PSPICE Quick Reference Guide: ECE1341

The following is a very brief list of commands and procedures for PSPICE version 8.

**Getting Started:**

From a NUNet machine: Start>NUNet>College of Engineering>MicroSim Eval8>Schematics

**Drawing the Circuit:**

Getting Components: `ctrl-g` opens a menu of parts that are listed alphabetically. Highlight the part you want to use, hit the Place button and move the cursor to the drawing area to place the part. *Remember* that every circuit must have at least one grounded node.

Rotation of Components: `ctrl-r` will rotate the component by 90° if you need the component in a different orientation. The command may be used multiple times to rotate 180 or 270 degrees.

Wiring Components: `ctrl-w` allows you to draw wires between components. *To avoid unwanted connections where wires cross, do not draw wires that cross near the intersections of other components and wires.*

Altering Component Values: Double click on the *default* component value to pop-up a menu. Use that menu to change the value.

Units and Suffixes: Units for all values are volts, amperes, ohms, farads, henrys, etc. You can use the following suffixes to indicate extremely large and small component values: `p=10^{-12}`, `n=10^{-9}`, `u=10^{-6}`, `m=10^{-3}`, `k=10^{3}`, `meg=10^{6}`. SPICE ignores all characters after the suffix. Do not leave a space between the number and the suffix. Example: a 1μF capacitor may be labeled as 1E-6, 0.000001, 1u, 1uF, or 1uxxxx. However, 1 uF will not work. SPICE is not case sensitive, `u←→U`.

Labeling Nodes: To label a node in the circuit, double click on a wire that is associated with the node. Enter a name for the node in the dialog box. (For example, it is useful to double click on the output node and label it Vout.)
This makes identifying the node much easier when plotting the simulation results.

**Preparing the Input Source(s):**

Double clicking on any voltage or current source will open a menu that allows the parameters of the source to be chosen. Make sure to use the "Set Attr" button to enter the actual values into the source description.

**Telling PSpice what you want it to do:**

Pspice can do several types of circuit analysis, the most commonly used are

- Transient: $V_{out}$ vs. time
- DC sweep: $V_{out}$ vs. $V_{in}$ (...a transfer characteristic)
- AC sweep: $V_{out}$ vs frequency

Select the type of analysis that is needed by pulling-down the "Analysis" menu and choosing "Setup…". Click the appropriate box to select the analysis type *and* push the button next to the box to set the limits of the analysis. For example, a DC sweep requires that you tell PSpice the *name* of the input source, the starting input value, the ending input value, and the increment size.

**Running the Simulation:**

Once the circuit is drawn and all of the parameters have been correctly entered, you can simulate the circuit's operation by selecting the "Analysis" menu and choosing "Simulate". When the simulation is complete, PSpice will automatically start a graphing program called "Probe". At this point you will see a black screen with axes plotted.

Note: The NUNet installation will generate several annoying error messages *even* if your circuit is working perfectly. These error messages are caused when PSpice attempts to write back-up copies of your work to a protected network drive. Just click *ok* to acknowledge that no back-up file was generated. If you made any errors in the setup, however, you will get REAL error messages. Be sure to read these messages in order to understand how to debug your circuit.
Using Probe:

_Probe_ allows you to plot any current or voltage in the circuit. To add a voltage or current to the plot, pull down the "Trace" menu and pick "Add…"

A dialog box will appear that lists ALL of the available quantities that can be plotted. Click on the ONE that you want and push the OK button.

It is probably clear that quantities like I(R1) are the current through the resistor R1. The notation V(R1:1) means the voltage at the "first" terminal of R1. So if you want the voltage across R1, it is necessary to enter V(R1:1)-V(R1:2). Mathematical functions can be selected from the menu on the right-hand side of the "Trace>Add…” dialog box.

IF you labeled nodes in your original circuit, these node voltages will also appear in the dialog box. For example, if you labeled the output node with "Vout", you should see an entry for V(Vout). This voltage is always the potential relative to ground. In complicated circuits, there may be hundreds of possible choices for plotting and it is useful to provide your own labels to reduce the confusion.

-JH…8Nov01