

Re: Basic queries on using LT Spice

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2004-12/0004.html>

From: Tony Williams (tonyw_at_ledelec.demon.co.uk)

Date: 11/30/04

Date: Tue, 30 Nov 2004 10:38:46 +0000 (GMT)

In article <csfoq0dbgbvv6aiqq37ua6tnf27le3p57s@4ax.com>, Terry Pinnell <terrypinDELETE@THESEdial.pipex.com> wrote:
> *These are very basic questions, some no doubt in the Duh!*
> *category. FWIW I've read the tutorial, albeit quickly. But a few*
> *quick pointers from the established LT Spice users could save me*
> *hours please! Or maybe there is an existing source for learning*
> *practical details which I've not yet found? I'm so familiar with*
> *CM that getting comfortable with LT Spice may take me a while,*
> *but I'd like to master it as its simulation facilities are*
> *plainly superior.*

LTSpice is my first Spice ever, so can't really answer many of your questions Terry..... but I do have to say that I'm mightily impressed with some of the simulations it has done on known circuits.

> *3. In the absence of such indicators, suppose I just choose the*
> *first and get a plot headed 'V(n001)'. Can I *now* get the*
> *schematic to show that choice? Is there any other way of finding*
> *what voltage or current I'm seeing, apart from clicking each node*
> *with the cursor and observing the result?*

Left click in the schematic window to make that window the one with the input focus. Then wander around around nodes and components to see the scope probe or current display being offered. As each one is offered it's description in given down in the bottom left corner, in text.

> *6. To edit the schematic, I take it you must first close all*
> *plots?*

No, just left-click in the schematic window to make it the active window. When done, re-run the simulation and a new plot will overwrite the old one.

> *10. For a Transient Analysis, with whatever defaults running,*
> *what are the steps for changing the key parameters: Start Time,*
> *Stop Time, Step Time, Max Step Time.*

sci.electronics.cad: Re: Basic queries on using LT Spice

Drop down the 'Simulate' pane, 'Edit Simulation Cmd' is the bottom option.

--

Tony Williams.