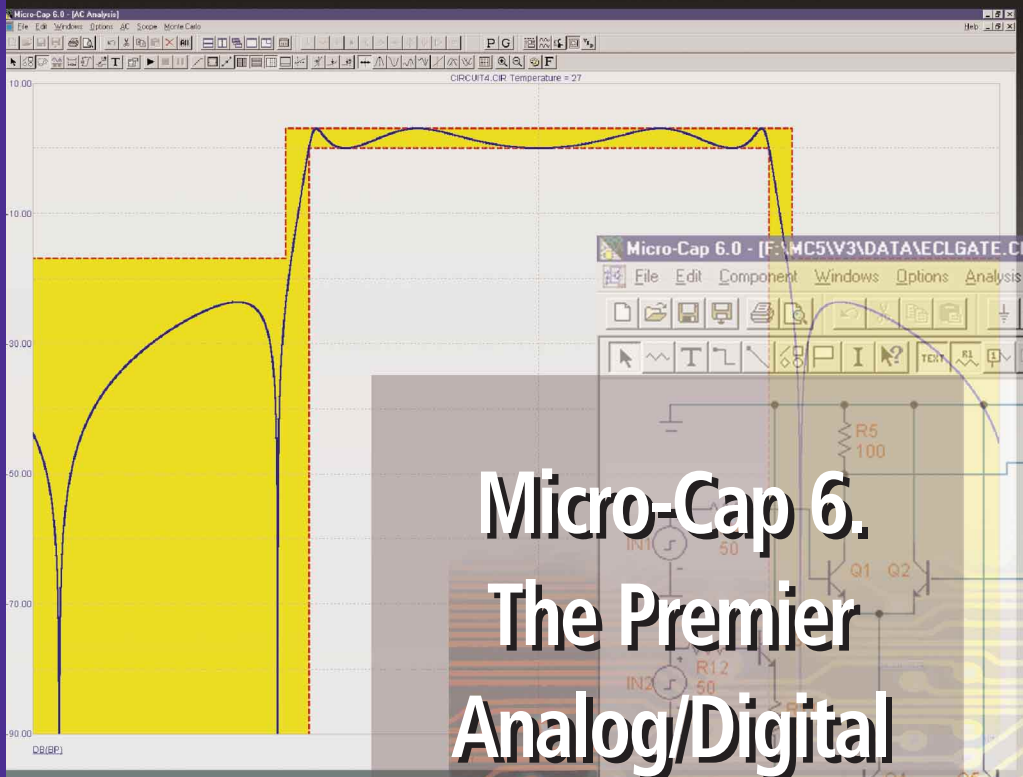
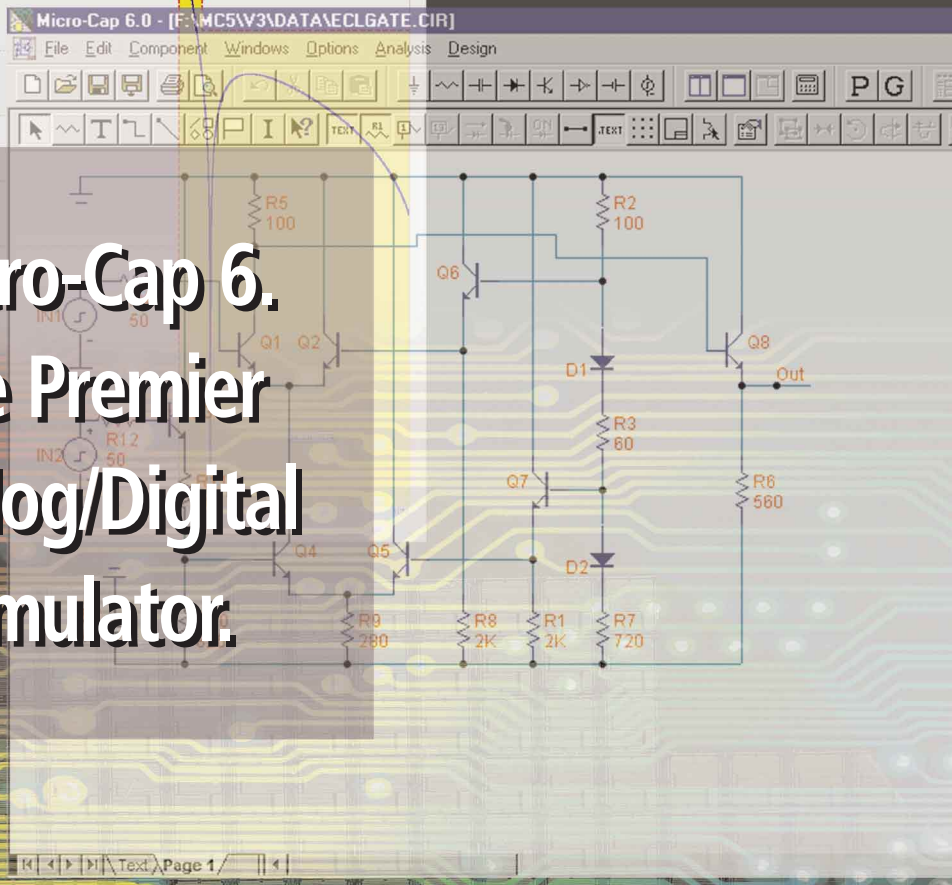




