EECE 311
Fall 2011

Lab Report #3
Using SPICE for Thévenin Circuit Analysis
9/20/2011

Submitted by:

Signature	Printed Name	Date
	Jeremiah Mahler	**Oct 4, 2011**

1 Objective

The objective of this lab is to gain experience performing detailed circuit analysis of a Thévenin Equivalant [4, Pg. 119] circuit using SPICE.

2 Equipment

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

For this Thévenin analysis the open circuit voltage, equivalent resistance and short circuit current will be found.

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.

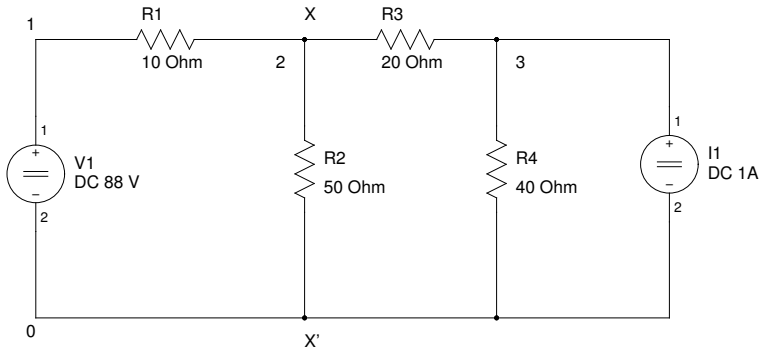

Figure 1: Circuit definition. Nodes denoted by numbers with 0 as common. V1 is a voltage source, I1 is a current source. The Thévenin Equivalent is calculated with respect to X and X' where X is positive.

```
EECE 311 Lab 3

V1 1 0 DC 88
R1 1 2 10
R2 2 0 50
R3 2 3 20
R4 3 0 40
* Use VSC for short circuit current
*VSC 2 0 DC 0
I1 0 3 DC 1

* Thevenin resistance
*.TF V(2,0) V1

* Open Circuit voltage
.DC V1 88 88 1
.PRINT DC V(2)

* short circuit current
*.DC V1 88 88 1
*.PRINT DC I(VSC)

.END
```

Figure 2: SPICE definition of circuit in Figure 1.

4 Results

The output of the simulation is shown in Figure 3, 4 and 5. 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 open circuit voltage, short circuit current and Thévenin resistance.

```
Circuit: eece 311 lab 3

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000


No. of Data Rows : 1

                                eece 311 lab 3
                                DC transfer characteristic  Tue Sep 20 09:55:21  2011
--------------------------------------------------------------------------------
Index   v-sweep         vsc#branch
--------------------------------------------------------------------------------
0    8.800000e+01    9.466667e+00

CPU time since last call: 0.004 seconds.

Total CPU time: 0.004 seconds.

Total DRAM available = 992.695312 MB.
DRAM currently available = 256.492188 MB.
Total ngspice program size = 5.745117 MB.
Resident set size = 654.000 kB.
Shared ngspice pages = 517.000 kB.
Text (code) pages = 1.553711 MB.
Stack = 0 bytes.
Library pages = 3.041992 MB.
```

Figure 3: Output from SPICE simulation for short circuit current.

```
Circuit: eece 311 lab 3

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000


No. of Data Rows : 1

Transfer function information:
transfer_function = 7.317073e-01
output_impedance_at_v(2,0) = 7.317073e+00
v1#input_impedance = 3.727273e+01

CPU time since last call: 0.004 seconds.

Total CPU time: 0.004 seconds.

Total DRAM available = 992.695312 MB.
DRAM currently available = 257.875000 MB.
Total ngspice program size = 5.744141 MB.
Resident set size = 661.000 kB.
Shared ngspice pages = 525.000 kB.
Text (code) pages = 1.553711 MB.
Stack = 0 bytes.
Library pages = 3.041016 MB.
```

Figure 4: Output from SPICE simulation for Thévenin equivalent resistance ("output_impedance").

```
Circuit: eece 311 lab 3

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000


No. of Data Rows : 1

                                eece 311 lab 3
                                DC transfer characteristic  Tue Sep 20 10:52:55  2011
--------------------------------------------------------------------------------
Index   v-sweep         v(2)
--------------------------------------------------------------------------------
0    8.800000e+01    6.926829e+01

CPU time since last call: 0.008 seconds.

Total CPU time: 0.008 seconds.

Total DRAM available = 992.695312 MB.
DRAM currently available = 188.226562 MB.
Total ngspice program size = 5.744141 MB.
Resident set size = 654.000 kB.
Shared ngspice pages = 517.000 kB.
Text (code) pages = 1.553711 MB.
Stack = 0 bytes.
Library pages = 3.041016 MB.
```

Figure 5: Output from SPICE simulation for open circuit voltage.

5 Correlation with theory

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

The first step in this analysis is to simplify the circuit using source transformation. The result is show in Figure 6.

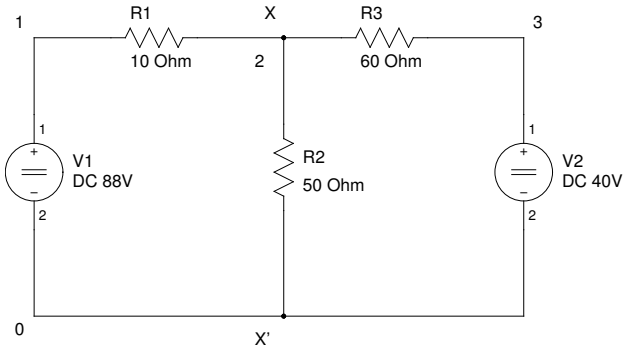

Figure 6: Simplification of the circuit (Figure 1) by using source transformation.

Then Node Voltage Analysis can be used to find the open circuit voltage (Thévenin Equivalent voltage). Since v_1 and v_3 are constant (Figure 6) we only have one equation with one unknown.

$$\frac{88 - v_2}{10} + \frac{40 - v_1}{60} + \frac{-v_1}{50} = 0 \tag{1}$$

Simplifying and substituting results in a Thévenin voltage of:

$$V_{\text{th}} = v_2 \tag{2}$$

$$= 69.268 \quad [\text{volts}] \tag{3}$$

The Thévenin equivalent resistance can be found by finding the resistance when both voltage sources are disabled (0 volts). This is equivalent to all the resistors in parallel.

$$R_{\text{th}} = \frac{1}{1/10 + 1/50 + 1/60} \tag{4}$$

$$= 7.317 \quad [\text{ohms}] \tag{5}$$

The Thévenin equivalent voltage and resistance found in these calculations were 69.268 volts and 7.317 ohms respectively. The Thévenin equivalent voltage and resistance found from the SPICE simulation were 69.268 volts (Figure 5) and 7.317 ohms (Figure 4) respectively. 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 Thévenin Equivalent circuit using SPICE. The calculations matched the theoretical values exactly with no measurable amount of error.

7 References

[1] Wikipedia, "Spice — Wikipedia, the free encyclopedia." `http://en.wikipedia.org/wiki/SPICE`, 2011. [Online; accessed 19-September-2011].

[2] Ngspice, "Ngspice, spice circuit simulator." `http://ngspice.sourceforge.net`, 2011.

[3] "Orcad, electronic design automation software." `http://www.cadence.com/orcad/`, 2011. Cadence Design Systems.

[4] J. Nilsson and S. Riedel, Electric circuits. Pearson/Prentice Hall, 2008.