The power-on behaviour is examined based on this simple schematic: a voltage source charges a capacitor via a resistor.
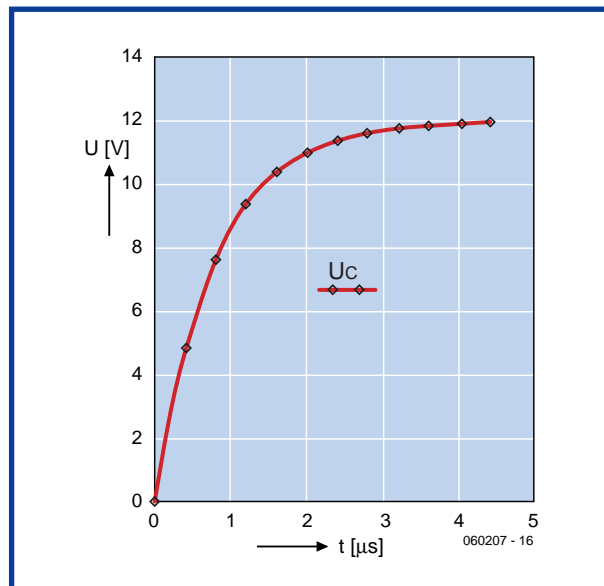

Figure 7.
The calculated progress of the voltage across the capacitor.

does not work. After you have checked all the connections and if it still doesn't work then it may be useful to go to the Internet and ask for help on a forum. Some manufacturers are very helpful. They have a vested interest in knowing what does and doesn't work in practice. Let's be honest here: us users — whether we want to or not — are always participating in one big beta test!

(060207-1)

19

Simulation

Harry Baggen & Thijs Beckers

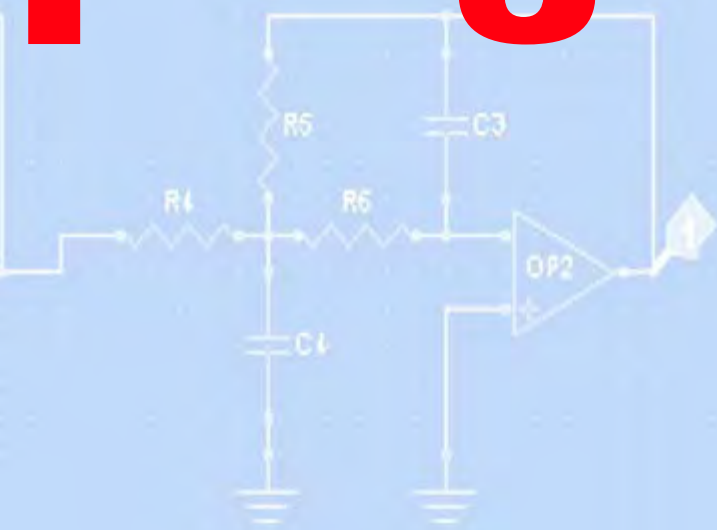## Overview of the simulation programs on the DVD in this issue

The personal computer has played an increasingly important part in the design of electronics circuits. Dozens of programs are now available, not only for creating PCB layouts, but also for simulating circuits. With this issue of Elektor Electronics we offer you a free DVD with a large number of demo, evaluation and even full versions of virtually all the popular simulation programs. With this DVD you can try out various programs on your PC and decide which one is best suited for you. In this article we give you an overview with a short description of the programs on the DVD.

All programs on this DVD are strictly for non-commercial use only!