Micro-Cap 6. The Premier Analog/Digital Simulator.

It just keeps getting better.

MICRO-CAP 6

6

Micro-Cap 6...

The Sixth

Generation

Micro-Cap 6 is an integrated schematic editor and mixed analog/digital simulator that provides an interactive sketch and simulate environment for electronics engineers. Since its original release in 1982, the Micro-Cap family has been steadily expanded and improved. Micro-Cap 6, the sixth generation of that family, blends a modern, Windows-based user interface, the robust and powerful numerical algorithms of SPICE3, and a fast, native, PSpice® compatible, digital simulator to produce the most powerful and easiest to use circuit simulator available for personal computers. It is affordable as well, costing less than half of comparable programs.

Micro-Cap 6 delivers measurable advantages, because it's...

Fast

Algorithmic improvements, optimized code, and an integrated, seamless, analog/digital simulation interface contribute to the stunning speed of Micro-Cap 6.

Powerful

Numerous features contribute to the power of Micro-Cap 6. Among them are:

- Multi-page hierarchical schematic editor.
- On-line simulator. No need to load another program.
- Integrated active and passive filter design function.
- PCB interface to popular packages.
- LAN version for collaborative projects.
- Native PSpice® compatible digital simulation engine.
- Device library with over 12,000 parts.
- Analog and digital behavioral modeling.
- Schematic waveform probing.
- Dynamic on-schematic voltage/state, current, power, and condition display.
- During the run plotting.
- Powerful plotting and post-analysis functions.
- Multidimensional parameter stepping.
- 3D plotting.
- Performance functions and plots.
- Monte Carlo analysis.
- Optimizing parts modeler.
- BSIM 1.0, BSIM 2.0, and the latest BSIM3 version 3.2 device models.
- Animated devices (7 segment displays, LEDs, and movable switches) provide visualization and interaction.
- Sample and hold and Z transform devices.
- Lossy transmission lines.
- Jiles-Atherton nonlinear magnetics model.
- GaAsFET models.

Easy to use

The graphical user-friendly interface is simple to learn and use. Familiar SPICE and PSpice® models, plus extensions, are easy to apply. Over 500 warning messages help you through problems as they arise, not later in a text file. Micro-Cap 6 gives you immediate feedback by plotting waveforms during the run.

Affordable

You can easily spend twice the cost of Micro-Cap 6 for other simulators without matching its power, speed, and ease of use.

Guaranteed

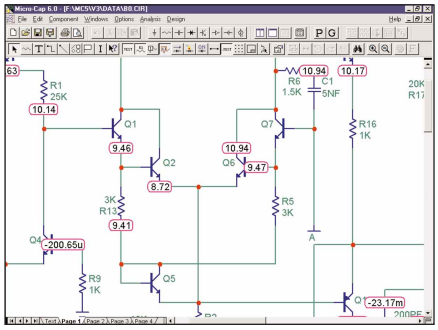
Our 30 day, unconditional, money-back guarantee makes Micro-Cap 6 easy to try.

