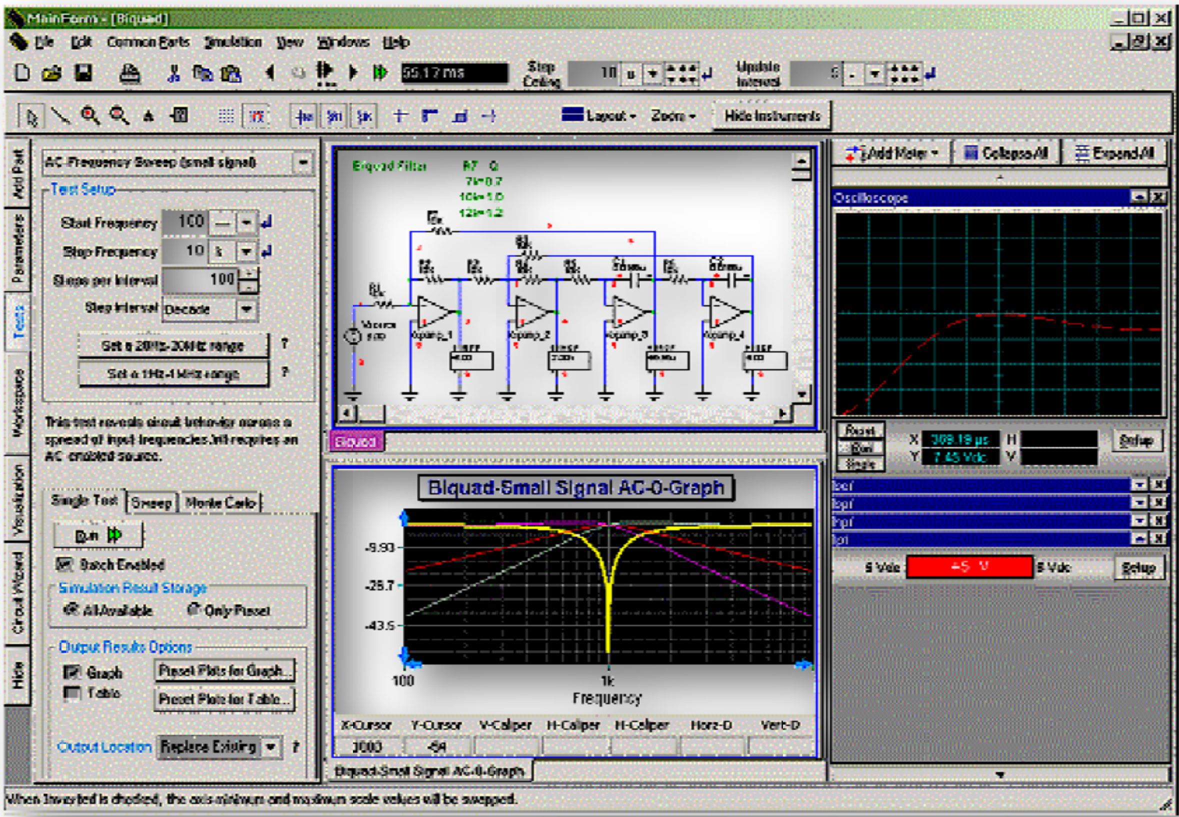
B2 Spice

Electronic Circuit Simulation Software



Standard Features Of B² Spice Professional

A range of powerful virtual instruments.
Circuit Visualization to display the actual current flow through the circuit and the relative voltage relationships by varying the wire's display color.
Quick, easy, and intuitive schematic entry.
Export to and import from Eagle, a world leader in PCB software.
A continuously run Simulation mode.
PCB export to make your designs a reality.
25,000 digital and analog parts including hundreds of REALISTIC behavoral models for such parts as resistors.
Multiple bit ports and buses.
Parameterized subcircuits.
Create a part from any circuit.
Password protected defects.
Shared models.
Database editor to import and manage the library of parts.
Integrated symbol editor.
PCB export and bill of materials.
Improved schematics with DIN and ANSI symbols.
RF simulations and network analysis.
Schematic borders and title-box for professional output.
Smith and polar plots.
Intuitive, full featured schematic editor.

Cut, copy and paste of selected items.
Undo support.
Full device rotation.
Device mirroring.
Parts browser.
Repeat placement of a device.
Easy to draw and edit wires.
Browse-able, filterable, device libraries.
User-defined devices and symbols.
Complete macro device capability.
Rubber banding of wires and devices.
Annotation of devices.
Quick menu selection of commonly used parts.
Zoom in or out on an area or item with custom zoom factors.
Fit circuit to window function.
User-selectable colours.
Export circuit drawings and waveforms.
Supports all Microsoft Windows driven printers and plotters.
Built-in Symbol Editor to create custom device symbols.
Modification of devices and symbols in schematic.
control over fonts and colours in schematic.
Export SPICE3 compatible net lists.
Generate SPICE3 sub circuits from the circuit.
Create part from the circuit.
View steady state results directly in the schematic.
View node numbers.
Colour coded digital wire states.
Copy circuit picture to the clipboard for pasting into other applications.
Print circuit to any output device.

Parts Library Of Over 30,000 Parts

All parts in Berkeley Spice 3F5 and XSpice are included in the parts library. Many digital parts from the most popular libraries. Parts from vendors such as AMP, APEX, Burr-Brown, Comlinear, Elantec, Linear Technology, Maxim, Motorola, National Semiconductor, Texas Instruments, and more.

Many types of parts specified as subcircuits including Zener diodes, power MOSFET's, operational amplifiers

Many types of digital parts, including gates, inverters, counters, registers, buffers, coder/decoders, mux/demuxes, and more.

Additional parts specified at a behavioural level including:

Analogue phase locked loop, continuous S domain transfer function, peak detector, sample and hold circuit, Schmitt trigger/ bistable network, voltage noise source, current noise source, piecewise linear system, discrete time Z domain transfer function, and operational amplifier.

These behaviour model parts can be customized from their behavioural parameters Resistor, Capacitor, Inductor, Coupled inductors, diode, switches, Bjt, jfet, mosfet (all 6 models), mesfet, lossless transmission line, lossy transmission line, uniform RC line, Controlled sources, voltage and current sources. Arbitrary source which can be used to model non-linear resistors and other parts. Parameterised sub circuits give you the ability to alter sub circuit part behaviour.

Library Management

Import your existing SPICE subcircuits and models into parts library.
Easily create and modify parts in database.
Modify symbols for the parts.
Match up symbol pins with model pins to reliably create parts.