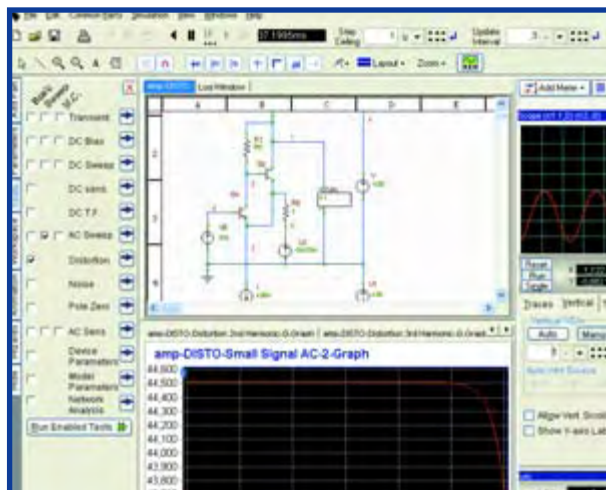|	
5Spice 1.22	5 Spice Analysis Software
AIM-Spice 4.3	AIM-Software
B2Spice 5.1.6	Beige Bag Software
Boardmaker 3	Tsien
Cadstar Express 8.0	Zuken
CIRSIM 3.0	Bells-Hill
DesignWorks Professional 4	Capilano Computing
Easy-PC/Easy-Spice 10	Number One Systems
eSketch Pro 1.5	Schematica Software
iSim	Inca Systems
LTSpice/SwitcherCAD 3	Linear Technology
Micro-Cap 8	Spectrum Software
Multisim DesignSuite 9	Electronics Workbench
OrCAD 10.5	Cadence
Profilab-Expert 4.0	Abacom
Proteus 6	Labcenter Electronics
PSIM 7.0	Powersim
SIMetrix 5.2	Catena
SIMWinXP 1.1	Visionix
SMASH 5.7	Dolphin
Sonnet-Lite 10.51	Sonnet
SpiceAge	Those Engineers Ltd
SpiceCreator Pro 5	AMS
Target3001! V12	Ing-Büro Friedrich
TopSPICE/Win32 V7	Penzar Development
Visual Spice 6	Island Logix
Win-Elektronik 3.1	Erwin Rössler

programs

Zoom Default

Expand Y Axis

✓ Logaritmic Frequency Axis

Expand Freq Axis

Contract Freq Axis

✓ Auto fit Y axis

✓ Auto fit Y2 axis

Configure Axes

Graph Colors

Thick Graph Lines

Save Graph as Bitmap

Save Graph as Metafile

Missing DVD?
Ask your newsagent!

200   500   1 k   2 k   5 k   10 k   20 k   50 k   100 k   200 k   500 k   1 M

Professional electronics designers have to manage their time very efficiently. Nowadays it is hardly worthwhile to design a circuit on paper and then build one or more prototypes to check that the design functions correctly. The ubiquitous computer (usually a PC) has been promoted a long time ago from a simple calculating tool to an intelligent assistant for circuit design, thanks to a number of clever programs. We already know of PCB CAD programs that simplify the design of boards. They have extensive libraries with package details, place the components onto the board automatically and have autorouters that find the best track layout on the board. Human insight and supervision is of course still a necessity for a successful design, but it is a huge improvement over the manual design of PCB artwork. Engineers also rely more and more on the computer for the design of circuits. The availability of good simulation programs have made it possible to see a realistic simulation of a circuit, once its schematic has been entered. When the circuit in the simulation program functions as expected, it should also do so in practice for 95 to 100% of the time. It is then usually sufficient to make a single prototype for a final test.

These programs are also very useful for students and hobbyists. When you have a brainwave you can quickly input a circuit and check that it behaves as expected, all without touching a soldering iron, components or measuring instruments. Furthermore, you will also gain a better insight in the workings of components and circuits when you create these simulations.

How it all started

The simulation of electronic circuits on computers finds its roots in the creation of SPICE, a program that was developed at Berkeley University. In this issue you'll find an accompanying article (The PC as Breadboard) that tells you more about the history and workings of SPICE. Virtually all current programs are based on this. SPICE can be used to simulate linear and non-linear electronic components. Following this, the Georgia Tech Research Institute developed XSPICE, which is used to create model libraries with extensive component definitions. Both SPICE and XSPICE are completely open source and are therefore ideally suited as a basis for simulation programs. Other well-known developments are Cider (digital simulation) and Ngspice (mixed-mode simulation).

Most of the modern simulation programs work under Windows because this is the most common platform these days. The biggest improvement of these programs has been in the user-friendliness of the interface. In older programs this could be a bit of a nightmare (draw a circuit on paper, number all the nodes, search for the correct models and create netlists, then use a different program module to input what should be calculated at each node, which, if you were lucky, could be shown graphically). These days you can use the mouse with most programs to create a circuit, and a few well-designed menus help you input the simulation settings. This is followed by the display of several clear graphs. The better programs

