

## Design:

When you design a circuit, the circuit should first be designed, and then validated. Pspice is a validation tool. In a design you should first show how you calculated all of the values that you found. Then, calculate numerically all needed specifications, for example voltage gain, input impedance, etc. (what you are designing for). Then verify your calculation, i.e. PSPICE. You will run a transient analysis showing the input waveform, and also the output waveform. This will show that your circuit is not saturating and operating correctly. Then run an AC sweep to see the frequency characteristics of the designed circuit. Use the markers to mark the voltage gain, and corner frequencies. Use  $\text{dB}(V_{\text{out}}/V_{\text{in}})$ . This will normalize the frequency plot, and show you then true voltage gain of the circuit. If you do not do this and just show the output voltage it is not in reference to anything, remember voltage gain is  $V_{\text{out}} / V_{\text{in}}$ . If you use  $\text{dB}(V_{\text{out}}/V_{\text{in}})$  it is easy to calculate the corner frequencies (mark them), bandwidth, voltage gain etc. Also when you run an AC sweep you can calculate the input impedance, which is  $V_s / I_s$ . Input Voltage / Input Current.

Now, that you ran your simulation and verified that your design was correct, print out all the above plots, schematics and calculations to prove to your professor or boss that your circuit indeed works.

NOTE: it is important to calculate the designed specifications on paper first because then you will have information to verify, and if you measure something wrong in Pspice or the lab you will know it because you have an idea of what the value should be.