Modify parameters for behavioural models.
Modify parameters for all Spice models.
Specify manufacturer, description, and category for parts and models.
Parts, models, and symbols are stored in a Microsoft Access database file.

Highly Accurate Simulations

Run simulation directly from schematic, or from a text window containing a spice deck 32-bit code:
DC Bias, DC Sweep, AC Sweep, Transient, Monte Carlo, Noise, Distortion, Fourier, Pole/zero, sensitivity, and transfer function simulations Show Device and Model parameters in a text window after the simulator has calculated them.
Full set of SPICE and XSpice options, including error tolerances, iteration limits, MOSFET defaults and more.
Pause or stop simulation at any time.
Digital command and vector files give you complete control over digital simulations.
view simulation results as the data is being collected.
Intuitive interface for setting bias point initial guess and transient initial conditions.
Multifunction signal generators (sine, single frequency FM, exponential, pulse, sawtooth, triangle, piecewise linear).
Multimeters (measure DC or AC voltage, current).
Unlimited number of instruments.
Full screen analogue waveform analysis.
Full screen digital trace.
View any number of waveform simultaneously.
Plot voltage, current, frequency response and dc characteristics.
View results in tables and/or graphs.
Log of simulation progress, errors, warnings, and statistics provided with each simulation run.

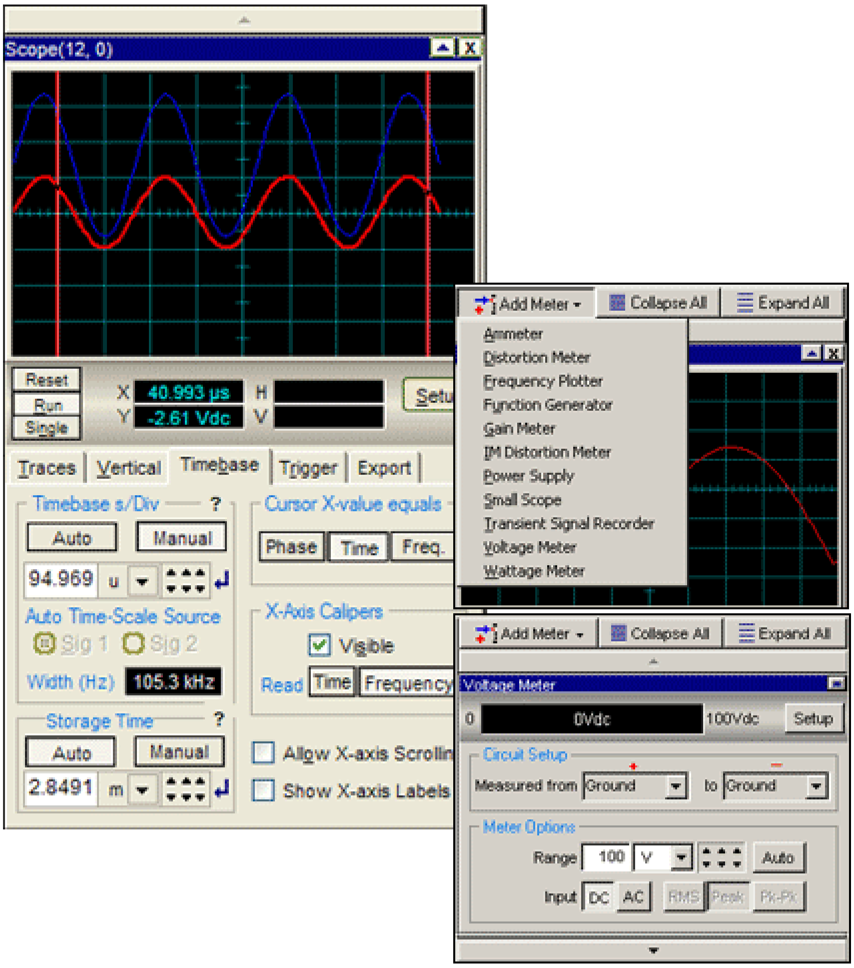
Graph Editor

Zoom in or out on a specific point with a magnifying glass cursor.
Stretch the plot horizontally or vertically.
Squeeze the plot horizontally or vertically.
Scroll around the graph.
Create your plots by simply entering mathematical expressions.
Value labelling of the X and Y axes for all graphics windows.
Find and label maximums and minimums.
Group plots by families or control them individually.
Measurement of differences between specific points on a graph.
Measurement cursors provide quick and accurate measurements in all analysis windows.
Non-linear math functions.
Ability to include "deep" plots of sub circuit nodes.
Highlight any plot by clicking on its name in the legend.
Zoom in on any plot in the graph by control-clicking on its name.
Unlimited number of plots.
Set plot colours.
Show or hide any plot.
Set graph font.
Set log or linear scales for x and y axes.
Set graph title.
Specify engineering or scientific notation.
Copy graph picture to the clipboard for pasting into other applications.
Print graph to any output device.

Virtual Instruments

One of the exciting new features of B2 Spice v.5 is the addition of Virtual Instruments.

Select from one of 11 instruments to power, test and analyze your circuit. Most instruments can only be used in Simulation mode (continuous Transient simulation) while a few are used to power and stimulate the circuit. Some instruments perform their owns analysis of the circuit.



All instruments are set up and viewed in the Instrument Panel that can be hidden when not in Simulation mode.

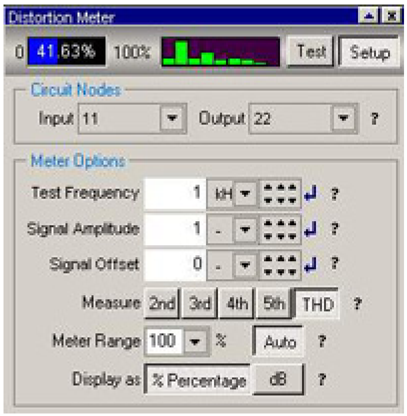
As a Digital Storage Oscilloscope, it has two input traces and an adjustable timebase, trigger source, and vertical signal scaling.

Each channel can be selected and edited by clicking on the Input Trace 1 or Input Trace 2 button in the Traces tab. Selecting a trace allows the signal node, the plot color and width and label to be set. (Plot widths over one pixel slow down the drawing of the waveform.) Additionally, the signal's gain and vertical and horizontal offsets can be set. Also note that each trace can also be set as DC or AC input mode. This can play an important part in getting a signalto display correctly in the scope. Please see the note at the bottom on the difference between AC and DC mode.

