Draft – Incomplete - Look for Coming Attractions☺

Procedure to Run LTSpice IV on Workstations in Hewlett Electronics Laboratory

Methods of Generating a SPICE Deck:

1. To use SPICE you have to build a text file with information on the netlist, the device models, and the desired calculations. To do this, you need some knowledge of SPICE syntax and naming conventions. Appendix A in your textbook has most of that information. I have put the SPICE appendix from an earlier textbook on the class website under Course Materials/Sedra and Smith on SPICE Syntax. Finally there are innumerable sites on the web with such introductions, including http://web.stanford.edu/class/ee133/handouts/general/spice_ref.pdf, http://www.ecircuitcenter.com/SPICEsummary.htm, and https://newton.ex.ac.uk/teaching/CDHW/Electronics2/userguide/sec2.html. I recommend staying away from descriptions of PSPICE because that proprietary version of SPICE has many non-standard features that may not be supported in LTSpice IV.

2. The first way to create a deck is to draw the circuit by hand, label its nodes, and then use a text editor like Notepad or emacs to write the file. I recommend using this method when getting started because it helps you understand the ideas behind the process. It works well for very simple circuits. You will also find you need to be able to read SPICE files if you continue to work with circuits.

3. LTSpice has a simple schematic entry tool built into it. Instructions on how to use it are included under the Help menu and it really is easy to learn and use. I use and recommend this capture tool only for circuits that will not actually be built either as an integrated circuit or as a printed circuit board.

4. Finally, the *DxDesigner* software suite that is used in ENGN1630 can be used to turn a schematic into a SPICE file. Draw the circuit using symbols from the En162 and SPICE_Primitives libraries. (Most of the other libraries do not support SPICE extraction. You can't model a simple connector in SPICE nor is it practical to model most digital devices with LTSpice.) See details below.

Using LTSpice IV Itself:

1. Start LTSpice from the start menu as: Start>All Programs>Electrical>LTSpice>LTSpice IV.

2. If you have already prepared a SPICE deck by hand or by schematic entry from Mentor Graphics xDxDesigner, then use the menu bar File/Open sequence and browse to your text file. Be sure to chane the file type option on startup from “Schematics (*.asc)” to “Netlists (*.cir, *.sp).”
3. If you wish to create the SPICE deck from the LTSpice schematic tool either choose File/Open/New Schematic and begin drawing or browse to the .asc file already drawn.
4. Check that the component model libraries are properly listed in the file. You need two SPICE directives to cover all the components you might use in a project in this class:

.LIB "P:\programs\MentorGraphics\PCBLibraries\SPICE_Libraries\AnalogModels.lib"

.LIB "P:\programs\MentorGraphics\PCBLibraries\SPICE_Libraries\ENGN1620LabComponents.lib"

5. Check that there is one and only one analysis directive, that is, a line asking for a transient analysis in the time domain (homework 2 is typical example), frequency response (.AC DEC N Fstart Fstop), or operating point (.OP) in the file. Mentor generated files will have three such commands all commented out. You uncomment the one you want. Files generated from the LTSpice schematic capture tool will only have a control directive if you add it to your schematic with the Edit/Add control directive command.
6. Run the program by hitting the button with a figure of a running man.
7. When the run finishes, there will be a blank plot screen. Right click on it and choose Add Traces; select the signals you want to see.
8. To put a set of cursors on the data, left click at the top of the plot on the name of the signal you want to examine. You can move the cursors by dragging with the left button of the mouse. Data appears in a data window to the right.
9. You can export a text file with the data using the File/Export action.
10. You can open another window for plotting using the Plot Settings/Add Plot Pane menu bar command. Usually you do this when a plot is crowded or when signals have very different scales such as current and voltage measurements.

Doing Schematic Capture within LTSpice:

1. The Help menu offers good support for schematic capture. Use it!
2. To start a schematic use menu bar File/New Schematic. Nothing appears to change when you do – the gray background is the schematic sheet.
3. Add components to the schematic with the menu Edit button. The Edit menu itself offers the basic resistor, capacitor, and diode elements directly. For other things you go through Edit/Component and select things from a larger dialog set. Naturally Linear Technology ships LTSpice with models for all their proprietary devices. The commonest basic blocks are listed in this dialog as:

Name in Component List	Object
voltage	Independent voltage source
current	Independent current source
npn & pnp	Bipolar junction transistors
e	Voltage dependent voltage source

f	Current dependent current source
g	Voltage dependent current source
h	Current dependent voltage source
opamp2	Operational amplifier with power connections. All standard macro models need power connections.

4. Be sure to give net names to at least the principal points in the circuit so they are easy to identify in the output data. You do this with the Edit/Add Label command.
5. You add properties, such as resistance, capacitance, model name, etc., to components by right clicking on one and filling in the dialog table that opens. This tool is really not satisfactory for board or IC design so you do not need to enter information that is irrelevant to simulations such as manufacturer, part number or tolerance.
6. Spice directives are statements like the .LIB include lines for library files. You add these to the SPICE deck by Edit/Add SPICE Directives and enter the text.
7. Analysis control lines, like .TRAN, are entered with Edit/Add Spice Analysis. Tabs guide you to correct syntax for the desired analysis.
8. Shortcut keys are quite useful for speeding entry. The keys I find most useful are:

Key	Function
<Alt> + E	Open the Edit menu.
F3	Add a wire or net.
F4	Add a net label.
F5	Delete a component, wire, or directive.

Using Mentor Graphics' *xDxDesigner* for Schematic Capture and SPICE Simulation in LTSpice IV: