

B² Spice[®] V5

Our hottest Spice ever



B2 Spice contains a plethora of additional features, from complex digital circuits to radio frequency simulation and PCB design and includes a standard library of over 25,000 parts. There's no limit on the size of your design and results are easy to understand and interpret.

Standard features of B² Spice Professional

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 behavioral 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

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

Parts Library of over 30,000

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 .

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

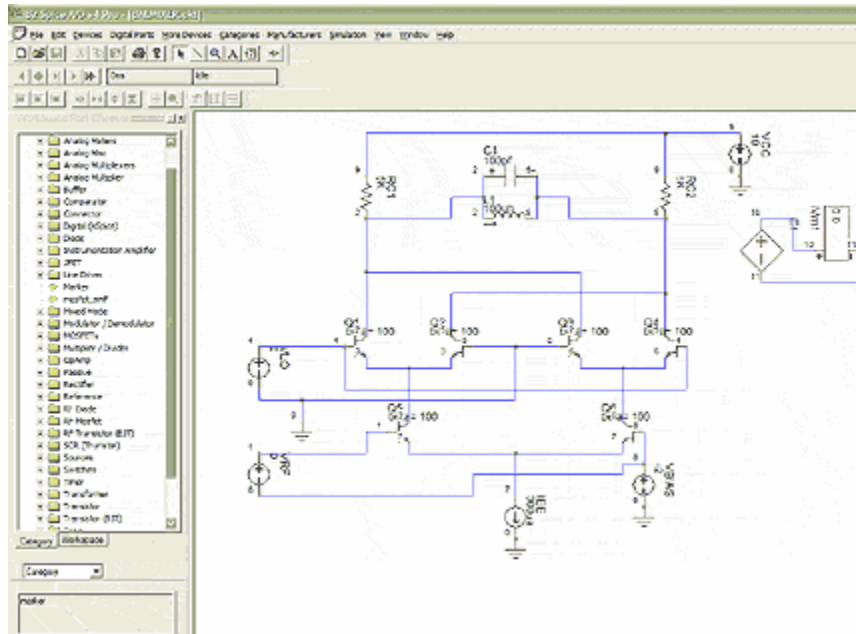
Free Technical Support

Extensive telephone and on-line help
Circuits can be represented either as schematics or spice netlists
Comprehensive User Manual
Site licensing available
Education discounts

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.

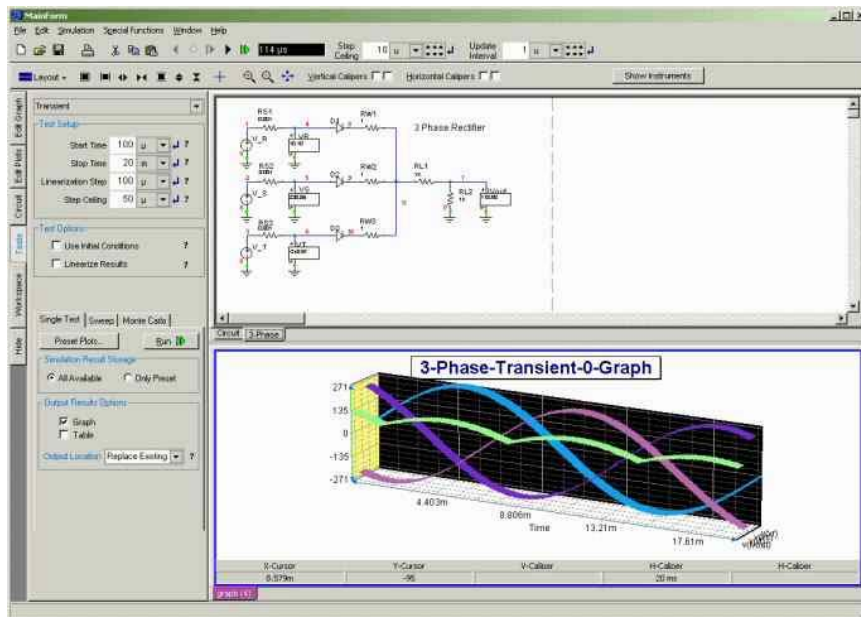
Simulation

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.

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
- stretch vertical/horizontal
- custom zoom
- independent plotting scales.
- "deep" subcircuit plotting
- signal family grouping
- easy copy and paste export to other programs to quickly put together reports



Parts Database

B2 Spice A/D 5 has a library of over 24,000 digital and analog parts (professional version).

