EXAMPLE

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.

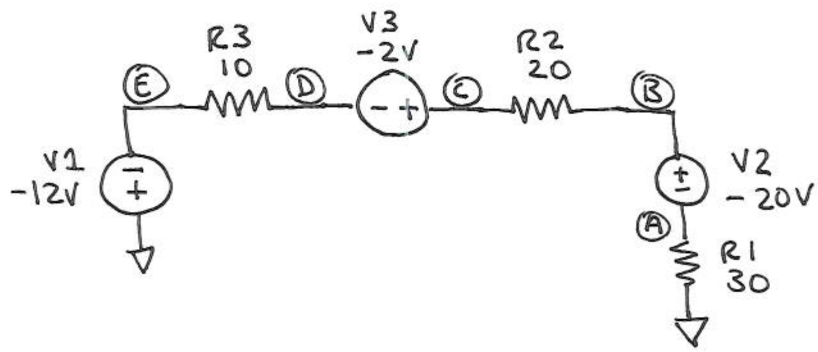
Run the simulator and redirect the output into another file:

```
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.



Result file printed from a2ps:

Jan 27, 11 22:45	**probXX.sp.results**	Page 1/1

```
******
** ngspice-20 : Circuit level simulation program
** The U. C. Berkeley CAD Group
** Copyright 1985-1994, Regents of the University of California.
** Please submit bug-reports to: ngspice-bugs@lists.sourceforge.net
** Creation Date: Tue May  4 20:40:59 PDT 2010
******

Circuit: .title spice example

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000


No. of Data Rows : 1

node voltages:
v(a) = 1.50e+01
v(b) = -5.0e+00
v(c) = 5.00e+00
v(d) = 7.00e+00
v(e) = 1.20e+01

ngspice-20 done
```

Your Netlist:

```
V1 gnd e  -12
R1 gnd a   30
R2  b  c   20
R3  e  d   10
V2  b  a  -20
V3  c  d   -2
```