Special editions

Several suppliers have supplied us with special editions of their programs that have a wider range of functions than their standard demo programs available from their websites.
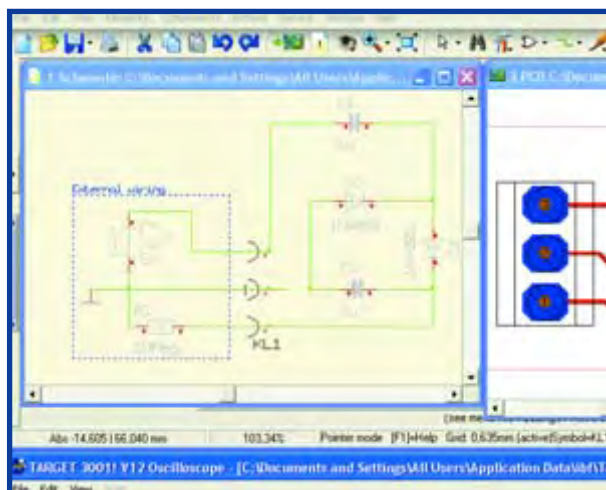
SpiceAge & **Spicycle**, made by the British firm Those Engineers, are on the DVD as special releases with a limited component library. Apart from that they are fully functional and can be used without a time limit (Spicycle Level 1).

Target 3001! from Ing. Büro Friedrich is on the DVD as a special 'light' version with a value of 49 euros (approx. £ 34). This can cope with up to 400 pins/pads and 2 track layers. You won't be able to use the extra component library from Target, but that doesn't matter too much since the included library contains about 1,000 standard components.

We should remind you that all programs are strictly for non-commercial use only.

With thanks to...

We would like to thank all suppliers for their help in making this DVD, in particular Linear Technology for allowing us to include LTSpice/SwitcherCAD, and Those Engineers and Ing. Büro Friedrich for their special editions.

also have extensive component libraries, which contain the properties of most semiconductors made by the larger manufacturers. There is no longer a need to study for days just to find out how a simulation program works. After a few hours you should be able to use it fairly well. Most programs are so-called mixed-mode simulators, which can be used to simulate analogue, digital and mixed circuits.

Types of programs

There are several types of simulation program available, and you'll find some of each on the DVD.

There are complete design suites that take care of the schematic input, simulation and track layout, such as MultiSim. With these packages you don't need any other software. These programs are usually fairly expensive because of their complexity, but the advantage is that there is a close integration of the various sections, which are controlled sequentially from one program and support is obtained from a single source.

Then there are pure simulation programs. Some have a full graphical input and output, such as Micro-Cap, others don't have the graphical input facility but instead make use of a separate schematic capture program (e.g. AIM-Spice). Some programs have been designed in such a way that they can be integrated with an existing PCB program (e.g. Easy-Spice in Easy-PC). Lastly, there are interface programs, such as iSim, which translate the output of a schematic capture program into a form that can be used by a SPICE program.

The DVD

We have tried to include as many simulation programs on the DVD as possible. We obtained permission from various suppliers to distribute about 30 programs on our DVD. The programs on the DVD have been stored in several different folders. The largest number can be found in the Windows folder. Almost all these programs can run under Windows 98/ME/XP. The majority of the programs have an English user interface and a few offer a choice of languages.

The Linux folder contains a few interesting simulation programs for Linux, as the name suggests. A few programs from the Windows folder are also found here because they have been made available as a Linux version as well. A separate open-source Windows folder contains programs that can be freely used and distributed.

The final folder called Extra contains a few special programs that deviate from normal simulation programs, such as a program to simulate magnetic fields and a program to help calculate component values in filters and timers.

We'll now give you a short description of the major programs on the DVD, along with any restrictions of the supplied version and the amount of space each program takes up on your hard drive.

We hope you enjoy trying out the various programs!
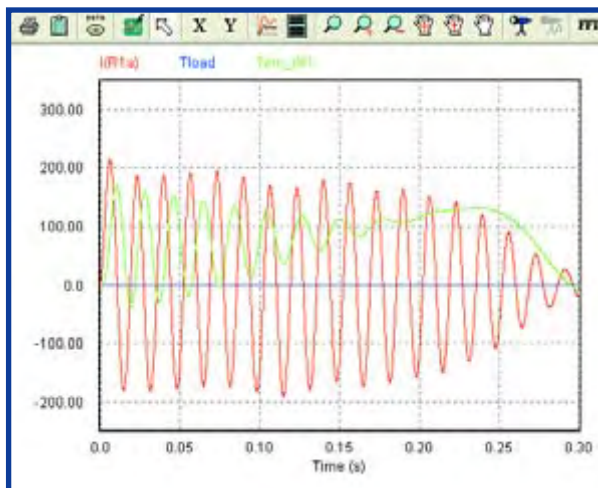
Now it's your turn!

We have done our best to include the most recent versions of the programs on our DVD, but suppliers continually strive to improve their programs. If you are interested in a specific program it is always advisable to look on the supplier's website to see if a more recent version is available.

