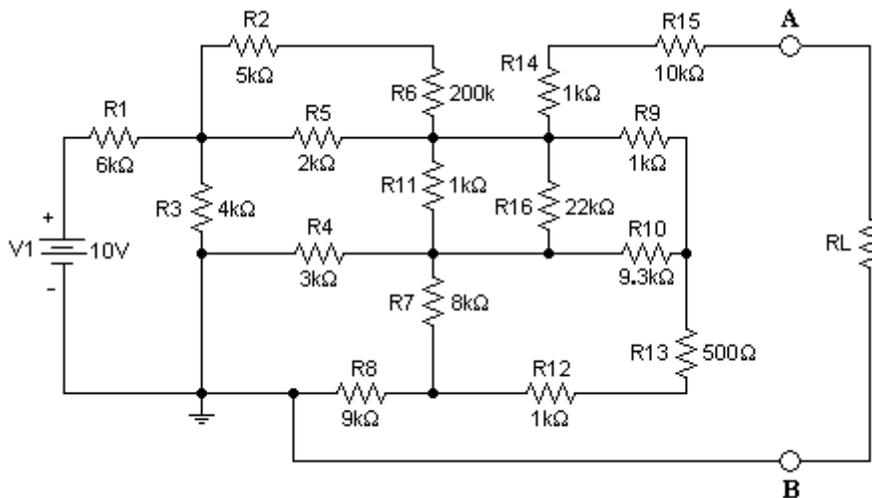


EE 206 Simulation 5

Maximum Power Transfer

1. Use PSpice schematics to draw the circuit given below. With R_L removed, use PSpice to find the Thévenin equivalent voltage, V_{TH} , and the Thévenin's equivalent resistance, R_{TH} . (To obtain the latter value, use PSpice to determine the short circuit current, I_{sc} , and then calculate R_{TH} using the standard equivalent circuit relationship.)
2. Use PSpice to find the maximum power consumed by a load resistor R_{LOAD} by plotting power vs. R_{LOAD} for various values of the load resistor. Vary the value of R_{LOAD} from $7.0\text{ k}\Omega$ to $20.0\text{ k}\Omega$ with a $200\ \Omega$ increment. [Hint - Set the value of your load resistor to a variable $\{RL\}$, and add part PARAM from the *special library* of PSpice. Set the value of parameter RL to $7.0\text{ k}\Omega$. Use global parameter sweep to vary the value of RL . Also, use DC sweep for a single step for $V1$ (DC 10V 10V 1V). Use the Probe feature of PSpice to plot power consumed by R_{LOAD} as it is varied through the RL values.]
3. Find the maximum power consumed by R_{LOAD} by plotting power vs. R_{LOAD} using MATLAB (via Thévenin values from step 1). Vary the value of R_{LOAD} from $7.0\text{ k}\Omega$ to $20.0\text{ k}\Omega$ with a $200\ \Omega$ increment.
4. Discuss your results. What can you conclude from the results?
5. Describe an alternate approach (different than that used in step 1.) for using PSpice Schematics to determine R_{TH} and V_{TH} .



Your simulation report should include the PSpice schematics you used in both steps 1 and 2 along with the PSpice results and plot, the programs and plot from MATLAB, and the discussion from both steps 4 and 5.

Your report is due at the beginning of your next recitation period.