The trigger source can be set to either channel 1 or 2 or another node in the circuit (Ext). The trigger mode can be set to AC or DC, and the triggering level can be adjusted in the Trigger Above and Below boxes. The Trigger Edge selects they how the signal is triggered. The display can be frozen by pressing the “Run” or "Single", allowing complex waveforms to seen clearly and measured. The Run buttons stops the display at the point the button is pressed. The "Single" button stops the display after one complete drawing of the scope screen. Moving the vertical and horizontal sets of calipers displays the either the vertical voltage or the percentage of the oscilloscope's left axis and the amount of time or the equivalent frequency implied along the bottom axis. The calipers can be activated in the Vertical and Timebase tabs, for vertical and horizontal calipers respectively.

Distortion Meter

The distortion meter measures the distortion generated within a circuit, usually an amplifier.

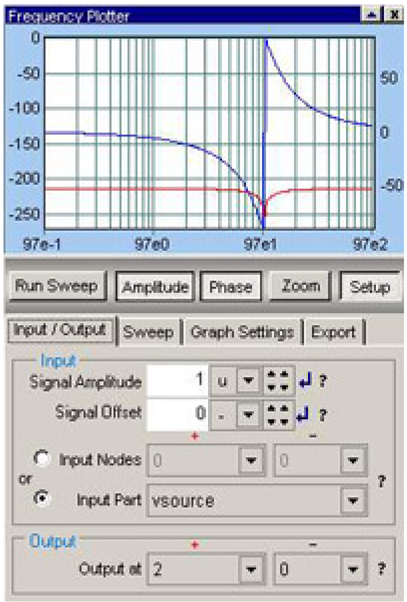


First specify an input and an output node. An input and output node must be chosen, as the distortion meter needs an input to inject a pure signal, which will compare the output's signal against in calculating the distortion in the output signal. Second a test frequency needs to be entered (usually 1kHz for audio circuits). You can also set the Amplitude and offset of the test signal.

The option to display the results as a percentage or in dBs is quite straight forward. One percent distortion equals the distortion's contribution to the output signal being –40dB down in amplitude relative to the output signal.

The bar's length expands and contracts with the distortion being measured, which makes it an analogue readout. When the measured current exceeds the distortion meter's range, the bar changes colour from its normal blue to red. Pressing the "Auto" button overrides the fixed range and auto adjusts the range to twice the highest distortion reading it sees.

Frequency Plotter



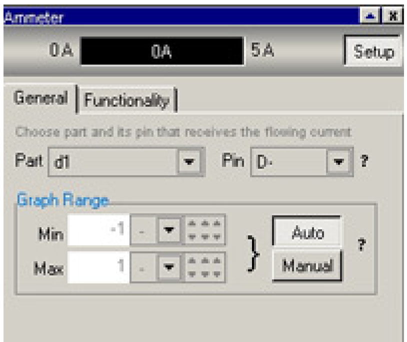
The frequency plotter measures a circuit's frequency response and displays its measurements in a small graph.

The Input/Output tab allows you to set up the test in regards to the circuit. Under this tab, choose two nodes in a B2 A/D Spice circuit or an input part. (Behind the scenes, the Sweep instrument adds a voltage source to the circuit or if there is already a voltage source connected to the node and ground, it takes over its settings.) You should also choose two output nodes. Then select a Signal Amplitude and you are ready to run the test.

Set up the graph under the Graph Settings tab. Here you can control the graph's display of both the amplitude and the phase across the frequency sweep's range. The Y-axis settings are the maximum and minimum limits of the graphs display, with the option to set the graph display to linear or in dB. Pressing the "Auto" button overrides the fixed Y-axis tick makings and it auto adjusts the graph's Y-axis limits to twice the highest current reading it sees. The X-axis settings are the beginning and ending frequencies.

The last tab, Export, allows you to export the graph to various formats, including graphic files, text tables, or even copy it to the Project's graph window. You can even copy the graph into memory to paste into other applications. Pressing the "Zoom" button brings up a new window with just the graph portion of the small frequency sweep instrument.

Ammeter



First choose a part in a B2 Spice circuit and then choose a pin on the part. This can be set under the General tab. The next option is the ammeter's display range. The meter displays its measurements in two ways: text and a moving bar graph.

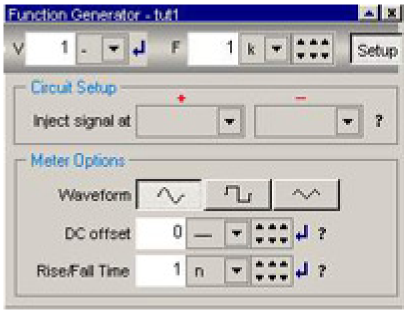
When the measured current exceeds the ammeter's range, the bar changes color from its normal blue to red. Pressing the "Auto" button overrides the fixed range and auto adjusts the range to twice the highest current reading it sees.

Under the Functionality tab, choose whether the current signal is a DC or AC one. If the signal is AC, then choose whether the RMS (Root Mean Square) value, Average, Absolute Average, Peak-to-Peak, or the peak value should be used.

Behind the scenes, the ammeter instrument severs the wire connection to the selected pin and then it bridges that gap with a SPICE ammeter in the netlist sent to the SPICE engine. A virtual ammeter is automatically inserted into the virtual instruments display if an ammeter device is placed into the schematic.

Pressing the “Setup” button toggles the length of the ammeter, so that when its setup is complete, the ammeter becomes only as tall as its display band.

Function Generator

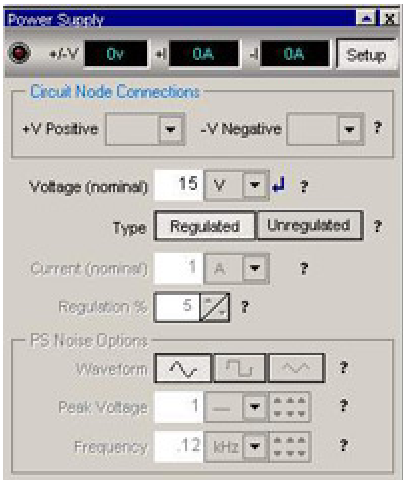


The function generator creates sine, triangle, and square waves to be injected into a circuit.

This instrument is functionally equivalent to the SPICE voltage source, but it adds easy frequency and amplitude changes. A node within the circuit must be specified to receive the function generator’s output signal. (the reference node is assumed to be ground.) The DC offset of the function generator’s output can be adjusted up and down.

Pressing the “Setup” button toggles the height of the Function Generator, so that when its setup is complete, the it becomes only as tall as its display band.