We have included a list of other interesting products that haven't made it to the DVD for one reason or other (e.g. no permission was given to distribute the software). It's certainly worthwhile to have a look at these as well.

The larger suppliers usually have a national distributor or an organisation that can give you further information about their products, which is more convenient since you can talk to them in your own language. They can also give you details of pricing and support for their software.

Finally, we thought we should include the well-known disclaimer: we have tested all Windows programs on several PCs, but we can't guarantee that they will run faultlessly on your system. Should you have problems with the installation or running of programs, please contact the relevant supplier directly. We are unlikely to be able to help you out in these matters!

Now it's your turn to try out the collection of software on this DVD and see what potential they have. Simulation is the future, that much is certain. And this DVD is a perfect introduction to this subject. Brought to you free of charge by Elektor Electronics.
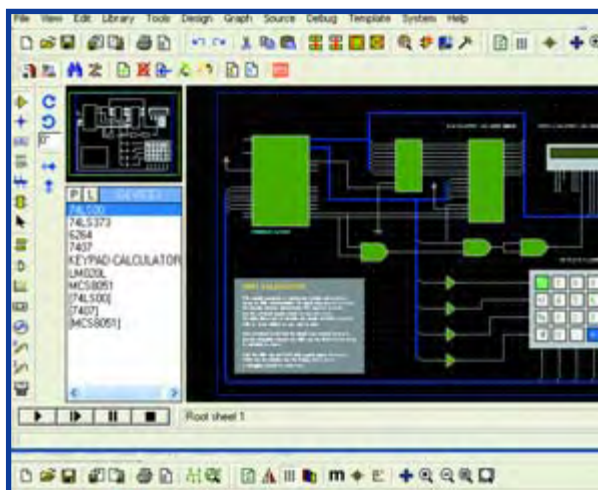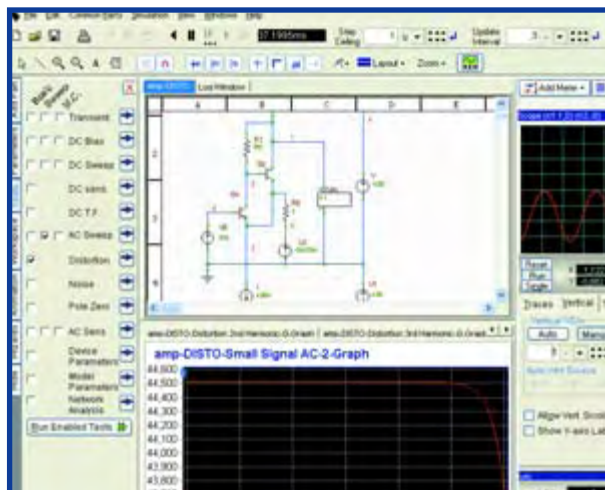
5Spice 1.22 (14 MB)

This program is a graphical shell built around a tradi-
tional Spice emulation engine. A schematic editor is used
to enter the electronic design, which can then be simu-
lated using Spice version 3f4/3f5.
The operation is quite simple. The program is not very
comprehensive, but can still simulate most types of circuit.
The size of the schematic is restricted in the demo version
and the numerical output of several types of analysis is
not possible. Another restriction is that schematics with
logic gates cannot be stored and a demo text is added to
all other schematics.

AIM-Spice 4.3 (10 MB)

This program doesn't have a schematic input, but is just a
SPICE-simulator. The software 'suite' consists of two pro-
grams: AIM-Spice, with a text editor for editing the Spice
netlist and simulation options, and AIM-Postprocessor,
which is used to display stored data files graphically.
Once you've become used to the textual input for a
schematic the program will be easier to use. A large
number of simulation parameters can be adjusted.
The student version is limited to 150 nodes and a maxi-
mum of 30 transistors per circuit.
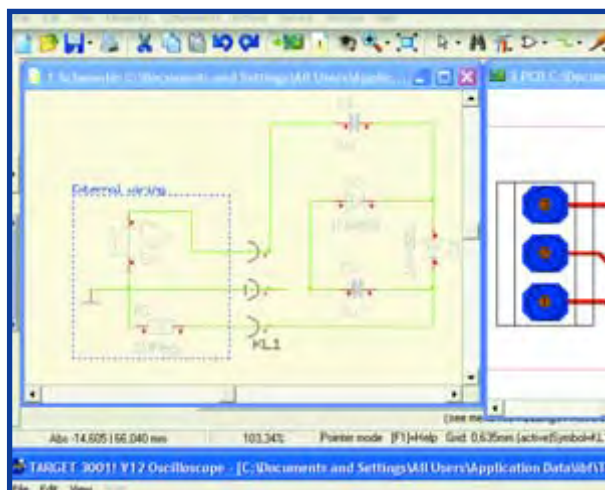
B2Spice V5.1.6 (131 MB)

