13. Print this page out. On that page, write just the SPICE netlist for the schematic below in the shaded area. Using the template file given on the website, enter your netlist into a file named "probXX.sp", where XX is the problem number.

Run the simulator and redirect the output into another file:
ngspice prob<XX>.sp > prob<XX>.sp.results

Print out the results file by typing:
a2ps -P <printer_name> probXX.sp.results

and paste the contents of the results file onto this page also. You must include the header timestamp.

Find $V_A, V_B, V_C, V_D, I_X$

to find $I_X$, find current through $R_2$ by using the command "show $R_2" inside the .control .endc block.

Result file printed from a2ps:

```
Circuit: - KVL prob 12
Doing analysis at TEMP = 27.000000 and TNOM = 27.000000
No. of Data Rows : 1
Resistor: Simple linear resistor
device    r2
    model    R
    resistance  8000
    ac        8000
    dtemp     0
    noisy     1
    l    0.000266667
    p    0.000568889
```

Your Netlist:

```
V1  A  12
r1  A  B  2k00
V2  C  B  -8
r3  C  D  5k00
r2  D  sw  8k
```