MICRO-CAP 6

MICRO-CAP 6

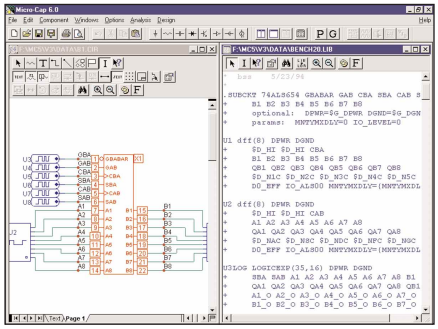
PRINCIPAL FEATURES

Integrated schematic editor and simulator.
The multi-page hierarchical schematic editor makes it easy to sketch a circuit. Once a circuit is created, you can do a Transient, AC, DC, Transfer Function, or Sensitivity analysis with two key



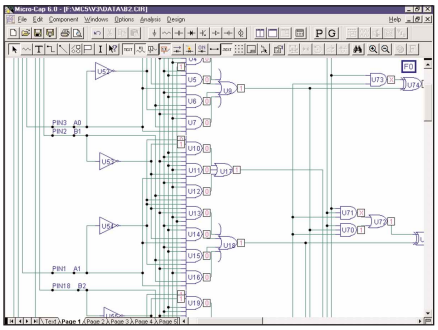
presses. Editing the schematic and rerunning an analysis is just as easy. The editor features stepping, scaling, panning, multiple-object selection, rotation about three axes, mirroring, and clipboard functions. You can copy and paste between circuits using the clipboard. You can probe the schematic with the mouse to display circuit waveforms and curves, or display DC voltages, currents, and power directly on the schematic.

SPICE compatible models and simulator.
Micro-Cap 6 reads, writes, creates, and analyzes standard SPICE and PSpice® text files as well as its own schematic files. You can use the schematic editor



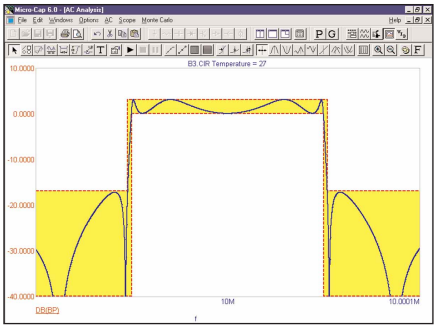
to create schematics or the text editor to build SPICE text file circuits. Micro-Cap 6 can analyze either format and can use text file subckt models in schematics. Micro-Cap 6 can create SPICE 2, SPICE 3, and PSpice® circuit files from its own schematics.

Native digital simulator with PSpice® compatible syntax.
The internal native, 5-state, event-driven digital simulator lets you run digital



or mixed mode simulations using your own models or those from our extensive digital library.

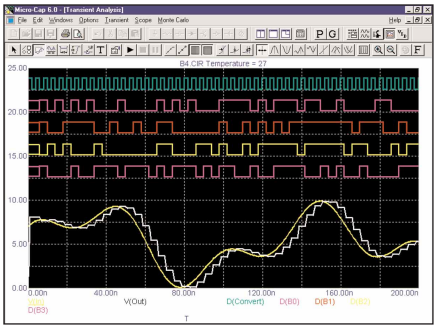
Active and passive filter designer.
The integrated filter designer creates low pass, high pass, bandpass, notch,



and delay filters with Butterworth, Bessel, Chebyshev, inverse-Chebyshev, and elliptic responses.

ANALYSIS FEATURES

Transient analysis–for plotting any time-domain waveform.
Available variables include digital state, voltage, current, power, charge, capacitance, inductance, B field, and H field. A variety of variables and mathematical



6

functions are available to simplify plots. Here are some examples:

PD(Q1): Power dissipation in Q1.

SUM(PD(Q1),T): Energy in Q1.

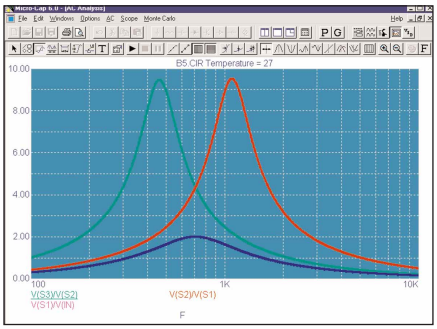
HARM(V(A)):Waveform A's harmonics.