To view a complete list of the Full and Standard database:

No extra charge for all libraries of models

<p>A wide selection of analog parts of the following categories: OpAmps Registers Diodes Mosfets (including BSIM3) Transistors Modulators/Demodulators Multiplexers Connectors Switches Meters Power sources Buffers Vacuum Tubes JFETs Comparators RF BJTs, Mosfets and diodes and many more...</p>	<p>From the following manufacturers: Motorola National Semiconductor Texas Instruments Analog Devices Advanced Linear Devices AMP Burr Brown Harris Semiconductor Apex International Rectifier Maxim Zetex Philips Polyfet Siemens</p>
<p>Or select from a wide array of digital categories: Gates - and, nand, or, nor, xor, xnor, inverters... Counters Comparators Adders Flip Flops Encoders / Decoders Multiplexers / Demultiplexers Registers Latches Transceivers RAM ROM digital sources and instruments and more...</p>	<p>Grouped by the following libraries: Standard TTL CMOS LS ALS AS S AC / ACT AHC / AHCT L F</p>

Database Editor

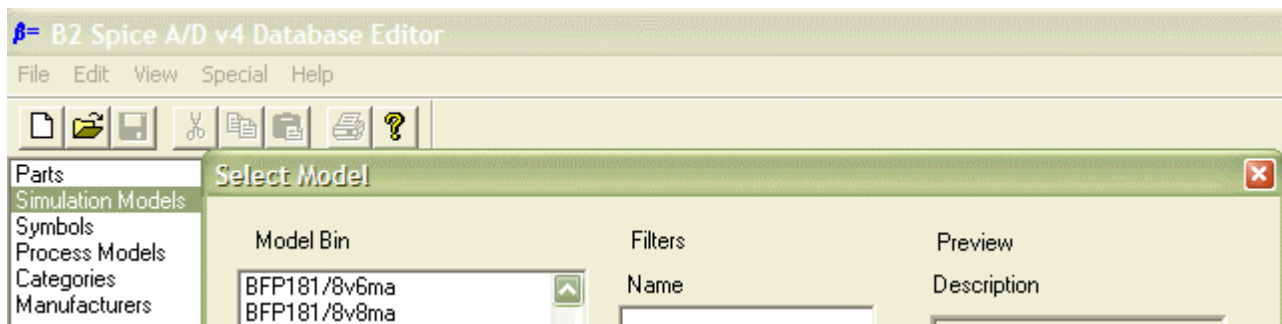
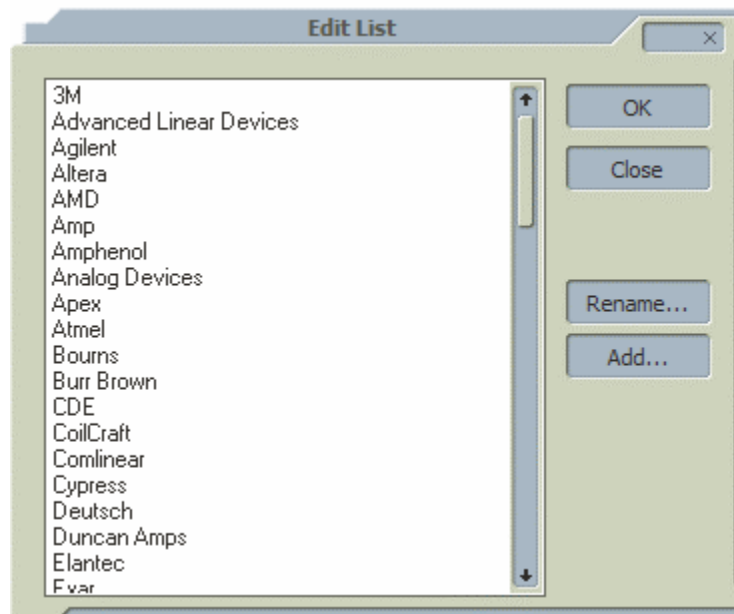
To help you maintain the library of parts, B2 Spice v5 comes with a separate Database Editor. Select from thousands of parts and hundreds of manufacturers. Create your own libraries and custom parts as well as commonly used components. Search and edit by model type, function and manufacturer.

The Database Editor, pictured below, allows you to add new parts or edit any existing device.

Access and edit the guts of any device, from primitive device model properties to symbols and names.