This program is of particular interest to Eagle users
because Eagle schematics can be directly imported and
simulated. You can also draw the schematic with B2Spice
itself. The user interface of this program is clear and intu-
itive. Simulating is very easy. Virtual instruments are used
to place probes onto the schematic in real-time.
The trial version can be used without restrictions for 45
days.

Boardmaker 3 (134 MB)

Boardmaker 3 is a complete CAD package. You can
draw schematics, simulate and create print layouts. PCB
layouts can even be shown in 3D.
The operation of the program is quite complex, mainly
due to the many functions it can perform. There are tutori-
als in PDF format (they're stored in the same folder where
the program was installed, but strangely enough there
are no shortcuts to them in the start menu). It is strongly
recommended that you go through these before you start
using the program. The demo version can't print or store
schematics, nor can it output Gerber files.

CIRSIM2006 (5 MB)

CIRSIM is a fairly straightforward program that is only
suitable for simulations with continuous input signals. The
description of the schematic can only be entered using
SPICE-code, i.e. there is no graphical input. Because the
program has only a few functions, its operation is self-evi-
dent. Registration of the program is just £10.
The demo version of the program is severely restricted,
with only six nodes maximum.

DesignWorks Professional 4 (38 MB)

With DesignWorks Professional you can simulate digital
circuits in a simple manner. The design can be input as
either a schematic or using VHDL. The operation has an
intuitive and logical feel to it. However, if you get stuck
you can refer to an excellent manual in PDF format.
DesignWorks can also be used to create analogue
schematics, but these can't be simulated.
The demo version has full functionality for 30 days.

Easy-Spice (& Easy-PC)

Easy-Spice is an enhancement for the print layout pro-
gram Easy-PC (both are on the DVD). You should install
Easy-PC first, then Easy-Spice.
Easy-PC can be used to draw electronic schematics and
PCBs, set simulation parameters and start a simulation.
The program then creates a netlist and starts Easy-Spice.
Easy-Spice automatically opens the netlist and uses it for
the simulation.
The folder …\Easy-PC Demo\Examples\SPICE contains a
few examples. Both analogue and digital circuits can be
simulated. The demo version of Easy-PC can't store files
or create CAM outputs. There are no known restrictions
in the demo version of Easy-Spice. Both programs require
a password to function, which is kp69ny31 for Easy-PC
and wa32pk65 for Easy-Spice.

eSketch (5 MB)

This (small) program has a polished look and is very
easy to use. You can draw passive analogue schematics

with it and simulate them. Unfortunately you can't use it to enter logic gates, transistors, diodes or other active elements.
You can try out the free version for 15 days without restrictions.

LTSpice/SwitcherCAD III (77 MB)
The program SwitcherCAD III supplied by Linear Technology can be used to simulate (almost) every switchmode regulator made by LT. The simulation even includes the power-up cycle. Several useful example circuits are included and you can of course input your own circuit as well. The program may not have a polished look, but its functionality leaves nothing to be desired.
LTSpice/SwitcherCAD III is completely free of charge.

Micro-Cap 8 (22 MB)
Micro-Cap is a well-designed simulation program with a schematic capture and a well-sized standard component library. The abbreviations used in the simulation setup are a little bit confusing at the beginning, but once you're used to them they are very easy to work with.
The evaluation version of Micro-Cap is limited to 50 components and 100 equations (nodes, inductors and signal sources). The simulation speed is slower and the number of optimisations, filter designs and 3D plots has been reduced.

Multisim 9 (191 MB)
Multisim is a complete design suite that comprises schematic capture, simulations and PCB design (Ultiboard). It's one of the most comprehensive packages that we've come across. Smart virtual instruments can be placed onto the schematic, which then show the simulated signals. The component library of Multisim is very extensive. The program is now also capable of exchanging measurement and simulation data with LabView.

The freeware version requires an Internet connection to simulate a circuit. After 45 days the simulation and autorouting functions stop working. Furthermore, the designs limited to 50 components, 750 pins and 2 layers.