RMS(v(1)*exp(-T/6n)): An RMS value.

D(A): The digital state on node A.

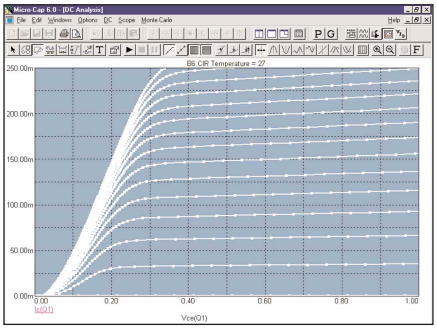
HEX(1,2,3,4): Hex value of nodes 1 to 4.

AC analysis–for investigating the small signal behavior of your circuit.
With AC analysis you can plot voltage or current and produce Bode plots, Nyquist diagrams, Nichols charts, impedance plots, and noise plots. Other



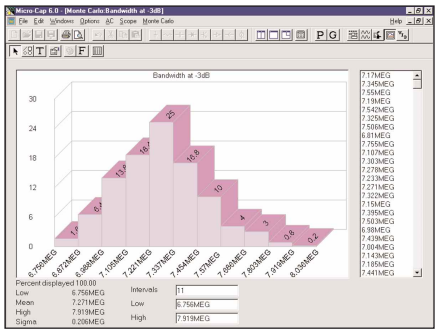
operators–including real part, imaginary part, magnitude, phase, and group delay–make analysis and plotting easy. The automatic frequency step control plots smooth curves with a minimum of calculation, greatly speeding analysis.

DC analysis–for plotting static DC variables.
You can use DC analysis for various plots, ranging from transfer function



curves, where one source is varied, to device IV curves, where two sources are varied. Transfer function plots help to determine DC offset, bias, and overall amplifier gain.

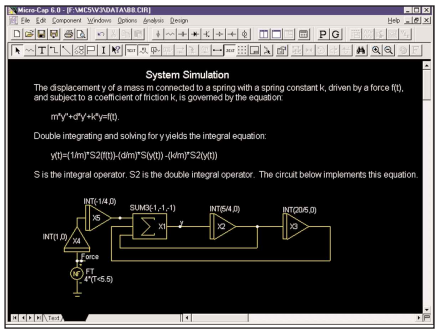
Monte Carlo analysis–for design centering and yield optimization.
Integrated Monte Carlo routines construct hundreds of circuits, each containing parts with parameters picked from distributions you choose. This helps identify circuit problems and



improves production yield of the circuit. You can use both absolute and relative tolerances and worst case, Gaussian, or uniform distributions. For easy inspection and review, results are generated in both numeric and histogram form. You can set performance criteria, and reconstruct circuits with parameters that failed during a Monte Carlo run.

MODELING FEATURES

Analog behavioral modeling for system and block-level simulation.
Laplace sources let you describe the S-plane linear transfer function of a circuit block. Function sources let you



model instantaneous nonlinear behavior. Expressions can also be used for resistor, capacitor, and inductor values. Here are some samples:

G*b0/(s**2+b1*s+b0): low pass filter

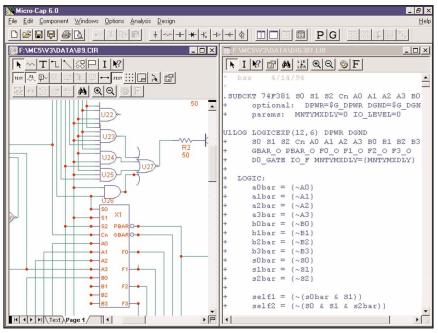
exp(-c*s*(r+s*l)**.5)): transmission line.

-k*(v(p)-v(c)+u*(v(g)-v(c)))**1.5: a triode.

VZ+tempco*(TEMP-27): reference source.

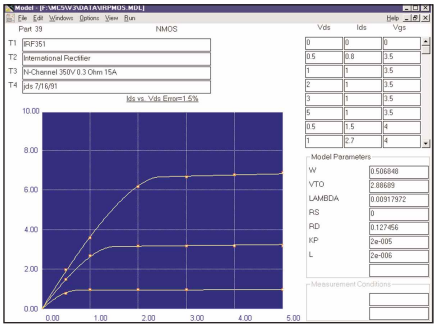
Sin(2*pi*T)*Exp(-T): damped sine wave.

