

# Software Review

## B2 Spice A/D v4

Reviewed by Tom Harman

*B2 Spice A/D v4 circuit simulation program. Beige Bag Software, 279 E. Liberty, Ann Arbor, MI 48104. (734) 332-0487, Fax (734) 332-0392, www.beigebag.com, e-mail info@beigebag.com. \$349 Professional Version, \$199 Standard Version, free download Lite Version. Previous B2 version upgrades and competitive product upgrades available.*

B2 Spice v4 is a full-featured analog and digital SPICE electronic circuit simulation program by Beige Bag Software. Beige Bag regularly advertises B2 Spice in *audioXpress*. It seems that Beige Bag believes Spice A/D v4 is a product *audioXpress* readers should use. It made sense for me to review this program as my first submission to *audioXpress*.

### PAPA'S GOT A BRAND NEW BEIGE BAG

There are three different levels of Beige Bag's Spice v4 program. The Lite Version is a junior version of the other two programs. Your schematic may contain a maximum of 25 components, and only six simulations are available. The parts library has 300 devices. At the cost of a free download, this is an excellent introduction to anybody who is curious about how SPICE programs work.

The primary difference between the \$349 Professional Version and the \$199 Standard Version is a larger library of parts (15,000 versus 5,000) and more

#### ABOUT THE AUTHOR

Tom Harman is a video systems engineer for Sony Electronics. He has been an audio hobbyist since age 14. He enjoys designing and building all audio equipment. He can be contacted at [audio@bestbatchyet.com](mailto:audio@bestbatchyet.com).

simulations (19 versus 12). Either version is very attractive and competitive in the SPICE market. Beige Bag has the Professional Edition, fully functional, as a free download. The catch: This version stops working 30 days after you install it on your computer. Thirty days should be long enough for you to decide whether you wish to pay the additional \$150 for the Professional version.

### SPICE, SPICE, BABY

SPICE programs allow you to design and test (the SPICE term for test is "simulate") an electronic circuit, using a computer. No components to buy, no soldering irons to heat up, no smoking components.

Why would you choose to use a SPICE program? For me, there are several answers:

1. Time—I can build a circuit quicker on my computer than I can on my test bench. I can change components quickly using a PC.
2. Education—I find technical material easier to follow if I build the circuit and "see" the ideas presented.
3. Math—SPICE does the boring math for me. More time to experiment.

I do not want to mislead you. A SPICE program will not eliminate the prototype stage. It can shorten the design stage considerably. There are some real-world factors that might not be practical to simulate, such as: lead inductance, PCB capacitance, RF interference, real-world power supplies. Of course, SPICE does not have capacitors that are low distortion, nor can it tell you which cable sounds better.

### INTRODUCING THE SPICE GOALS

SPICE software has been around since the early 1970s. SPICE stands for "Simulation Program with Integrated Circuit Emphasis." Students and faculty at UC Berkeley first developed SPICE and have maintained it since. This is why some people call it "Berkeley SPICE." It has gone through three major revisions during the last 30 years. It has received a digital circuit simulation add-on, XSPICE (for Extended SPICE), which students at Georgia Tech University wrote.

Berkeley SPICE has become the world standard simulation engine. Almost all circuit simulation programs use the Berkeley SPICE engine as their core. This has the advantage of competition between products, but still continuity and familiarity among vendors. SPICE models need to be produced in a specific format; this makes them (more or less) interchangeable from one vendor's version of SPICE to another vendor's version. In fact, many component manufacturers supply free SPICE models of their products to encourage you to use their products in your designs.

SPICE was developed using public funding. The program is an open license. You can legally get versions free. FREE! The audio amateur's favorite word. Why should you spend \$10,000 or even \$99 for something you can get free?

Using the core SPICE program is cumbersome and anti-intuitive. There is no graphic input. You define your circuit using a text editor, such as Windows Notepad. Consider the following example. *Figure 1* is a simple schematic and *Table 1* is the SPICE file version of *Fig. 1*.