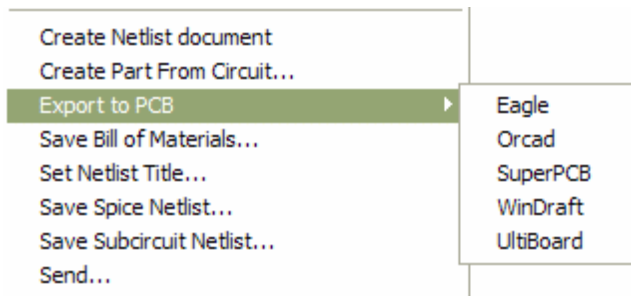
Assign PCB packages to any part or edit any assignment. Add your own PCB package if you can't find what you're looking for.

With the Database Editor, adding new parts is a simple process consisting of a few clicks.

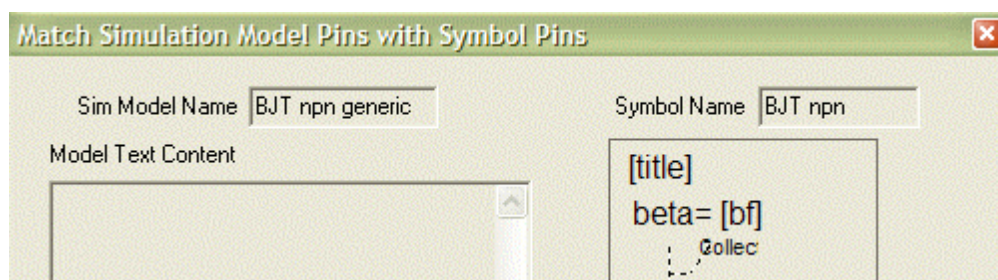


PCB Export

One of the exciting new abilities of B2 Spice A/D v4 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



























































You can also use the included Database Editor to add and edit any of the packages in the database.

























How does B2 Spice V5 measures up



















More for Less - Everything you want for a lot less than the competition, including features they don't even **SPICE/ XSPICE Simulation**

Characteristics	B2 Spice v5 Pro	CircuitMaker 2000	Multisim 2001 Personal
------------------------	------------------------	------------------------------	-------------------------------

DC Operating point			
DC Parameter Sweep		2 variables	
Temperature Sweep	(part of DC parameter sweep)		NO
Transient			
Fourier			
Parameterized transient sweep			NO
AC Analysis (frequency sweep)			
Parameterized AC Sweep			NO
Pole Zero		NO	NO
Transfer function			NO
AC Sensitivity		NO	
DC Sensitivity		NO	
Distortion		NO	
Noise			
DC Operating Point Monte Carlo			NO
DC Sweep Monte Carlo			NO
AC Monte Carlo			NO
Transient Monte Carlo			NO

RF Analysis		NO	
Simulation Options			
Digital Logic Simulation			
Fully interactive, free running digital logic simulation.			NO
Digital probe tool			NO
Wire Value Coloring			NO
Variable simulation speed			NO
Single step feature			
Run from test vector		NO	NO
View unlimited waveforms in a separate timing diagram			Limit- 16
Programmable device in truth table format, espresso format, or state equations			NO
Save simulation into a test vector		NO	NO
Set propagation delays globally to maximum, minimum, or typical		NO	NO

Calculate power dissipation in digital mode for entire circuit		NO	NO
Devices			
# parts/ Models	25,000	8,000	6,000
# symbols	648	400+	150+
Import any standard SPICE library			
Devices may be selected by category			
Devices may be selected by manufacturer			
Devices may be selected by keyword			
Part Search using wildcards			NO
Repeat placement of device			NO
Rotation of devices in 90 degree increments			
Auto or manual designation of devices			
Support for BSIM & MOS 6 models		NO	NO

Create part from circuit and store to parts database		NO	NO
Symbol editor			NO
Store modified models, symbols, or complete parts to database		NO	NO
Wiring			
Manual wire routing			
Wire angle	90/any	90/any	90/any
Reposition any wire with click and drag			
Wire "rubberbands" with device movement			
Bus wires may contain any number of independently numbered wires			NO
Markers can be used to create virtual wire connections		NO	NO
Additional Capabilities			
Import SPICE netlist for simulation		NO	

Export to PCB programs			
Export circuits & waveforms as graphics			
Generate SPICE 3 netlist			
Print Scaling, Fit to page			
Undo/Redo	yes/unlimited	1 lvl undo/redo	1 lvl undo/redo

Animated Schematics

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

"Circuit animation gives me both general and specific insight into the behaviour of the circuit as the simulation is running. For example, I was building an amplifier today and I noticed that the current through the emitter of a key transistor was changing direction, implying avalanche breakdown in the base-emitter junction. Without circuit animation, I never would have realized that this was occurring," said John Broskie, editor of the Tube CAD Journal

