File: g:\courses\fall2009\spice_try.doc RWN 08/31/09

To Get Used to PSpice with Models, etc.

Op-amp circuits

1. Go to the web page: http://electronicdesign.com
2. In the upper right search box enter the ED online ID number: 21519
   This will bring up the paper "Full-Wave Active Rectifier Requires No Diodes"
3. Implement Figure 1 in PSpice. For the op-amp you can use almost any one but the uA741 should be available in most versions. Choose the bias voltage Vs to be 15V (this will be listed as Vdd in many op-amp models). Insert a sinusoidal source, Vsin (not Vac), of amplitude less than the bias voltage. Choose all resistors to be 1Meg Ohm. Run Spice transient analysis for at least 5 periods and check that your results match the rectified ones of the paper.
4. Investigate the op-amp model, {select an op-amp and under edit use PSpice Model}.
5. Insert the part PARAM (in the special PSpice library) and choose the value of R2 to be the parameter Rpar [on the schematic as {Rpar}with default 1Meg] and do a parametric run [you will need to configure/profile with a check in the parametric sweep box and on the use of a global parameter Rpar, no {} are used here or in PARAM] over the range of 1KOhm to 10 Meg Ohms .
6. Go to the Texas Instruments web page: http://www.ti.com and search for the op-amp OPA374 used by the author of the paper. Read over its data sheet and note that there are PSpice files for it near the bottom of the page under simulation files OPA(these are for version 8 and if used for later versions would need to be converted). Read the model file, Opa374.MOD, and note the large number of components (too many to run in evaluation versions).

Diodes

1. Locate any/all of the diodes 1N4001 = standard pn junction diode, 1N4148 = high speed diode, 1N4735 = zener diode [some may have the Spice designator D in front of their number].
2. For the ones available make a PSpice file with the diode, a series resistor and bias battery of 3V (and ground = node 0 {you may need to change the node name to 0} which is always needed for Spice to run). Do a DC run to plot the diode curve and from that choose the resistor to intersect above the break point. Compare what is needed for the different diodes.

Transistors (most of these will be on the ENEE 303 web page)

1. Locate the PSpice files for the BJTs 2N3904 and 2N3906 (they may have a Q Spice designator in the Spice part name. Look at the model data in their library files and compare.
2. Locate the PSpice files for the MOS 4007 transistors (under the RCA part 3600) and compare the Pchannel with the Nchannel models.

G and E value components

1. Locate Gvalue which is in the ABM PSpice library. Use it to set up the differential Equation $3dx/dt=5\tanh(2x)$ with x the voltage of a 3Fd capacitor and input to the Gvalue.
3. Set up Evalue to model an ideal op-amp with gain of 10^6 near the origin and saturating at Vdd and Vss=-Vdd; you can use (.)tanh(.) to accomplish this.