OrCAD 10.5 (707 MB)
This is another comprehensive design suite! OrCAD creates a number of shortcuts for the various programs in the package. The main program is Capture CIS. This program acts as a type of manager for all the files that make up a design. There is also a tutorial (OrCAD-directory\OrCAD_10.5_Demo\tools\capture\tutorial\CAPTUTOR.EXE). The simulation section (PSpice A/D) isn't controlled directly by Capture CIS and so you need to point it to your project manually. The many functions of the package make it somewhat difficult to use, so a bit of training is to be expected before you can fully use it. There is no time limit in the demo version, but there are restrictions on the size of designs.

ProfiLab Expert 4 (17 MB)
ProfiLab Expert looks similar to Labview, but is a lot simpler. Various dials, displays and other functional blocks can be added to a design via a clear, well-designed
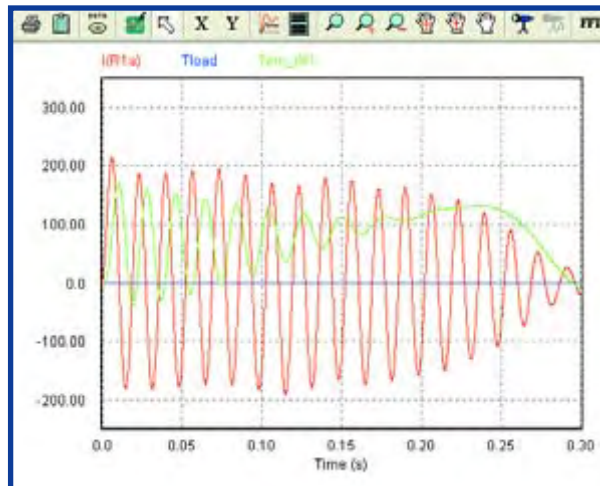
screen. The simulation then shows the results for your design.
The demo version can be used with a maximum of 10 components. You can't store or compile your designs and the simulation time is limited to 30 seconds.

Proteus 6 (112 MB)
Proteus 6 consists of 2 programs: ISIS 6 and ARES 6. ISIS is used to draw schematics and simulate them; ARES is used to design the PCB layout. It is very easy to simulate a circuit with ISIS (play button at the bottom-left). The help files included are very clear, as is the comprehensive user interface. What makes Proteus special is that mixed-mode simulations can include microprocessors, which have their code executed during the simulation.

The demo version does not allow printouts or saving of files. Neither can you input your own microcontroller designs, but you can modify the examples included.

SIMetrix 5.2 (34 MB)
SIMetrix can be used to input and simulate analogue and digital circuits. Despite the separate program windows the package is easy to use. The various settings are
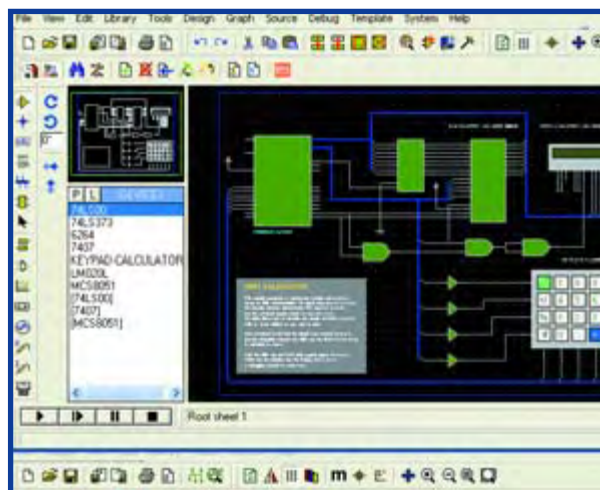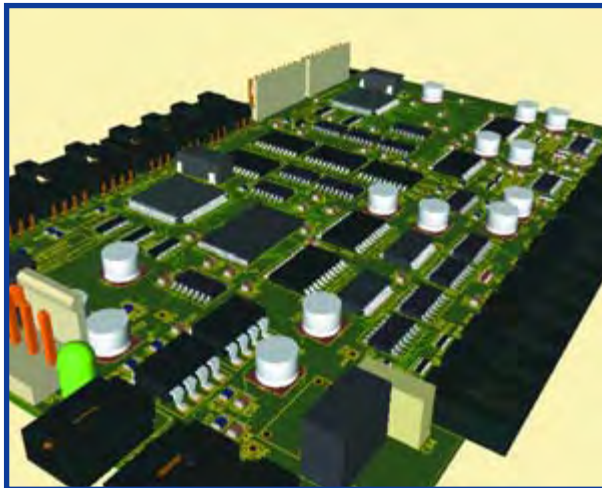
quickly and intuitively found.
When installing the program you should choose SIMetrix Intro. SIMetrix uses five characters for extensions and so avoids any potential problems with other programs.
The demo version is only limited to the number of components that can be used.