One line of text describes one component and how it is connected to the rest of the circuit. "R1 1 2 100.0Ω" tells you that R1 is a resistor, the first lead connected to node 1, the second lead is connected to node 2 and has a value of 100Ω. "Node" is SPICE speak for conductor.

A file of a typical power amplifier would be large, difficult to edit or troubleshoot. Most (if not all) SPICE programs will come with a schematic drawing tool, called "schematic capture." Select your components, place them on your sheet of paper, and then play connect-the-dots to wire your circuit. The software translates your graphic work into a file like the previous example to be used by the simulator.

The core SPICE program comes with only a few basic components: resistors, capacitors, voltage sources, generic NPN transistor, and so on. Commercial SPICE programs come with a library of SPICE models for specific components, such as Analog Devices AD-711, TIP 32C transistor, IRF610 power FET, 12AX7 vacuum

tubes. The SPICE program uses these models to predict how your circuit will behave.

When you specify a Texas Instruments TL071 op amp, you simulate the circuit using a TL071's electrical characteristics, such as bandwidth, gain, current draw, and so forth. The more accurate the model, the more accurate your simulation results will be. When evaluating a SPICE package, examine the library of components carefully.

### WHAT IS A SIMULATION?

A simulation is the computer equivalent of testing your circuit on the test bench. Different vendors will make different simulations available. The core SPICE program is capable of generat-

ing a great amount of data, which can be presented in many different ways, usually in the form of tables and graphs. Manipulating and displaying this data requires programming, and programming costs money. How much programming do you want to buy?

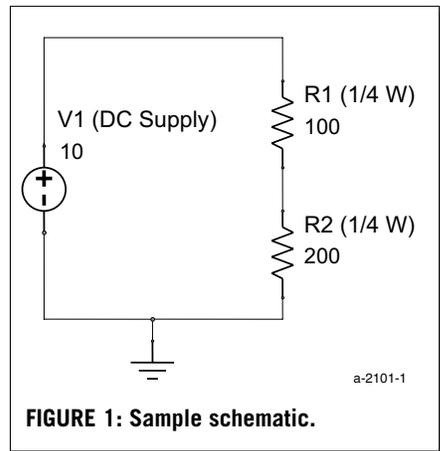
That is about it. Draw your circuit and then run the simulations.

This review is addressed to a "typical" *audioXpress* reader. I do not intend this to be for professionals. Should your international mega-corpor-

**TABLE 1**  
**SPICE FILE OF FIG. 1**

```

*Voltage divider circuit
VIN 1 0 10.0V
R1 1 2 100.0Ω
R2 2 0 200.0Ω
.End
    
```



# LANGREX SUPPLIES LTD

DISTRIBUTORS OF ELECTRONIC VALVES, TUBES & SEMICONDUCTORS AND I.C.S.  
1 MAYO ROAD, CROYDON, SURREY, ENGLAND CR0 2QP  
24 HOUR EXPRESS MAIL ORDER SERVICE ON STOCK ITEMS  
E-MAIL: LANGREX@AOL.COM