Digital behavioral modeling–for rapid modeling of new digital functions.
Powerful digital behavioral primitives



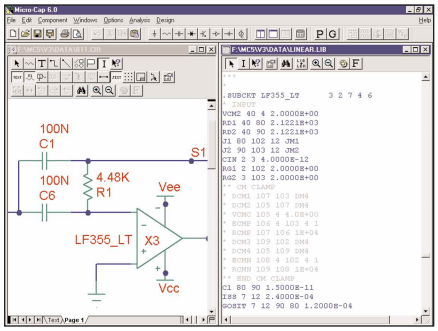
include logic expressions, pin delay blocks, and constraint checkers.

Optimizing MODEL program–for fast creation of optimized device models.
If you can't find what you need in our large library, you can use MODEL to



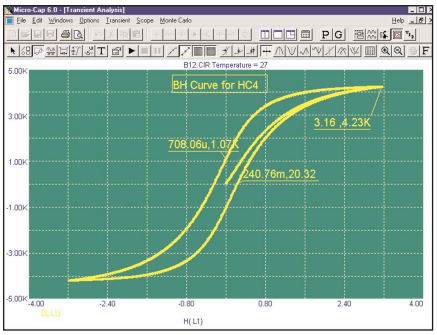
produce optimized model parameters from data sheet values or graphs.

SPICE subckt models–for easy access to manufacturer's modeling work.
Micro-Cap 6 directly uses SPICE subckt models provided by many semiconductor manufacturers for their devices. They are represented in schematics with



the subckt component. This feature provides easy access to the substantial modeling work done by the parts manufacturers.

Nonlinear magnetics model–for realistic simulations of nonlinear magnetic materials.
Using the Jiles-Atherton magnetics model provided by Micro-Cap 6, you can analyze the nonlinear behavior of



cores, reactors, and power transformers.

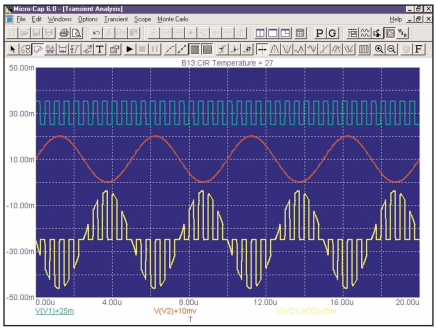
You can plot the current, voltage, flux, inductance, and B and H fields. The parts library includes models for hundreds of commercial devices-generated by our MODEL program.

Individual device temperatures–for precise simulation of on-chip devices.
All resistor, capacitor, inductor, and semiconductor models have individual device operating temperatures to let you model thermal differentials between on–chip devices.

Three BSIM models–for flexible modeling power.
Micro-Cap 6 includes three BSIM models, BSIM 1.0, BSIM 2.0, and the latest version BSIM3V3.2

DISPLAY AND PLOTTING FEATURES

Real time waveform plotting–for observing waveforms during the run.
Micro-Cap 6 produces waveform plots and graphs during the run. As a result,

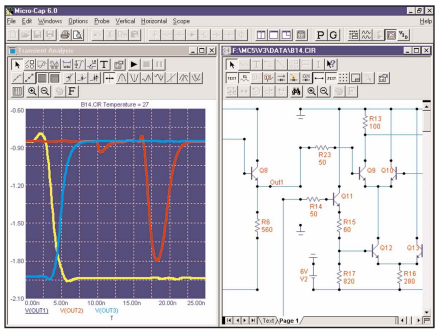


you stop the run to correct problems and avoid wasted time...and it's also much more interesting

You can also change simulation

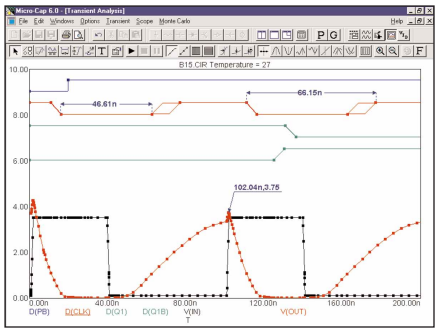
parameters–like temperature, simulation time, or initial conditions-and start a new run without re-compiling the circuit.

