

Re: Wanted: LM-709 (Spice model) National Op-Amp

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2005-09/msg00119.html>

- *From:* Paul Rako <sp_a_mpa_u_l@xxxxxxxxx>
 - *Date:* Fri, 23 Sep 2005 10:11:23 GMT
-

Well, it was an LT guy that told me LT-SPICE was a PSPICE variant but I don't have the need to trash people, (even anonymously) that some people do so I will not mention who. I certainly will yield to Mr. Engelhardt on anything about LT SPICE because my LT friends say he is THE MAN for this. After all, we know Swanson would rather work to death one man rather than hire a department. Mike does the work of two departments and I really respect that. LTSPICE is his baby and he has a right to be proud of all that work.

Can he tell me the relationship between SwitcherCAD and LTSPICE? Are they the same thing? And I was once told that LTSPICE does not allow import of models from other vendors like ADI and National. Sounds like bunk but it is a proprietary program after all.

BTW Mike, Bill Gross is a recently retired Vice-president and a former IC designer so I will give him the complement tomorrow that you consider him an apps guy like me. Swanson really must have you chained to a workstation.

Now as to my SPICE experience-- well, Berkeley SPICE and Hollerith cards yeah, a good bit of PSPICE, Intusoft ICAPs, at HP we had this thing I think called DR Deatch or something that was supposed to converge really well and my impression of all of them was they are crap. But this was almost 10 years ago.

I will look for some of the oscillators and stuff that blow up or when I get around to it publish a circuit I was using SPICE on a few months ago and everyone can excoriate me just because I didn't know to set abstol to something and put the .bs command in the deck. Well duh, whenever I have to slow down edges or loosen up accuracies to get a convergence it just seems like a real good time to put down the mouse and pick up a soldering iron. Maybe I am just a scaredy-cat.

Re: Wanted: LM-709 (Spice model) National Op-Amp

I would like to graduate past 7th grade and "you are full of shlt" comments so let us act like technical people and deal with facts.

Several years ago EDN magazine did a circuit and gave it to 6 SPICE vendors and Jim Williams at Linear Tech. If I remember about half of the programs failed to converge and the rest gave wrong results, sometimes wildly wrong compared to Jim's real board. Was there a memo I missed? Have models and SPICE engines gotten that much better?

Can anyone really get any kind of mid-range SPICE to deal with non-linear magnetics? Does anyone trust it to design a complex flyback converter?

Are there really A to D models that give the representative data output of the real-world signals? Not just the math and correlations involving the sampled-data theory but the real things going on in the analog and digital sections? (All board-level models of course, not "real" transistor-level models.) Does LT offer models of those new fast converters they make?

OK, the SPICE behind National's WEBENCH uses a later version of the SPICE engine than PSPICE. I have heard one called level or stage two vs a three. So what are the substantive differences? Does PSPICE suck as much as everything else Cadence seems to ruin? Maybe I am complaining about my Model T when everybody else is in a Prius.

I was at Arrowfest tonight where somebody said all SPICE does is solve a matrix. That is what Berkeley SPICE is. What everyone else is doing is writing code to try and get the solution to converge when the math blows up. I had dinner with a guy from PSPICE years ago and he said all of their work is doing code like that, to keep things from blowing up. How comforting. Is this wrong?

When I was at HP we were designing automotive diagnostics. I defy anyone to make a good model of a spark plug gap since most attempts had real trouble converging and then you realize the flame-front and pressure in the cylinder affects the signal. Did I miss that memo as well?

People, people, I am not being combative, I work in the on-line SPICE group at National for crying out loud and really want to use it as much as possible. Please don't jump on me like I am criticizing your religion or politics or wife. It just seems like every time I wade into another type of complex circuit with SPICE I soon feel like I need a CS degree and a month of trial and it is just so much easier to just build the thing. Remember I am talking board-level here, not something that you want to simulate to death since there are 100k of masks at stake. That is why I like our WEBENCH tool. We have

Re: Wanted: LM-709 (Spice model) National Op-Amp

a whole department including a couple of apps guys like me to insure that we can give good results when we run a simulation. But we build the circuits with the same exact components and make sure that the SPICE agrees with real-world values so our customers don't have to. Is everybody out there designing things with such similarity to their previous designs they know they can trust the simulations?

Oh, if I have brought Kevin and Mike together then I guess there is redemption in electronics after all.

Now to the important stuff, maybe I can get Pease to sign-up for Google Groups but failing that I can at least post his reply to my lederhausen question today:

=====

*** Hello, Paul,

In reply to your comment.....

**** I do not recall ever wearing or owning Lederhosen, when I was in college. I recall specifically that I did not. But I did wear shorts. In the winter. When bicycling. In the snow. When I went winter-mountaineering, up in New Hampshire, I wore shorts plus long-johns. Red long-johns.

*** I know nothing about silk-lined Lederhosen, and I know nothing about the Lampoon's parody.

*** Since I never had or wore Lederhosen, then I'm sure that some of the ones I didn't wear were silk-lined, and some of the ones I didn't wear were NOT silk-lined. / rap
=====

Hey Mike; Tim Regen's birthday tomorow, come over to Bldg T (The Tasty Subs on Lawrence Expressway by Arques) and I'll buy you a beer.

Re: Wanted: LM-709 (Spice model) National Op-Amp

Paul

Kevin Aylward wrote:

Paul Rako wrote:

And, if you buy PSPICE for 10k and still spend the 5 or 10 million those two departments, the modeling and process departments, you can still get good results for transistor-level simulation. Linear Tech uses PSPICE for IC design. I have been told that LT-Cad is just a variant of PSPICE so I am somewhat baffled how people can claim it works "better".

Well, not often I support Mike, but you way off base here. The LTSpice engine is probably the best there is on PCs as far as simulation speed and convergence goes. Its the GUI that leaves a lot to be desired.

Kevin Aylward
informationEXTRACT@xxxxxxxxxxxxxxxx
<http://www.anasoft.co.uk>
SuperSpice, a very affordable Mixed-Mode
Windows Simulator with Schematic Capture,
Waveform Display, FFT's and Filter Design.