Power Supply



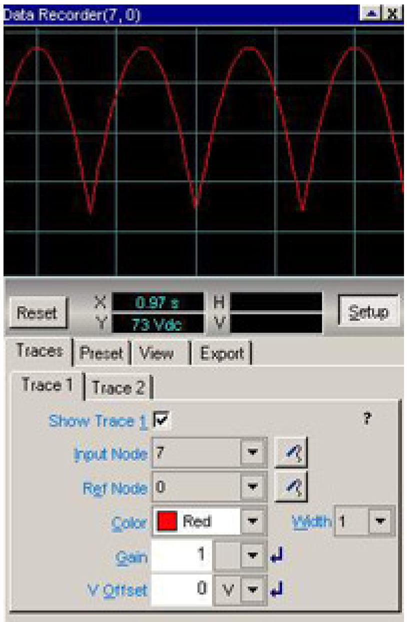
Much like an actual bench power supply, the power supply instrument provides a selectable output voltage(s). Both regulated and unregulated power supply types are available and the actual current delivered to the circuit is displayed. Unlike an actual bench regulated power supply, the power supply instrument allows a degraded mode, wherein it functions like an unregulated power supply with ripple on its output, much like an actual raw DC power supply.

The output voltage is set in voltage-selection edit box. If voltage-regulated performance is desired, then select the Regulated button for type. On the other hand, if you wish to simulate a cheap power supply, such as a wallwart, then select Unregulated for type and set the regulation to 25%, the noise to 500mV and the current to the idle current of your circuit. The regulation percentage refers to the over voltage an unregulated power supply develops when unloaded.
The power supply noise’s frequency equals twice the wall voltage’s frequency, for example, 120 Hz, when the wall voltage is at 60 Hz. The noise’s waveform is selectable. Actual capacitor smoothed power supplies produce a triangular noise waveform; choke input power supplies, something closer to a sine wave waveform; and some switching power supplies, a square wave waveform.

Transient Signal Recorder

The transient signal recorder is designed to record up to two signals for either a predetermined period or the entire length of the simulation run and save the data to a file.

Setting up the Transient Recorder is as simple as specifying the Trace node(s) and reference node(s) and then setting up the capture window. The Traces tab is where you set up the signal(s) to record.

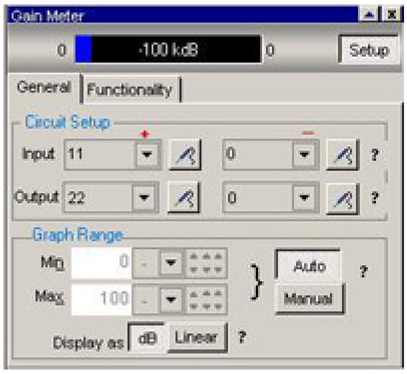


The Data Capture's All setting captures and saves the data for the entire length of the simulation run until you stop the simulation. The Most Recent button activates the Duration box and allows you to specify a certain period of data to save.

If you specify the most recent 5 seconds to capture, and the simulation runs for 23 seconds, only the data from seconds 18-23 are captured. Everything else is discarded. The Fixed button activates both the Start and Duration boxes and allows you to specify that a certain time interval's data is kept. If you specify a start of 1 second and a duration of 5 seconds, then only the data from seconds 1 to 6 are kept. Note that the Duration is not the same as a stop time. It specifies the LENGTH of time, not a fixed time.

Again, the Interactive Viewing settings are independent of the Data Capture settings. The All, Most Recent, and Fixed boxes function like the Data Captures settings, but this only affects the graph display. Note that even data that is not displayed in the graph can be retained for saving to a file. If a Most Recent viewing interval of 50ms is specified and the Data Capture setting is set to All, then only the most recent 50ms of data is shown, but ALL the data is being stored in memory and can be saved to a file.

Gain Meter



Purpose: The measurement and display of relative strength of two voltages (AC or DC) across or inside a circuit in dBs (decibels).

Signals often differ by great magnitudes. The miniscule output voltage from a moving-coil phono cartridge (.1mV) becomes amplified to 1000 volts peak-to-peak at the tube-amplifier's output tube's plate.

The ratio between these two voltage is 10,000,000.

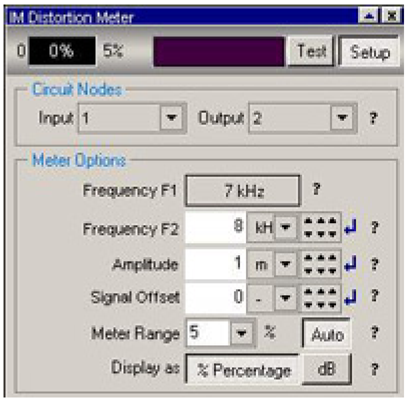
But expressed in dB (decibels), this ratio becomes only 140 dB, a figure that is much more manageable.A linear display of relative signal strength can be used by pressing the "Linear" button. The display will then show the second signal divided by the first signal.

To measure the ratio between to signals in or through a circuit, the Gain Meter must be attached to two nodes and their references (usually, ground). The gain meter displays its measurements in two ways: text and a moving bar graph. The bar's length expands and contracts with the ratios/gain being measured, which makes it an analog readout. When the measured signal gain exceeds the gain meter's range, the bar changes color from its normal blue to red. Pressing the "Auto" button overrides the fixed range and auto adjusts the range to twice the highest distortion reading it sees.

If the signal is AC, then choose whether the RMS (Root Mean Square) value, Average, Absolute Average, Peak-to-Peak, or the peak value should be used. Then select a sample period for the ammeter.

Inter-Modulation Distortion Meter

This instrument measures the distortion of a circuit when two sinusoids of different frequencies (Fa and Fb) are applied at the input.



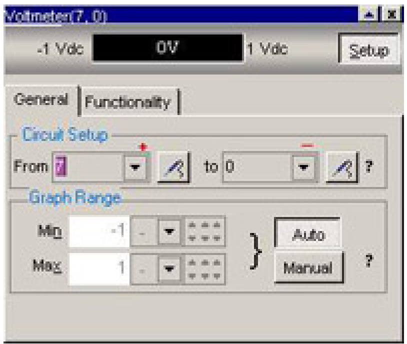
The Intermodulation distortion meter has three set-up modes, SMPTE (Society of Motion Picture and Television Engineers), CCIF, and manual setup. The SMPTE test uses a signal made up of two separate sin waves.

