

PSPICE NOTES

INDEX

AC Sweep.....1	parts 1	simple voltage source..... 1	units1
bias point detail1	PER 2	sinusoidal voltage source2	VPULSE.....2
DF2	piecewise linear voltage	step ceiling 1	VPWL.....2
distortion factor.....2	source 2	TD2	VSIN.....2
labels.....1	pulsed voltage source.....2	TF2	VSRC.....1
multiple parameters.....2	PW.....2	TR2	
PARAM.....2	setup 1	transient.....1	

USING SPICE

GETTING STARTED

Start **Pspice Schematics**. Other Pspice components will be launched automatically as needed.

From the menu select **Draw/Get New Part** to bring up the part dialog box (or use Ctrl + G or click the part button). Select a part, click place and click the screen to place it. You can rotate the part before or after you place it by pressing Ctrl + R. Part parameters may be adjusted by selecting and double-clicking on the parameter if it is displayed or by double-clicking on the part to bring up the parameter dialog box.

Parts can be moved by first selecting and then dragging, or by drawing a box around several parts and moving as a group.

To connect parts select **Draw/Wire** from the menu (or use Ctrl + W or click the Draw Wire button). Press ESC to stop this. Wires and parts may be repositioned by selecting and dragging.

Every circuit needs a **ground**.

VALUES AND UNITS

Units of time, seconds, milliseconds, microseconds, nanoseconds, picoseconds are written with the abbreviations s, ms, us, ns, ps, e.g. 10us.

Units of resistance, k-ohms, meg-ohms, are written with abbreviations k and meg, 10k and 10meg. **Note that 10M is evidently 10 micro-ohms.**

LABELS

Labels can be edited by selecting and then double-clicking on them. Letter and numbers are fine; most other characters will cause errors, e.g. r' won't work, but r- is okay.

Parts that normally don't have labels, like wires, may be labeled as well. All these labels will be available to reference when creating plots.

SETTING UP TO PLOT DATA

The menu commands **Analysis/Setup** bring up the setup dialog box. This dialog box offers numerous sub-dialog boxes to set up various types of data collection.

AC Sweep Use this dialog to set up to collect data for a frequency response curve.

Bias Point Detail This dialog is checked by default.

Parametric This dialog is used when you wish to analyze using multiple parameters for a component such as multiple values for a resistor. See PARAM on page 2.

Transfer Function The input source must be a voltage or current source, I believe.

Transient The first order of business is to enter something in the **Step Ceiling** field, like 20ns; this controls the resolution of plot data.

The MicroSim Probe box comes up after a circuit analysis is run. To plot something, select Trace/Add.

There seems to be no way to get the voltages and currents to print legibly on my Xerox printer. They are solid black boxes.

PARTS

PART ABBREVIATIONS

Some of the part abbreviations are not intuitive:

capacitor	C
current-controlled current source	F
current-controlled voltage source	H
ground, analog	AGND
source, AC/DC current	ISRC
source, AC current	IAC
source, transient sine current	ISIN
terminal	BUBBLE
voltage-controlled current source	G
voltage-controlled voltage source	E

VSRC Simple Voltage Source

DC: Only for DC bias point analysis

AC: For AC Sweep simulation; the amplitude of the sine wave

TRAN: For Transient Analysis; voltage

VSIN Sinusoidal Voltage Source

DC: Only for DC bias point analysis
AC: For AC Sweep simulation; the amplitude of the sine wave
VOFF: For Transient Analysis; voltage offset
VAMPL: For Transient Analysis; voltage amplitude
FREQ: For Transient Analysis; frequency
TD: For Transient Analysis; time delay for starting the sine wave
DF: For Transient Analysis; distortion factor (I think) for the negative-going amplitude of the sine wave:
 $A \exp(-DF T/2)$

VPULSE Pulsed Voltage Source

DC: Only for DC bias point analysis
AC: For AC Sweep simulation; the amplitude of the sine wave
V1: Voltage at the bottom of the pulse
V2: Voltage at the top of the pulse
TD: For Transient Analysis, the time delay between $t=0$ and the start of the pulse
TR: Rise time of the pulse
TF: Fall time of the pulse
PW: Pulse width, width of the top of the pulse
PER: Period

VPWL Piecewise Linear Voltage Source

DC: Only for DC bias point analysis
AC: For AC Sweep simulation; the amplitude of the sine wave
The other parameters consist of time and voltage pairs, T1, V1, T2, V2, etc., that represent points which are connected by straight lines to form the waveform.

PARAM Parameters

Example: Setting up multiple values for a resistor.
Place the **PARAM** part on the drawing. Double-click on the part to bring up its dialog box.
Under **NAME1=** assign a unique name for the parameter list. (Up to three different components can be assigned parameter lists with this one PARAM part.)
Under **VALUE1=** assign a default value. Close the dialog box.
Double-click on the resistor value at the resistor. Replace the value with the parameter name that you assigned under NAME= in the PARAM dialog box and enclose the name in curly braces.
From the menu select **Analysis/Setup/Parametric**. Select Global Parameter. Fill in **Model Name** (the resistor "value") without the curly braces this time.
If you want to use discrete resistor values, select **Value List** and fill in values in the **Values** box like this: 50k 100k 200k.
Run the simulation. When you go to **MicroSim Probe**, you will be presented with a list of the parameters and can choose to plot any or all of them.