Direct schematic waveform probing–for fast measurements and quick display of analysis results.
Micro-Cap 6 lets you probe schematics directly for waveforms. Simply point the mouse at a device or circuit node



and click. You can probe for digital states, voltage, current, power, charge, capacitance, flux, inductance, and B/H fields. Probe can display transient, AC, or DC analysis results. It's like probing a circuit with a scope, spectrum analyzer, or a curve tracer.

Scope–for easy review and analysis of waveforms and plots.
This feature lets you zoom, pan, size, scale, tag data points, inspect values, and analyze waveforms and curves. Scope lets you magnify a waveform, read out its value, check its slope, find

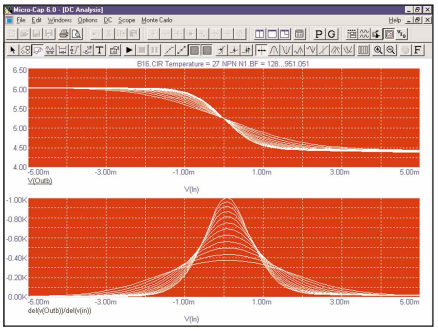


a peak, compare it to another waveform, or measure its rise time, fall time, width, period, peak-to-peak value, and many performance criteria.

You can also annotate the plot with text, add graphics, and numeric tags that show the X and Y values at individual data points or between two data points.

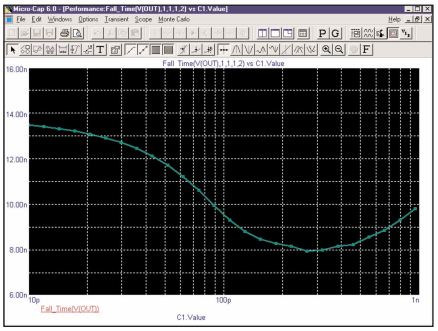
PARAMETER ANALYSIS FEATURES

Parameter stepping–to see how parameters affect circuit performance.
Step symbolic, value, or model parameters to see how circuit performance is affected. For example, you can step



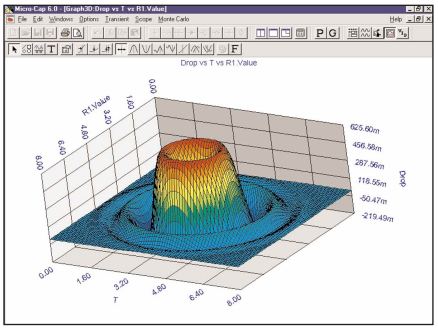
the beta of one or more transistors to see the effect on circuit behavior. This lets you try different designs, explore design limits, and optimize performance.

Performance Plots–to see parameter performance sensitivity.
You can directly measure and plot performance characteristics such as rise



time, fall time, pulse width, period, peak, frequency, bandwidth, and many more.

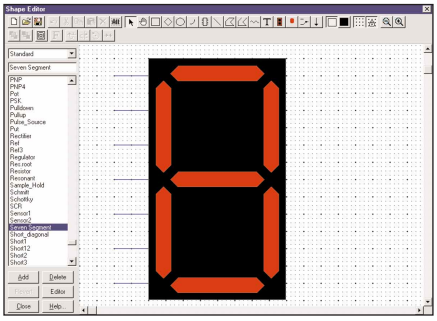
3D Plots–for easy design visualization.
Plot an expression or performance function vs. any two stepped parameters to



quickly see the effects of temperature or parameter variation.

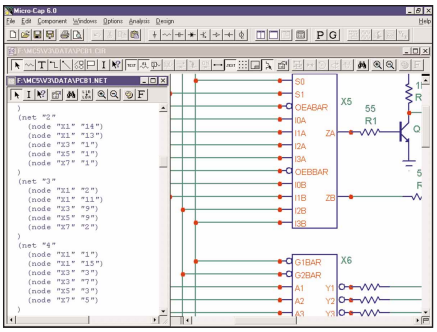
ADDITIONAL FEATURES

Integrated shape editor–for customizing your own shapes.
This full-function shape editor provides a full set of graphics primitives, including line, arc, rectangle, diamond,



ellipse, both open and closed polygons, and a generic logic block with programmable pin icons. It even allows shapes to include other shapes for easy creation of a structured shape library.