PHONE 44-208-684-1166 FAX 44-208-684-3056

---

**A SELECTION OF OUR STOCKS OF NEW ORIGINAL VALVES/TUBES MANY OTHER BRANDS AVAILABLE**

STANDARD TYPES			AMERICAN TYPES			SPECIAL QUALITY TYPES		
ECC81	RFT	3.00	5R4GY	RCA	7.50	A2900/CV6091	G.E.C.	17.50
ECC82	RFT	6.00	5U4GB	SYLVANIA	15.00 pounds each	E82CC	SIEMENS	7.50
ECC83	RFT	8.00	5Y3WGT	SYLVANIA	5.00	E83CC	TESLA	7.50
ECC83	EI	5.00	6BX7GT	GE	7.50	E88CC G. PIN	TESLA	8.50
ECC85	RFT	5.00	6FQ7	SYLVANIA	7.50	E188CC	MULLARD	20.00
ECC88	BRIMAR	6.00	6L6GC	SYLVANIA	20.00	ECC81/6201	G.E.	5.00
ECC88	MULLARD	10.00	6L6WGB	SYLVANIA	15.00	ECC81/CV4024	MULLARD	6.00
ECL82	MULLARD	5.00	6SL7GT	USA	7.50 pounds each	ECC81/M8162	MULLARD	7.50
ECL86	TUNGSRAM	10.00	6SN7GT	USA	7.50	ECC81/6201 G. PIN	MULLARD	10.00
EF86	USSR	5.00	6V6GT	BRIMAR	7.50	ECC82/CV4003	MULLARD	10.00
EF86	MULLARD	15.00	12AX7WA	SYLVANIA	6.00	ECC82/M8136	MULLARD	12.50
EL34	EI	6.00	12BH7	BRIMAR	12.00	ECC83/CV4004	MULLARD	40.00
EL37	MULLARD	30.00	12BY7A	G.E.	7.00			
EL34G	SOVTEK	5.00	211/VT4C	G.E.	85.00			
EL84	USSR	3.00	807	HYTRON	7.50			
EL519	EI	7.50	5687WB	ECG	6.00			
EZ80	MULLARD	5.00	6072A	G.E.	10.00			
EZ81	MULLARD	10.00	6080	RCA	10.00			
GZ32	MULLARD	25.00	6146B	G.E.	15.00			
GZ33/37	MULLARD	20.00	6922	E.C.G.	6.00			
PL509	MULLARD	10.00	6973	RCA	15.00			
UCH81	MULLARD	3.00	7308	SYLVANIA	5.00			
UCL82	MULLARD	2.00						

**SOCKETS**

B7G	CHASSIS	0.60
B9A	CHASSIS	1.00
OCTAL	CHASSIS	1.00
OCTAL	MAZDA	2.00
LOCTAL	B8G CHASSIS	2.50

**SCREENING CANS**

ALL SIZES	1.00
-----------	------

**MANY OTHER BRANDS AVAILABLE**

These are a selection from our stock of over 6,000 types. Please call or FAX for an immediate quotation on any types not listed. We are one of the largest distributors of valves in the UK. Same day dispatch. Visa/Mastercard acceptable. Air Post/ Packing (Please Enquire). Obsolete types are our speciality.

ration buy a license for every engineer's desk? I do not answer that question. This is a review written for *audioXpress* readers, by an *audioXpress* reader.

B2 Spice has the ability to export your schematic to certain PCB layout programs. I do not have access to any of the supported PCB formats. I could not verify the functionality or quality of this feature.

B2 Spice v4 also has digital circuit capabilities. You can design circuits with TTL logic and all that nonlinear stuff. I am an analog/linear person. I do not feel comfortable criticizing or endorsing the digital section of the program. If you are interested in the digital section of B2 Spice v4, please be careful how you apply my comments. Or go ahead and try the download.

### PUT ME IN, COACH, I'M READY TO PLAY!

It was not until after I started writing this review that I revealed my literary intentions to Beige Bag. I purchased the product and tested their customer service as a civilian. I received no special attention. I believe you will receive the same level of service I describe here.

Let me start off on a big positive note. I was impressed with Beige Bag's customer service. I placed my order late Tuesday afternoon, and the CD-ROM was in my mailbox two days later. The product was delivered just as I had ordered. Installing the program on my IBM-style PC was as easy as it gets.

As a test for this review, I e-mailed some technical questions to see how quickly and completely Beige Bag tech support would respond. On my first attempt, they experienced a glitch in their e-mail system. After a couple of days I did not receive a response. I made a phone call to the Beige Bag office and obtained a prompt reply.

The next week I sent several more questions. I made these questions slightly annoying and of a pestering nature. I always received a quick, complete, and courteous response—sometimes within two hours. Beige Bag passed the tech support/customer service test with flying colors.

### RATING THE PROGRAM

I really like this program. It does an excellent job delivering all the reasons to buy a SPICE program. You get features galore and tons of simulation options. The simulations run smoothly. The program was stable and never locked up on me.