Suppliers' websites

5 Spice Analysis Software	www.5spice.com
Abacom	www.abacom-online.de
AIM-Software	www.aimspice.com
AMS	www.advancedmsinc.com
Beige Bag Software	www.beigebag.com
	Benelux: www.franklin-industries.com
Bells-Hill	www.bells-hill.freeserve.co.uk
Cadence	www.cadence.com/orcad
Cadmigos	www.cadmigos.com
Capilano Computing	www.capilano.com
Catena	www.catena.uk.com
Dolphin	www.dolphin.fr
Electronics Workbench	www.electronicsworkbench.com
Erwin Rössler	www.win-elektronik.de
Inca Systems	www.incasystems.fi
Ing-Büro Friedrich	www.ibfriedrich.com
Island Logix	www.islandlogix.com
Labcenter Electronics	www.labcenter.co.uk
Linear Technology	www.linear.com
Number One Systems	www.numberone.com
Penzar Development	http://penzar.com
Powersim	www.powersimtech.com
Schematica Software	www.schematica.com
Sonnet	www.sonnetusa.com
Spectrum Software	www.spectrum-soft.com
Those Engineers Ltd	www.spiceage.com
Tsien	www.tsien.info
Visionix	www.visionics.a.se
Zuken	www.zuken.com

If you want to use the program for switchmode supplies you should choose the SIMetrix/SIMPLIS option during installation. SIMPLIS is between 10 and 50 times faster in simulating these, compared to the extensive Spice simulation of SIMetrix.

SIMWinXP 1.10 (261 MB)

SIMWinXP is the little brother of the EDWinXP design suite. It is a standalone program for drawing the circuits that can then be simulated using the included mixed-mode or EDSpice simulator. With EDSpice you can also simulate circuits that have been created in Spice. SIMWinXP can cope with both analogue as well as digital designs.
To install SIMWinXP you should run Setup.exe from the SIMWinXP folder.
The evaluation version is fully functional, but works for only 30 days.

Smash 5.7.0 (424 MB)

Smash is a powerful mixed-mode simulation program without a schematic capture input. The program has some unusual features, such as estimating the power consumption of digital circuits and a mixed-mode facility of SPICE and VHDL-AMS.
The required netlists can be loaded as .cir, .nsx or .sp files. The program can work with several other popular programs from e.g. Matlab, Keil and National Instruments.
We were pleasantly surprised by the large number of PDF documents that were included, with details of various design problems and simulation methods.
The evaluation version has a limit of 25 analogue nodes.

SpiceAge & Spicycle (49 MB)

Before you can install SpiceAge (simulation) and Spicycle (schematic capture) you should copy the complete SpiceAge folder from the DVD to the C:\ drive, including the folder structure.
Spicycle is used to draw schematics and PCB layouts. You can start a simulation from Spicycle that is then run in SpiceAge. From SpiceAge you can set many more parameters and can also start different simulations. The user interface is clear and well designed. Most of the control icons can be reached via the menu bar.
The demo versions have been specially provided for Elektor Electronics and have a limited component library, but are otherwise fully functional and don't have a time limit.

Spice Creator Pro V5.12 (39 MB) & Visual Spice (39 MB)

These two programs are like peas in a pod.
Spice Creator and Visual Spice are used to create and simulate analogue and digital circuits. There is a useful help-browser, which makes it easy to find instructions on using the program. The main program screen overflows with icons, but this can fortunately be reduced.
The trial versions have quite a number of restrictions: there is no undo, save, export, print, copy/paste and simulation of edited circuits.

