Spice HW Example

Print this page out. On that page, write just the SPICE netlist for the schmatic shown in figure **??** 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 tedirect the output into another file:

```
ngspice prob<XX>.sp > prob<XX>.sp.results
```

Print out the results file by typing:

```
a2ps -P <printer_name> prob<XX>.sp.results
```

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

Figure 1: Spice problem 1.

Result file printed from a2ps:

```
Circuit: example problem

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

No. of Data Rows : 1
node voltages:
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
```

Your netlist:

```
.title netlist
v1      gnd     e       -12
rl      gnd     a       30
r2      b       c       20
r3      e       d       10
v2      b       a       -20
v3      c       d       -2
```