

## Re: Basic queries on using LT Spice

**Source:** <http://sci.tech-archive.net/Archive/sci.electronics.design/2004-12/0066.html>

---

**From:** John Smith (*kd5yikes\_at\_mindspring.com*)

**Date:** 11/30/04

Date: Tue, 30 Nov 2004 15:38:58 GMT

----- Original Message -----

From: "Terry Pinnell" <terrypinDELETE@THESEdial.pipex.com>

Newsgroups: sci.electronics.cad,sci.electronics.design

Sent: Tuesday, November 30, 2004 3:42 AM

Subject: Basic queries on using LT Spice

- > *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.*

A dedicated LT Spice newsgroup:

<http://groups.yahoo.com/group/LTspice/?yguid=161100399>

- > *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.)*

If the component is connected in the schematic, using drag will pull the wires along with the component when you move it. Move moves the component without stretching the wires. Note that you can draw a rectangle with drag or move to select an entire area to move or drag.

See Help/Schematic Editing.

- > *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.*

I believe this question has been answered. If you don't want to pass the crosshairs over the wire to read the node ID below, you can label the wire. See Help/Trace Selection.

- > 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?*

No. Not that I aware of. Note that you single click a node to plot it. Clicking the same node twice in succession discards all plots but that node. This is not what you asked, but it is handy to know.

- > 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!*

On a new schematic, or when you have closed the plot window, LT Spice doesn't know which node you want plotted, so it asks. You must either tell it which node to plot or it will plot node 001. After the initial run, it will continue to display the node or nodes previously selected until you close the plot window again. If you have a display which you like, you can click on the plot window somewhere to make it the focus, then save the plot. Save the plot with the same name as the schematic. The next time you do a run after closing the plot window (or after closing LT Spice and then reloading), LT Spice will look for a plot file and use it to display your plots as selected previously.

See Help/Save Plot Configurations

- > 5. *With several voltages plotted, how do I separate them into*
- > *individual windows?*

I think this has been answered, but, right click the plot pane. Select Add Plot Pane. Click the node of interest.

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

No.

- >
- > 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.*

I've never used this, but look under Help in Shortcuts.

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

Look in Help/FAQs/Adding Third Party 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.)

With focus on the schematic, click Simulate/Edit Simulation Command. A dialog box opens for you to insert the values without resorting to directives. Otherwise, see Help/Dot Commands.

Note that my answers here are based on my experience and there may be other and better answers.

LT Spice comes with example programs and pretty good help, especially for a free, powerful program. I strongly recommend you read the Help and run the examples. I should do this, too.

Good luck.

John