

PSPICE Tutorial
Dr. Peter Osterberg

1. Login to a PC and go to your P-drive. Create an EE351 folder. Within the EE351 folder, create a PSPICE folder. Within the PSPICE folder, create a hwxx folder (e.g. hw3).
2. Launch PSPICE Schematics.
3. In the blank sketching window:
 - a. Sketch your circuit (r=resistor, l=inductor, c=capacitor, VAC=AC voltage source, ground=GND_EARTH, etc)
 - b. Click on interesting nodes (e.g. the output node) and label them with appropriate names (e.g., VOUT).
 - c. Place Voltage Markers and Voltage Phase Markers (if needed).
 - d. Save As: hw2.sch (for example)
Note: make sure you explicitly type in the .sch extension
 - e. In Analysis Set-Up, set parameters as needed for:
AC Sweep (Decade, Total Pts, Start Freq, End Freq)
DC Sweep
Transfer Function
Transient
 - f. Simulate (you will probably have to do this twice, since it almost always crashes the first time).
 - g. Check PROBE results
 - 1) Delete the V(VOUT) trace.
 - 2) Add a new plot
 - 3) Add back in the V(VOUT) plot using Add Trace.
 - 4) Change the Y-axis settings of the V(VOUT) plot to log-scale.
 - h. Print-out Probe Plot using "screen-shot" method.
 - l. Print-out circuit schematic using "screen-shot" method.
 - o. Logout
4. Hand-in:
 - a) Circuit schematic print-out (annotated)
 - b) Probe plot print-out (annotated)
 - c) xxx.out file print-out (edited and annotated)