When I was experimenting with the program, I trusted the data generated by B2 Spice. Any time I derived a result that didn't make sense, I assumed (quite rightly) that the error was mine. For example: The RIAA preamplifier I simulated did not have enough LF boost. After an hour of adjusting simulation variables, I realized that I had used the incorrect multiplier on a capacitor value (.1 $\mu$ F, not .1F. Duh!).

The 233-page printed manual that comes with the program is adequate. It starts off with some step-by-step tutorials. It took me a couple attempts to go through each tutorial to figure out exactly what was intended. Still, the tutorials are a good introduction to get you up and running. A feature-by-feature description of each part of the program occasionally makes reference to terms that I did not find in the printed manual index or in the program's help file.

At first, I was disappointed and expected more from the supplied documentation. A trip to the bookstore and a search of the Internet, however, caused me to soften my criticism. I reminded myself that this isn't a video game; this is a complicated beast requiring much learning. The volumes that cover the subject in more detail cost \$40 to \$75. I realized that to expect a text like that with a \$199 program is unrealistic.

If you are serious about learning SPICE, expect to purchase additional material on the subject. So far, I have invested \$120 in additional textbooks. Maybe someday, there will be a "SPICE for Dummies."

As I looked over the notes I made while preparing this review, I realized that most of my criticisms of B2 Spice was not the SPICE program itself. Where B2 Spice is lacking is the Windows-related part of the program.

The schematic capture section of B2 Spice is usable and functional, but graphically awkward. I come from a graphics background and have paid my

rent drawing schematics. Maybe that is why I am so critical. The schematics generated by B2 Spice are not presentation quality.

For example: You have the option to "show device names." You wish to show 1N4148, with the mnemonic D1. (Or add 50V to C25, ¼W to R1, and so on.) The 1N4148 appears in parentheses after the D1. What you get is a single string of text, "D1 (1N4148)." When you reposition the D1 text, "(1N4148)" moves along with the D1.

This is an option you turn on globally to all components. There is no way to turn off the parentheses. If there is no appropriate device name, the

### UPDATE

Since submitting the review, Beige Bag has updated A/D Spice to Version 4.1.2. I updated my version by downloading the free upgrade available on [beigebag.com](http://beigebag.com).

I tested the items I specially criticized in my review. Beige Bag nailed it. Printing the schematic is predictable, the graphs display exactly as I specified, zoom works as I want. My criticism of the device mnemonics and ancillary information has been addressed. They are now two independent lines of text that you can place where you like, without parenthesis.

I guess I did a poor job of making my point about the supplied documentation. Jon called it "my big complaint." While there is room for improvement, I was trying to explain why I shouldn't complain. Beige Bag A/D v4 is a tool. I would not expect a 300-page home remodeling book to come with every saw and drill I buy.

The second point I was trying to make is that you will have to do some additional study and obtain additional materials. The same as buying a \$3,000 CAD program doesn't make you an architect, and a 1962 Fender Statocaster won't make you Jimi Hendrix.

Publishing deadlines being what they are, I did not have much time to test beyond my original gripes. I did get some glimpses of new features that are very intriguing. I gave it a strong review in the first place. Beige Bag addressed my complaints in the second place, which amplified my original enthusiasm. Throw in a free trial and the 30% discount Jon mentions, and you have the best deal this side of winning the lottery.—TH

mnemonic has a “( )” tagging along behind it. See Fig. 1 for an example.

To improve their schematic capture Beige Bag should: A) Allow the ancillary text to be visible on a component-by-component basis. B) Kill the automatic parentheses. C) Make the ancillary text position independent of the mnemonic.

The zoom is not accurate. A command to “fit circuit into view” does not work exactly correct. Instead of making every part of my schematic visible, some elements appear outside the schematic window. I got into the habit of executing a “zoom to fit circuit” command, then a zoom out a bit to see the missing part of the schematic. Two steps instead of one.

Printing a schematic is not smooth. I had to adjust printer settings using trial and error. The print preview did not always accurately show what will be printed. To produce a more professional drawing, I used a different graphics program to draw and print border/title block. I made a second pass through the printer to print the schematic.

