# *PSPICE Hints and Tips*

**Components** *(Select Draw>GetNewPart... or use Cntl-G)*

Basic parts

| | |
|---|---|
| GND_ANALOG | Ground |
| R | Resistor |
| C | Capacitor |
| L | Inductor |

***<u>CAUTION:</u>*** *"M", when used as an engineering suffix on things like resistor values, is interpreted as "milli Ohm" instead of "Mega Ohm"!!! The cause of this apparent stupidity is that SPICE generates a "netlist" (a text description of the circuit elements and connectivity), and sends this to the simulator. The simulator was developed in 1970 when upper and lower case was not distinguished by programming languages. It has plagued Spice users ever sense... YOU MUST USE "MEG" to get MegOhm.*

Voltage/current sources

| | |
|---|---|
| VDC | Battery/supply |
| VAC | Small signal AC analysis source (with DC value too) |
| VSIN | Sinewave for Transient (time-domain) analysis |
| VPULSE | Pulse train, Step, or square-wave source |
| IDC, IAC, ISIN, IPULSE | Current source versions of above. |

***<u>CAUTION:</u>*** *Currents are positive going <u>into</u> a source! (except for IDC sources which show polarity explicitly.)*

Misc parts

| | |
|---|---|
| POT | Potentiometer |
| C_var | Variable capacitor |
| XFRM_NONLINEAR | Transformer with magnetic core |
| E | Voltage controlled voltage source |
| G | Voltage controlled current source |

Popular / useful Semiconductors (in Draw>GetNewPart>Libraries>Eval)

| | |
|---|---|
| D1N4148 | Common small-signal silicon diode |
| D1N4002 | 1A Silicon rectifier |
| Q2N2222 | General purpose medium power NPN BJT |
| Q2N3904 | Small signal NPN amplifier |
| Q2N3906 | Small signal PNP amplifier |
| IRF150 | Power MOSFET (n-channel) |
| IRF9140 | Power MOSFET (p-channel) |
| uA741 | Popular (but old) dual-supply opamp |
| LM324 | Inexpensive single-supply opamp |

**Analysis Setup and Simulation** *(Select Analysis>Setup..., and then Analysis>Simulate)*

Pspice, like standard Berkeley SPICE, has several "analysis modes". The most useful are:

| | |
|---|---|
| Bias Point Detail | DC bias solution with caps open and inductors shorted |
| AC Sweep | Frequency response (using small-signal linearized BJT/etc models) |
| Transient | Time domain analysis (using most accurate models.) |

**Viewing Bias Solution**

Use *Analysis>DisplayResultsOnSchematic* to show node voltages and/or branch currents directly on schematic. (Note: Currents are positive when entering a node.)

**Plotting AC Sweep or Transient Solution**

Label nodes of interest before simulating, by double clicking on a node on the schematic.
Then simulate.
Pspice "Probe" window should open when the simulation completes.
Select *Trace>Add* and then select a voltage or current from the list.
Double click on axis of plot to allow changing limits/scaling.

**Useful Tricks**

AC Sweep Analysis Setup

You must set the ***AC*** parameter on your circuit's input signal source to a non-zero value.
The circuit behavior is insensitive to the value you use. That is, no "clipping" or other real-world saturation effects will occur, even if you use a value of 100,000 V !
A value of 1V (or 1A for a current souce) is often useful, since the circuit output is then equal to the circuit gain.

Transient Analysis Setup

Determine the highest frequency in your circuit (usually the frequency of the VSIN source you apply).
Then, on the ***Analysis>Setup>Transient*** form, set ***final-time*** to be some multiple of the period corresponding to this frequency. (E.g. for 1kHz, set to 2ms to simulate for two cycles.)
On the same form, set ***print-step*** <u>and</u> ***print-ceiling*** to be equal to about 1/20th to 1/50th of a period, to guarantee enough points per cycle for smooth plotting.

Measuring Input/Output Z

Perform an AC Sweep analysis with a current source of 1A (see discussion of AC analysis above.)
Circuit voltages shown in plots are then equal to the impedance.

Stability

An unstable circuit will NOT show up as having an infinite gain in AC sweep analysis.
You must temporarily modify the circuit to examine the loop gain/phase margins to know if the circuit will be unstable.

Simulating Oscillators (and other circuits known to be unstable)

Real-world unstable circuits begin oscillating due to noise in the components. Pspice circuits may not oscillate, since noise is not present in a normal Transient analysis. You can trick the circuit into oscillating however, by either:

> Changing your Power Supply sources from DC to Step type (using VPULSE) to mimic the turning on of a supply, or
> Forcing a small current pulse into a node (using IPULSE) to "kick" the circuit and get it started.

3/25/02 WBK