1. 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:
```bash
ngspice prob<XX>.sp > prob<XX>.sp.results
```

Print out the results file by typing:
```bash
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 $V_A$, $V_B$

$V_A = 60\text{OV}$
$V_B = 4\text{V}$

![Schematic Diagram]

Result file printed from a2ps:

```
---
** ngspice-2iplus : 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: Thu Jul 22 22:02:18 UTC 2010
---

Circuit: ngspice template file for ECE112

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

No. of Data Rows : 1
node voltages:
$v(a) = 6.00e+00$
$v(b) = 6.00e+00$
$v(c) = 3.00e+00$

:ngspice-2iplus done
```

Your Netlist:

```
I1  GND  A  9
R1  A    GND 8
R2  A    B  16
I2  B    A  2
R3  B    GND 4
R4  B    C  2
V1  GND  C  -3
```