The first sin component has a high amplitude and low frequency (60Hz, Fl), and the second has a high frequency (7kHz, Fh) and low amplitude (¼ strength of the 60Hz component's amplitude). Sidebands appear in the frequency response at 60Hz intervals around the 7KHz tone. The percent intermodulation distortion is defined as the percentage of amplitude modulation represented by the 2nd and 3rd sidebands, i.e. Fh +/- Fl, and Fh +/- 2Fl).

The CCIF test uses a source made up of two high frequency, equal amplitude components whose frequencies are separated by a small frequency (e.g. 1Kh). The program defaults to source signal components at frequencies of 13kHz and 14kHz. With this test, the percent intermodulation distortion is defined as the amplitude of the frequency component at Fh-Fl as a percentage of the source amplitude.

The single frequency distortion test and the intermodulation distortion tests are often performed on amplifier circuits because with this class of circuit you want to minimize the distortion of the signal during the amplification process.

Voltage Meter



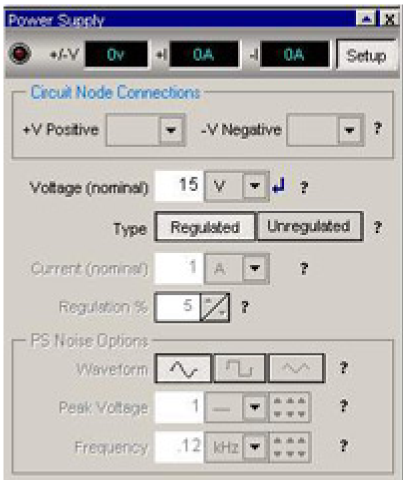
The voltmeter measures the voltages present in a circuit at specific points.

Choose a node to for the positive and negative nodes using the drop down box or by using the "Probe" tool next to the drop down boxes. Using the probe tool and clicking on a node that you would like to measure automatically selects that node in the drop down box.

You should also be aware that voltmeter instruments are automatically inserted if you place a voltmeter part in the schematic.

The meter displays its voltage measurements in two ways: text and a moving bar graph.

Pressing the “Auto” button overrides the fixed range and auto adjusts the range to twice the highest voltage reading it sees.



Pressing the “Setup” button toggles the height of the voltmeter, so that when its setup is complete, the voltmeter becomes only as tall as its display band. Note that a voltmeter virtual instrument is automatically inserted if a voltmeter part is placed in the circuit.

Watt Meter

The watt meter measures and displays the power dissipation by a circuit element.

Connecting the power meter to a circuit can be done in two ways in the Circuit Setup tab: by selecting an individual part or by selecting two nodes within the circuit. The Meter Options tab control the display of the meter.

The meter displays its measurements in two ways: text and a moving bar graph.

The bar's length expands and contracts with the power being measured, which makes it an analog readout. When the measured current exceeds the power meter's range, the bar changes color from its normal blue to red. Pressing the "Auto" button overrides the fixed range and auto adjusts the range to twice the highest power reading it sees.

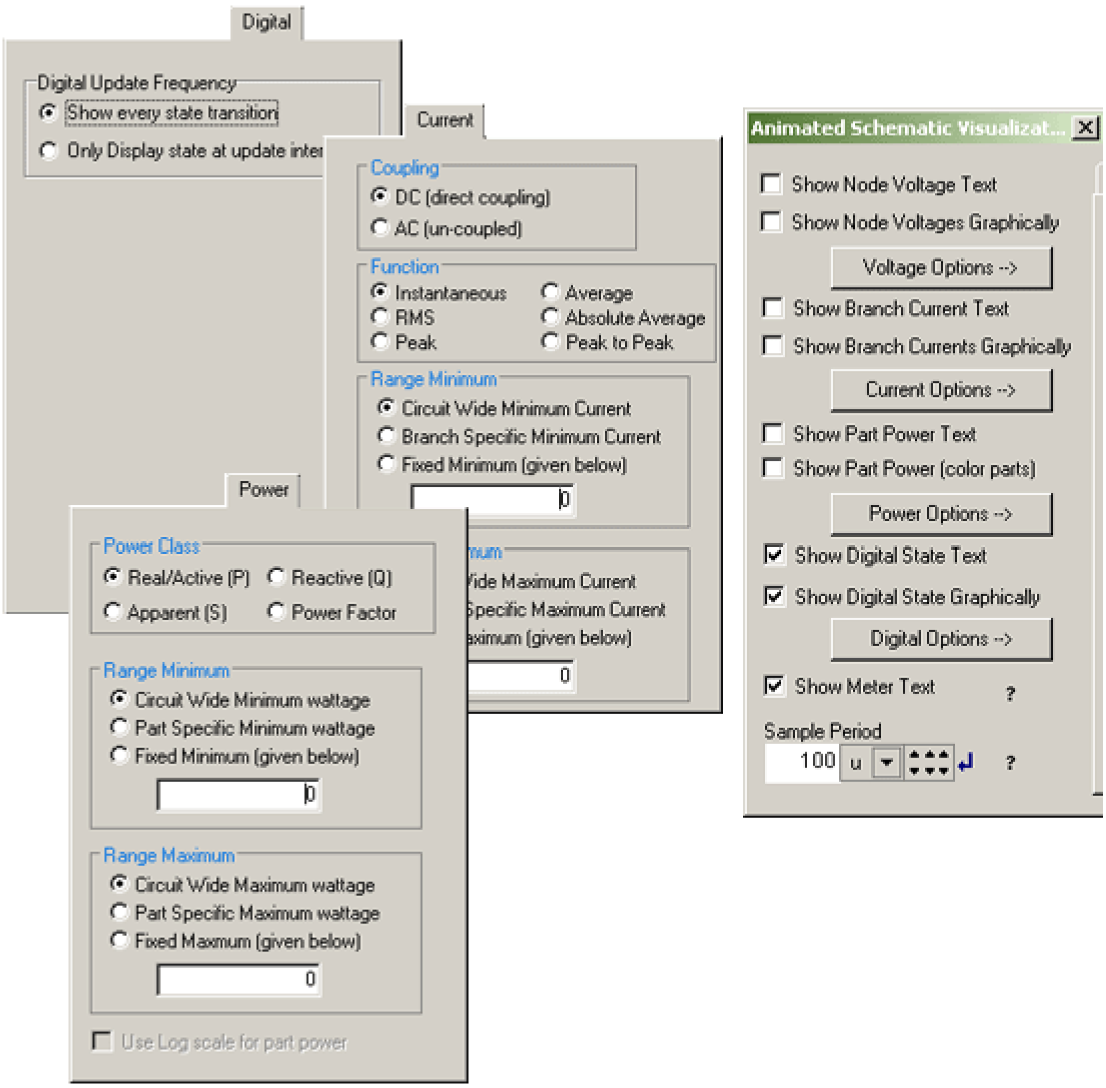
Circuit Visualizations

This revolutionary feature shows the connecting wires changing shape and color to reflect current and voltages in the circuit, with respect to DC stepping, frequency sweep and time.

