

Basic queries on using LT Spice

Source: <http://sci.tech--archive.net/Archive/sci.electronics.cad/2004-12/0002.html>

From: Terry Pinnell (*terrypindel@THESEdial.pipex.com*)

Date: 11/30/04

Date: Tue, 30 Nov 2004 09:42:04 +0000

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.

1. What's the difference between Move and Drag? (Both appear to move a selected part or group of parts around, until left click fixes them in new position.)
2. With a circuit drawn, I click Run and get the 'Select Visible Waveforms' dialogue, with choices like V(n001),V(n002),etc. But how do I know what those nodes are? IOW how do I first ensure the nodes are displayed on the schematic.
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?
4. If I decide to take the Help's advice "The easiest method is to simply probe the schematic. You simply point and click at a wire to plot the voltage on that wire," how do I by-pass that initial dialogue? Even with none of its choices selected, if I click Cancel, it still plots V(n001). I could of course then close that, to get a clean slate ... if I knew which node to click!
5. With several voltages plotted, how do I separate them into individual windows?
6. To edit the schematic, I take it you must first close all plots?
7. In CM I can select a 'component' called '.IC' and connect it at a node. In LTS I see I add this as a Spice Directive. But how do I connect it?

8. Is there any way to re-assign shortcuts? For example, I have ctrl-z ingrained for Undo, and Ctrl-c for Copy, not F9 and F6.

9. Where can I find step by step examples of adding/importing new models?

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. Are these all entered as Spice Directives? If so, is there a succinct summary of the main directives and their syntax please? (In CM, I use only the GUI.)

--

Terry Pinnell
Hobbyist, West Sussex, UK