The save file command does not always work correctly. Sometimes to save the file, I needed to put a resistor into the drawing and then immediately delete it. Then I could save the file.

The program can generate many different types of graphs. Supposedly you can control the color scheme of the graphs as they are displayed on your monitor. The default is white background and various colors for different plots. My preference is a black background.

Every time I changed the default, I got into trouble. At one point, the program started plotting black plots on a black background. Eventually, I resigned myself to living with a white background on my graphs.

I do not want to complain too much. These things fall under the category of annoyances. The excellent value of B2 Spice A/D v4 makes up for the little bits that could be better. It reminds me of an amplifier that has poor cosmetics, but sounds fantastic and costs less than everything else in town.

Beige Bag has only three employees. They put their effort into making a

product that is a great value. I am willing to overlook some of these problems to get my hands on this very useful product. If you are looking for a design tool/aid such as a SPICE program, do yourself a favor and take advantage of Beige Bag Spice A/D v4.

*Manufacturer's Response:*

*Regarding the specific problems in the schematics and graph appearance, we fixed up some of these problems in our 2001 year-end revision. We fixed the Zoom to Fit feature, and the black graph background feature, for example. Regarding the appearance of the parts in the schematic, the user can modify the parts in the schematic to show the mnemonic and device name in separate fields. All symbols can be fully customized and even stored back to the database. This was not well documented in the user manual when Tom reviewed the product, but we are taking steps to remedy this in our documentation. Also, we will change the symbols in the database to separate those two fields.*

*Tom's big complaint is that B2 Spice*

*A/D v4 is difficult for novices to get a grasp of. In the past, we spent a lot of energy working on the flexibility and power of this product. For example, we added monte-carlo complete with histograms for displaying results, and high-frequency models and two-port simulation capabilities. I agree with Tom that circuit simulators are challenging for new users and it's time to focus on solving this problem. I am already working with an audio electronics expert to add help, possibly in the form of wizards, for users to design and test their circuits. I'm planning to release a new, more novice-friendly version in early 2003.*

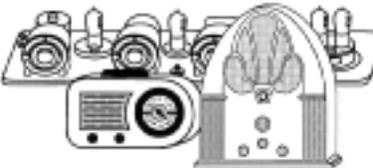
*This is a live product, and we regularly release free patches to our customers with small improvements and fixes to known bugs.*

*In the meantime, please try out our free 30 day, full-featured trial and let us know what you think of the program. And mention this review to get 30% off the retail price when you decide to purchase the program.*

*Jon Engelbert  
President  
Beige Bag Software*

**FREE SAMPLE**

*IF YOU BUY,  
SELL, OR COLLECT  
OLD RADIOS, YOU NEED...*



**ANTIQUE RADIO CLASSIFIED**

**Antique Radio's Largest Monthly Magazine**  
100 Page Issues!  
Classifieds - Ads for Parts & Services - Articles  
Auction Prices - Meet & Flea Market Info. Also: Early TV, Art Deco,  
Audio, Ham Equip., Books, Telegraph, 40's & 50's Radios & More...  
*Free 20-word ad each month for subscribers.*

**Subscriptions: \$19.95 for 6-month trial.**  
**\$39.49 for 1 year (\$57.95 for 1st Class Mail).**  
*Call or write for foreign rates.*

---

**Collector's Price Guide books by Bunis:**

<b>Antique Radios, 4th Ed. 8500 prices, 400 color photos .....</b>	<b>\$18.95</b>
<b>Transistor Radios, 2nd Ed. 2900 prices, 375 color photos .....</b>	<b>\$16.95</b>

Payment required with order. Add \$3.00 per book order for shipping. In Mass. add 5% tax.



**A.R.C., P.O. Box 802-A16, Carlisle, MA 01741**  
**Phone: (978) 371-0512 — Fax: (978) 371-7129**



**Web: [www.antiqueradio.com](http://www.antiqueradio.com) — E-mail: [ARC@antiqueradio.com](mailto:ARC@antiqueradio.com)**