This is not easy to capture in a screen shot but the options shown indicate how the simulation can be setup.

Relative voltages are color-coded to magnitude and arrows display the actual current paths as they flow within the circuit.

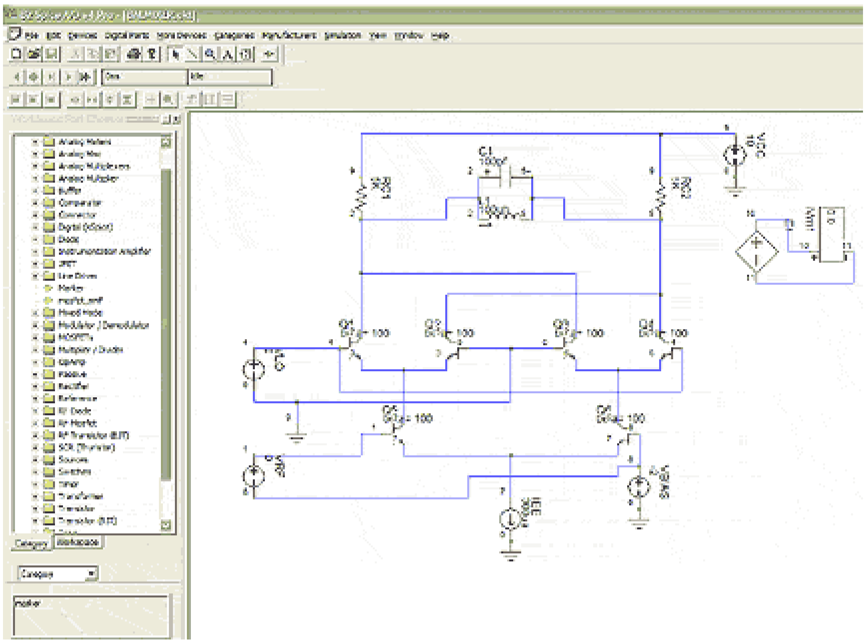
In other words, now you can see into the circuit.



Schematics Are Now Vastly Configurable

B2 Spice had an already intuitive interface, but we have added new features and streamlined others to help make B2 Spice V5 schematics the easiest and most powerful yet.

You can now easily organize and control any open file or window in B2 Spice. A new graph or circuit can be inserted with just a few simple clicks. Open and close windows, access and control simulations, and browse simulations results all from within the Workspace.



Browse the large Parts Database using our new Parts Chooser window. Use the tree/branch style navigation to quickly find a part. Parts are organized by their category, manufacturer, or first letter. Highlight a part to get a preview. Simply double click the part to place it, or simply drag it over to the schematic.

Some of the schematic options now include:

Grid spacing, resolution and visibility.
Wire colours.
Node/pin colours.
Visible titlebox and page borders.
Scalable symbol sizes, ANSI and DIN symbol styles.
Part text display control.
Font control.
90 degree or any angle wiring.
and much more...

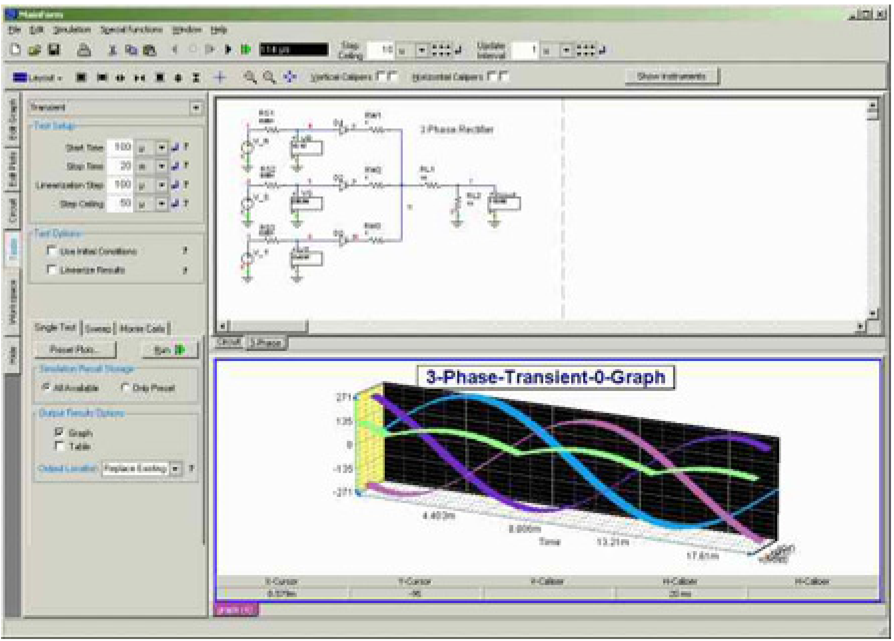
Auto-save is now available along with a user specifiable period for auto-save. And with the new unlimited levels of undo and redo, schematic editing is virtually foolproof.

Parts Database

B2 Spice A/D has a library of over 24,000 digital and analog parts.

Graphs and Timing Diagrams

With B2 Spice 5 greatly improved graphing module, we have given you even more powerful to view your results in any way that you want to. Some of the Version 5's new capabilities include:

