

**EECE 311  
Fall 2011**

**Lab Report #2  
Using SPICE for Nodal Circuit Analysis  
9/13/2011**

Submitted by:

Signature

Printed Name

Date

	<b>Jeremiah Mahler</b>	<b>Sep 20, 2011</b>
--	------------------------	---------------------

## **1 Objective**

The objective of this lab is to gain experience performing detailed circuit analysis of a circuit with a dependent source by using both SPICE and manual calculations.

## **2 Equipment**

To perform the circuit simulation a SPICE[1] simulator was used. Specifically, Ngspice[3] was used. But other programs such as Orcad[2] should work as well.

## **3 Procedure**

The procedure for this experiment involves two major steps.

1. Build a SPICE definition of the circuit in Figure 1.
2. Run the simulation and record the output.

At a minimum all the node voltages should be included in the output.

The simulation can be run using Ngspice with the command:

```
ngspice -b your_file.cir
```

with `your_file.cir` replaced by the name of your file containing the SPICE definition. To save the output to a file a redirect can be used as in:

```
ngspice -b your_file.cir > your_file.out
```

If Orcad is being used the same can be accomplished through its GUI interface.

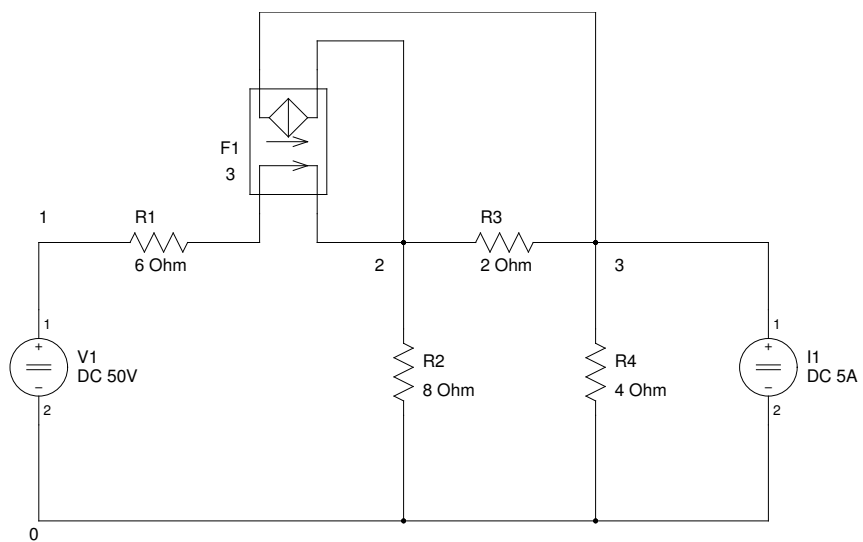


Figure 1: Circuit definition. Nodes denoted by numbers with 0 as common. F1 is a current dependent current source, I1 is a current source, and V1 is a voltage source.

```

EECE 311 Laboratory Project #2
* run using: ngspice -b this_file.cir

* + -
V1 1 0 DC 50

R1 1 10 6
VSENS 10 2 DC 0

* current controlled current source
* - +
F1 3 2 VSENS 3

R2 0 2 8
R3 2 3 2
R4 0 3 4

* current, in -, out +
* - +
I1 0 3 DC 5

.DC V1 50 50 1

.PRINT DC V(1), V(2), V(3)

.END

```

Figure 2: SPICE definition of circuit in Figure 1.

## 4 Results

The output of the simulation is shown in Figure 3. Most of the output is extra information about how long the process took to run and other things that are of no concern in this lab. The important values are the voltages ( $v(1)$ ,  $v(2)$ , and  $v(3)$ ).

Circuit: eece 311 laboratory project #2

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

No. of Data Rows : 1

eece 311 laboratory project #2				
DC transfer characteristic Mon Sep 19 16:30:48 2011				
Index	v-sweep	v(1)	v(2)	v(3)
0	5.000000e+01	5.000000e+01	3.200000e+01	1.600000e+01

CPU time since last call: 0.008 seconds.

Total CPU time: 0.008 seconds.

Total DRAM available = 992.695312 MB.

DRAM currently available = 79.210938 MB.

Total ngspice program size = 5.745117 MB.

Resident set size = 658.000 kB.

Shared ngspice pages = 521.000 kB.

Text (code) pages = 1.553711 MB.

Stack = 0 bytes.

Library pages = 3.041992 MB.

Figure 3: Output from SPICE simulation.

## 5 Correlation with theory

To correlate the simulation with theory the manual calculations are performed here and then they are analyzed at the end of this section.

First the node voltage equations <sup>1</sup> are created.

$$\frac{v_1 - v_2}{6} = i_1 \quad (1)$$

$$v_1 = 50 \quad (2)$$

$$\frac{v_1 - v_2}{6} + \frac{-v_2}{8} + \frac{v_3 - v_2}{2} + 3 \cdot i_1 = 0 \quad (3)$$

$$\frac{v_2 - v_3}{2} + \frac{-v_3}{4} + 5 - 3 \cdot i_1 = 0 \quad (4)$$

Simplifying and substituting results in the following set of three equations with three unknown variables.

$$\frac{1}{2}v_3 - \frac{19}{24}v_2 + 3i_1 = -\frac{50}{6} \quad (5)$$

$$-\frac{3}{4}v_3 + \frac{1}{2}v_2 - 3i_1 = -5 \quad (6)$$

$$0v_3 - \frac{1}{6}v_2 - i_1 = -\frac{50}{6} \quad (7)$$

Solving this set of equations results in the following solutions.

$$v_2 = 32 \quad [\text{volts}] \quad (8)$$

$$v_3 = 16 \quad [\text{volts}] \quad (9)$$

$$i_1 = 3 \quad [\text{amps}] \quad (10)$$

From the theoretical calculations  $v_2 = 32$  volts (Equation 8) and  $v_3 = 16$  volts (Equation 9). From the SPICE simulation (Figure 3)  $v_2 = 32.0$  volts and  $v_3 = 16.0$  volts. These values are identical indicating that the theory corresponds exactly with the simulation.

## 6 Conclusion

This experiment was a complete success in performing a detailed circuit analysis of a circuit with a dependent source by using both SPICE and manual calculations. The calculations matched the theoretical values exactly with no measurable amount of error.

---

<sup>1</sup>For a description of the Node Voltage Method see [4, Pg. 97]

## 7 References

- [1] Wikipedia, “Spice — Wikipedia, the free encyclopedia.”  
<http://en.wikipedia.org/wiki/SPICE>, 2011. [Online; accessed 19-September-2011].
- [2] “Orcad, electronic design automation software.”  
<http://www.cadence.com/orcad/>, 2011. Cadence Design Systems.
- [3] Ngspice, “Ngspice, spice circuit simulator.”  
<http://ngspice.sourceforge.net>, 2011.
- [4] J. Nilsson and S. Riedel, Electric circuits. Pearson/Prentice Hall, 2008.