PSPICE Tutorial

Dr. Peter Osterberg

1. Login to a PC and go to your P-drive. Create an EE352 folder. Within the EE352 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)