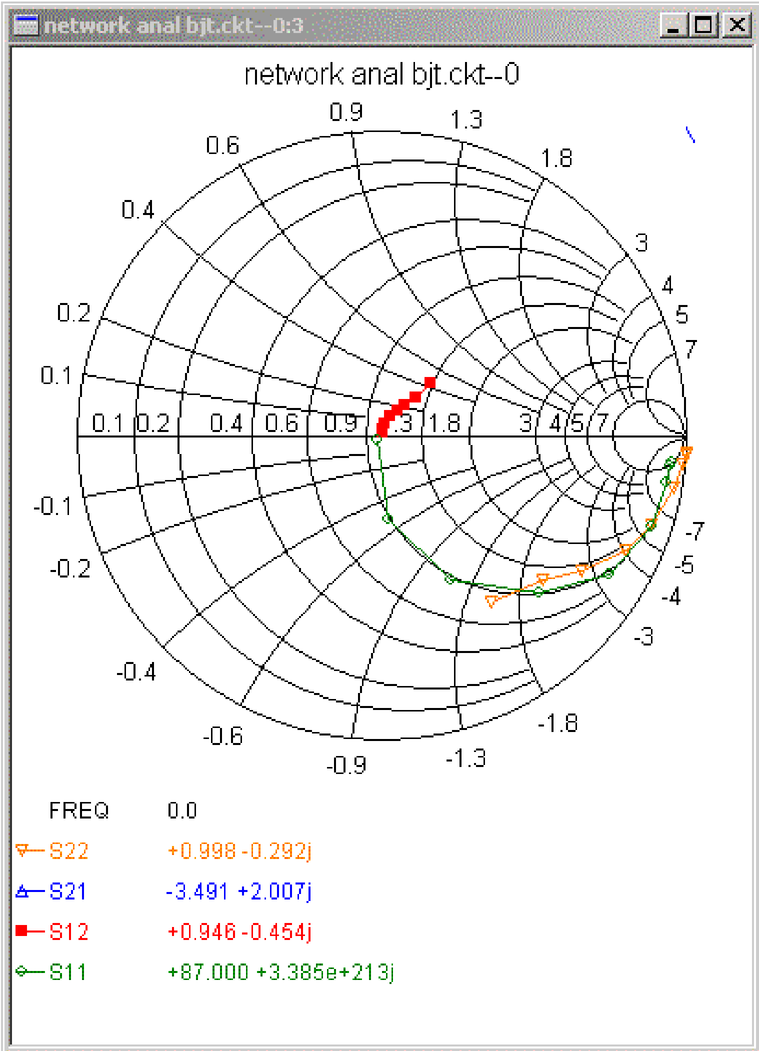


Analog and digital signals in one graph.

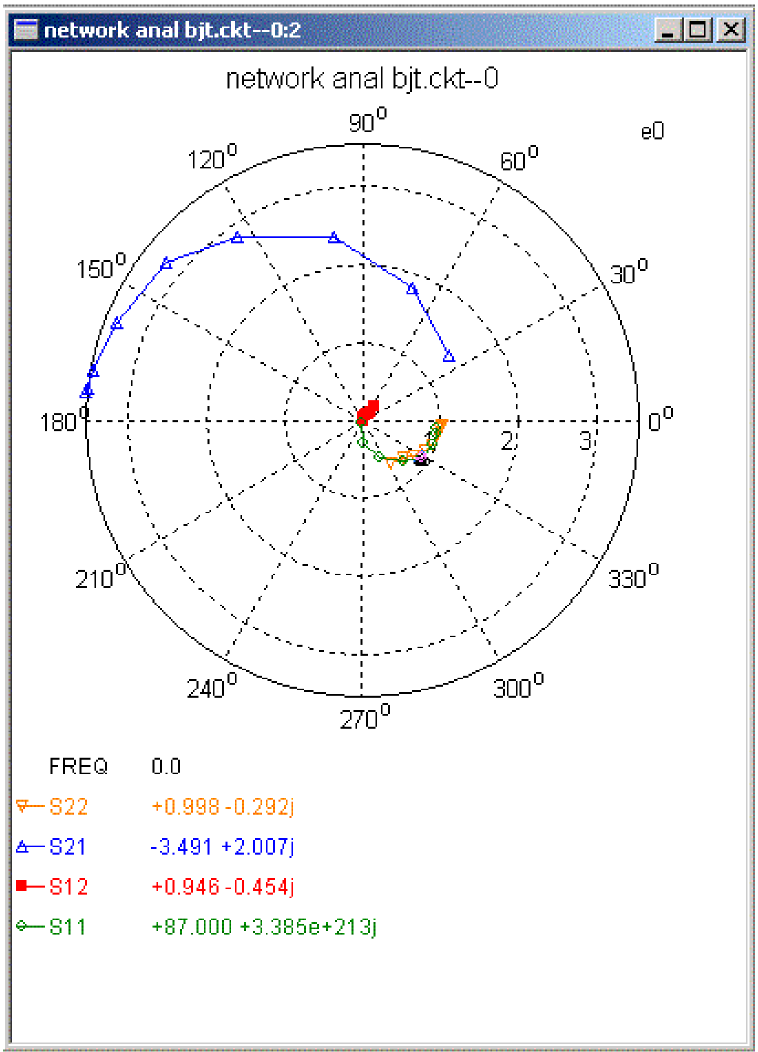
User editable signal list - display only the plots that you want.

Cartesian graph.