PCB interface–lets you create netlist files from Micro-Cap 6 schematics.
This feature lets you translate your schematic to a netlist file that can be used by your PCB package to generate printed circuit board films. Interfaces to



Protel®, Accel®, and OrCad® are available now, with more on the way.

Large device library.
With over 12,000 parts in the parts library, you'll be able to quickly find most digital logic parts, and analog parts like diodes, MOSFETs, BJTs, OPAMPs, JFETs, magnetic cores, crystals, and SCRs.

spectrum

Waveform import capability.
This feature lets you import waveforms from SPICE or Micro-Cap 6 output files for direct comparison. In addition, you can use the difa() and difd() functions to automatically compare two analog or digital waveforms. These functions report any differences between the two waveforms, simplifying testing.

Extensive mathematical operators and variables.
Operators include arithmetic, trigonometric, hyperbolic, Boolean, relational, integration, differentiation, and signal processing types. You can even do Bessel functions and infinite series expressions. Variables include voltage, current, power, charge, capacitance, flux, inductance, B field, and H field. Device variables include lead currents and lead-to-lead voltages, such as the base current and the base-emitter voltage of an NPN.

Extensive list of device models.
Micro-Cap 6 supports a long list of native device models, in addition to macros built from these basic device models. These include:

Analog primitives.
- Battery voltage source
- SPICE voltage source
- SPICE current source
- MC6 Pulse voltage source
- MC6 Sin voltage source
- MC6 User file source
- Resistor
- Capacitor
- Inductor
- Diode
- SPICE dependent sources
- Linear dependent two port source
- Transmission line (lossy or ideal)
- Transformer
- K device (magnetic coupling)
- Bipolar Junction Transistor
- MOSFET (3 models 1, 2, and 3)
- BSIM (3 models 1, 2, and 3.3.2.)
- OPAMP
- GaAsFET (3 models)
- JFET
- Analog Behavioral Devices
  - Laplace Function Source
  - Laplace Table Source
  - Function Source
  - Table Source
  - Z Transform Source
  - Sample and Hold Source
- Switches (3 models)
- Macros
  - Absolute value
  - Amplifier
  - Center-tapped transformer
  - Clip function
  - Crystal
  - Delay
  - Differentiator
  - F(S) Laplace block
  - FSK Frequency Shift Keyer
  - Gyrator
  - Integrator
  - Multiplier
  - Noise
  - Potentiometer
  - Programmable unijunction transistor
  - PSK Phase Shift Keyer
  - Resonant tank circuit
  - Schmitt trigger
  - Silicon-controlled rectifier
  - Slip circuit
  - Subtractor
  - Summer (two input)
  - Summer (three input)
  - Triac
  - Triode
  - Voltage-controlled oscillator
  - Wideband transformer

Digital primitives.
- Standard and tri-state gates
  - Buffer
  - Inverter
  - And
  - Or
  - Nand
  - Nor
  - Xor
  - NXor
- Edge-triggered flip-flops
  - JK type
  - D type
- Gated flip-flops and latches
  - SR
  - D-Latch
- Digital loads
  - Pullup
  - Pulldown
- Delay line
- Programmable logic array
- Analog to digital converter
- Digital to analog converter
- Analog to digital interface
- Digital to analog interface
- Digital behavioral modeling
  - Logic expression
  - Pin delay
  - Constraint checker
- Stimulus generators

Extensive help system.

Over 25,000 lines of on-line help are context sensitive, indexed, and topically arranged for easy learning.

Over 500 error messages help you pinpoint circuit problems. Most error messages come with a "More" button for additional description of the nature of the problem.

Over 125 sample circuits give you plenty of examples to learn design and simulation techniques.

Over 100 Help Bar notes describe program features as you move the mouse over them.

Over 15 live demos illustrate the workings of the program.

Spectrum Software
408-738-4387
Fax: 408-738-4702
www.spectrum-soft.com
sales@spectrum-soft.com