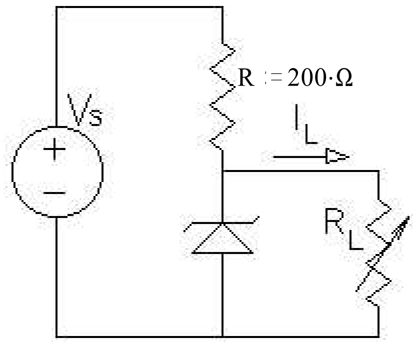
ECE 2100 Homework # 12 & Spice Project # S1

Homework 12 Due: Mon, 2/24/03

Ex3.24 - 26 on p178

Problem 3.75



5. Consider the shunt regulator at right.
   The diode has the following characteristics:

   $V_Z := 6.80\cdot V$ at $I_Z := 10\cdot mA$

   $r_Z := 20\cdot\Omega$

   $I_{Zk} := 4\cdot mA$ below which the voltage is no longer considered "regulated".

   a) (10 pts) With the load removed ($R_L=\infty$),
   What is the smallest value of V_S for which the output remains regulated?

   b) (10 pts) $V_S := 10\cdot V$ What is the smallest value of R_L for which the output remains regulated?

   Answers: 7.48V, 530Ω

Ex3.6 - 9

ECE 2100 Spice project # S1 **Due Wed, 3/5/03**

1. Find a computer wih a version of spice that you can use to work this problem. PSpice is available for download off the Internet at http://www.cadencepcb.com/products/downloads/PSpicestudent/default.asp, (link on class web page). It is also available on the computers in the lab. HSPICE should be available on computers in the CADE lab (I have not checked this). I'd be interested to hear of any versions of SPICE for Linux.

   CDs with the student version of PSPICE 9.2 may soon be available at the lab parts counter for a small copying charge.

   Specific help and instructions for PSpice 9.1 & 9.2 are available on the web.

2. Exercise 3.33, p.204
   I want:
   1. Transient signal graphs, just like the bottom of Fig. E3.33. For this you will repeat what the author did.
   2. A drawing, showing all the parts and nodes, like the top of Fig. E3.33, but add a 1 kΩ load resistor.
   3. A Spice input list like that shown in Fig. D2, p.D2 in appendix D, but again, add the load resistor.
   4. Two more sets of transient signal graphs, like the bottom of Fig. E3.33. One for R_L = 1 kΩ and one for R_L = 10 kΩ.

   Notes:
   Check out appendix C and appendix D in your textbook, they will help.

3. Compare SPICE results to Doubler lab results.

   When you did The Doubler lab, you were instructed to make the same circuits in lab that you ran here in SPICE. You should have sketches of the output waveform for all three cases, showing the average DC value as well as the ripple.

   Compare the 3 measured DC voltages to those calculated by SPICE.

   Compare the measured ripple for the two loaded cases to those calculated by SPICE. Compare both shape and voltage values.