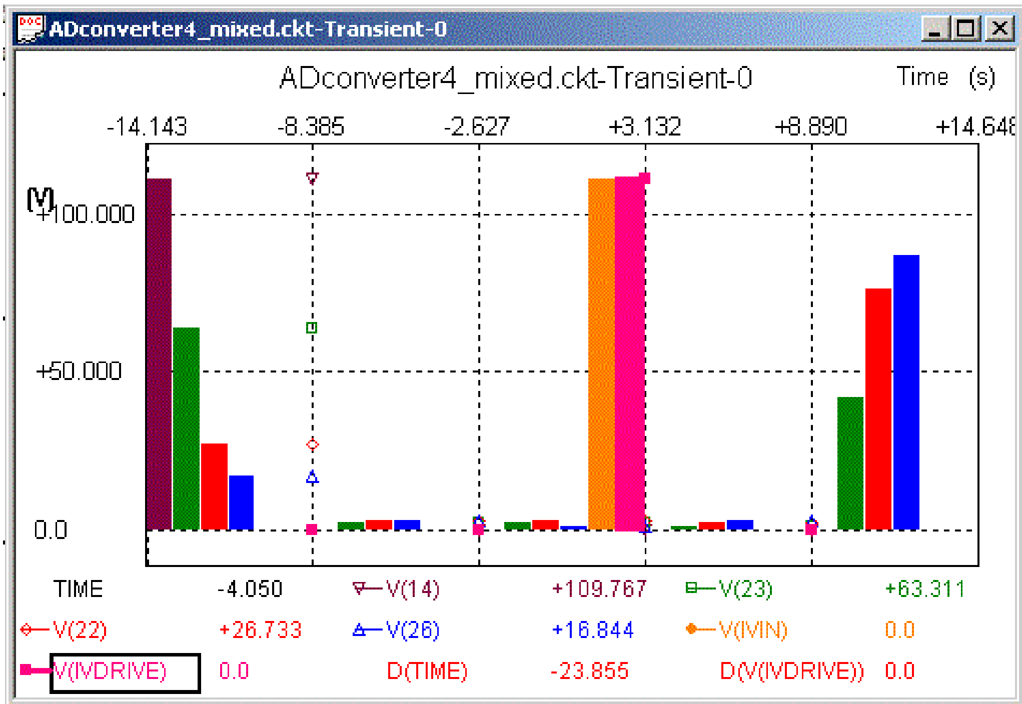
Smith Charts



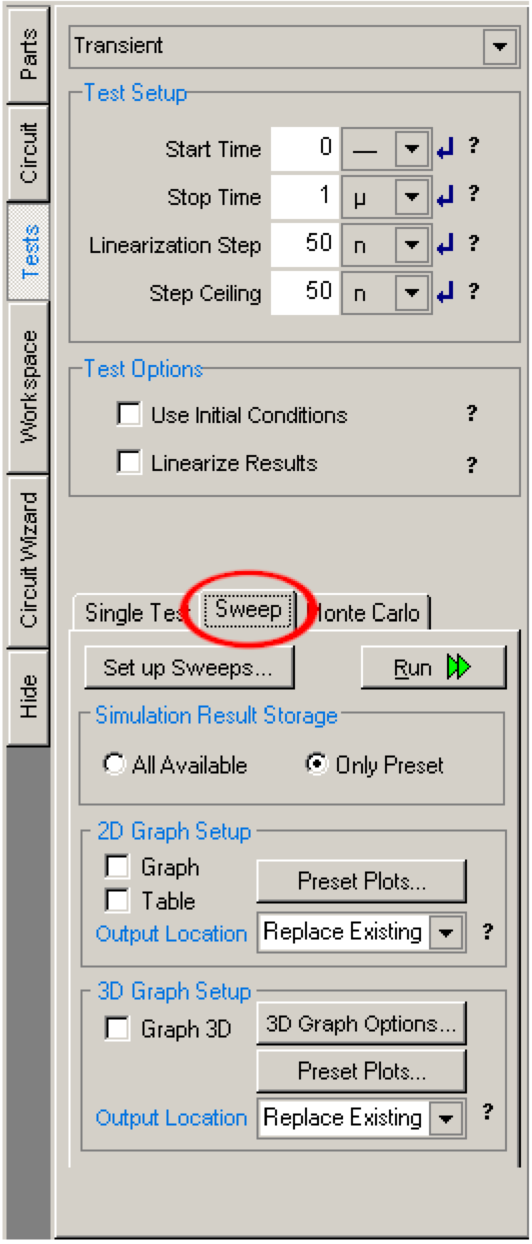
Polar Plots



Histogram



Custom plots - add your own plots using our library of mathematical functions.
3D views.
Combine plots and digital signals from any other graph, circuit, or simulation result.
Optional independent graphs for each signal.
Precise measurements between any two points on a graph - click one to mark the first point and hold and drag the mouse. The X and Y difference are displayed at the bottom of the legend.
Show next maximum and minimum and zero crossing command locates and labels these points on the plot.
Log for both X and Y axes.
Selectable graph fonts and plot colours.
Custom zoom.
Independent plotting scales.
"Deep" subcircuit plotting.
Signal family grouping.
Easy copy and paste export to other programs to quickly put together reports.



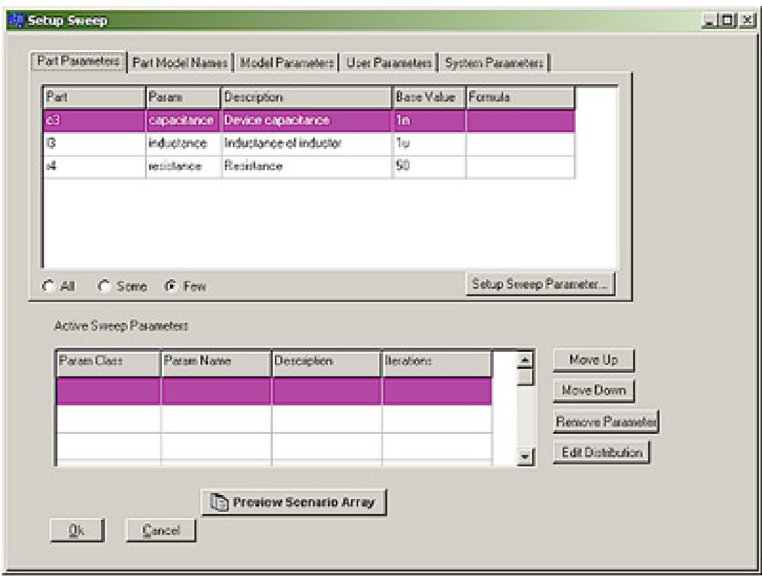
Parameter Sweeping

Every Test comes with an option to "Sweep" or Step a parameter over the entire run of the test. In other words, you can vary a particular parameter for each run of the Test.

For instance, you can Step a particular resistor's resistance from 1K to 10K by increments of 2K for a Transient Test. When you run the Transient test, the first transient run will be run with the resistor's value at 1K. The second transient run will be run with the resistor's value set at 3K, since the increment was set at 2K. Each step of the resistor's value will result in Transient test so the Transient test will be run 5 times, with resistor's values of 1K, 3K, 5K, 7K, and 9K. The combined results will be displayed in a graph.

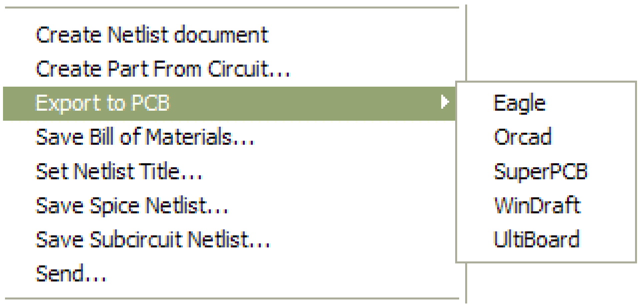
To set up a sweep, select the test you would like to run from the Test tab in the toolbox and set up the test parameter. Then click on the "Sweep" tab in the Test setup.

In the Sweep tab, click Set up Sweeps to enter the dialog to set up your sweep parameters. This will bring up the following dialog box (the data will depend on what you have in your circuit):



PCB Export

One of the exciting new abilities of B2 Spice A/D v5 is the ability to export a schematic to PCB for layout and routing. Now with a few clicks, you can export your schematics to some of the most widely used PCB programs to take your design to reality.



B2 Spice A/D v4's parts database comes with a library of hundreds of PCB packages. Most of the models in our library already have PCB packages assigned to them. However, you can easily change the package assigned to any part.