Target 3001! V12 (61 MB)

Target 3001! is a CAD program with various extra features. In the first instance it is a program designed for

inputting schematics and creating print layouts. The design can be simulated and it can also check for EMC issues. It also offers the facility for the design of front panels.
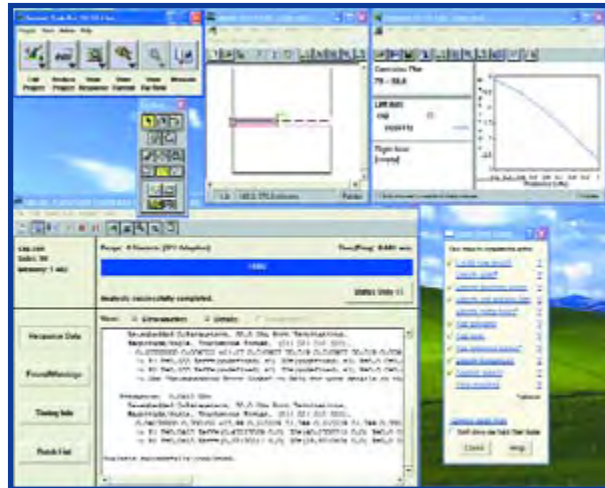
To use all of the features to their full potential you will have to read the manual first. Although it is possible to use the program straight from the box, you'll need a bit of study and practice before you can grasp the finer points of this program.
A special version has been made available to *Elektor Electronics*, which offers more than the standard demo version: Target 3001! V12 'light' (400 pins/pads, 2 track layers, costing 49 Euro), which can't access the component library on the Target server.

TopSPICE 7.09g (23 MB)

TopSPICE can be used to open both schematic diagrams as well as Spice netlists. Simulations are fast and easy. It is self-evident how to use the program, but if you should get stuck you'll find many 'help' and 'getting started' documents to help you out.
There are several limits imposed in the demo version, such as on the size of the schematic, the number of nodes, transistors, top-level components, data points per plot and a maximum data memory of 1 MB.

Win-Elektronik 3.1 (1 MB)

This simple (German language) program is perfect for checking how an analogue network functions. It is mainly intended for students and schools. It has a limited set of features.
The demo version has a maximum of 8 components and 1 opamp.

Extras

-iSim (14 MB)
(& CADStar Express 8 & PSpice A/D 10.5)

iSim is an interface between schematics drawn with CADStar and the SPICE simulations in PSpice A/D. You can run iSim from within CADStar Express to start the simulations. The results are then shown by PSpice A/D. This plug-in has been well designed and leaves nothing to be desired. The demo version is limited for use with up to 50 components.

PSIM (22 MB)

PSIM is intended mainly for simulating power circuits and motor drivers. It consists of two programs: PSIM and SimView. PSIM is used to draw the schematic and to start a simulation. SimView then automatically takes over and shows the simulation graphs.
The operation of the programs doesn't really require an explanation. The examples that come with the programs are very useful as they give you a quick insight how the programs work and what they're capable of.

Sonnet Lite 10.51 (91 MB)

The Sonnet software is suitable for (nearly) all calculations and simulations that are concerned with RF electromagnetic interference. A few examples of the capabilities

of the program are the analysis of PCB track crosstalk, microstrips and coupled transmission lines. The free Lite version has a few restrictions compared with the full version, but you can still evaluate the latter for 30 days. You can find more information on the supplier's website.

(060206-I)

Interesting links

Altium Designer	www.altium.com/Products/AltiumDesigner/
Tina Pro 6	www.designwareinc.com/index.shtml
NGSpice	http://ngspice.sourceforge.net/relapp.html
AnaSoft SuperSpice	www.anasoft.co.uk/
APLAC	www.aplac.hut.fi/aplac/
AKNM Circuit Magic	www.circuit-magic.com/
Intusoft ICAP/4	www.intusoft.com/demos.htm
PC-ECAP	www.cdquickcache.com/pcecap.htm
Digital Simulator	www.mit.edu/people/ara/ds.html
Spice+	http://spicep.sourceforge.net/
WinEcad	www.winecad.com/winecad.htm
DxAnalog	www.mentor.com/products/pcb/expedition/ analysis_verification/dx_analog/index.cfm
NGSpice	http://ngspice.sourceforge.net/
+ GSpiceUI	www.geda.seul.org/tools/gspiceui/index.html
PSpice 3f4	www.ee.washington.edu/circuit_archive/software/ spice3f4.tar.gz
Spice3f4 (Mac)	www.kivadesigngroupe.com/Kiva%20Professional/ professionalpage.htm
MacSpice 3f5 (Mac)	http://newton.ex.ac.uk/teaching/CDHW/MacSpice
Pulsonix	www.pulsonix.com/index.asp
CSiEDA	www.csieda.com/
Crocodile Technology 6.01	www.crocodile-clips.com/crocodile/technology/ index601.jsp
Qucs	http://qucs.sourceforge.net/news.html
Simplorer Student Version	www.ansoft.com/about/academics/simplorer_sv/ index.cfm