

EECE 311
Fall 2011

Lab Report #1
Using SPICE for Electric Circuit Analysis
9/6/2011

Submitted by:

Signature

Printed Name

Date

	Jeremiah Mahler	Sep 13, 2011
--	-----------------	--------------

1 Description/Objectives

The objective of this lab is to gain experience simulating and analyzing circuits using SPICE simulations.

2 Procedure

In general the procedure involves creating a SPICE definition of the circuit and then analyzing the results to produce raw values or plots. Shown in Figure 1 is the graphical representation of the circuit that will be used in this lab.

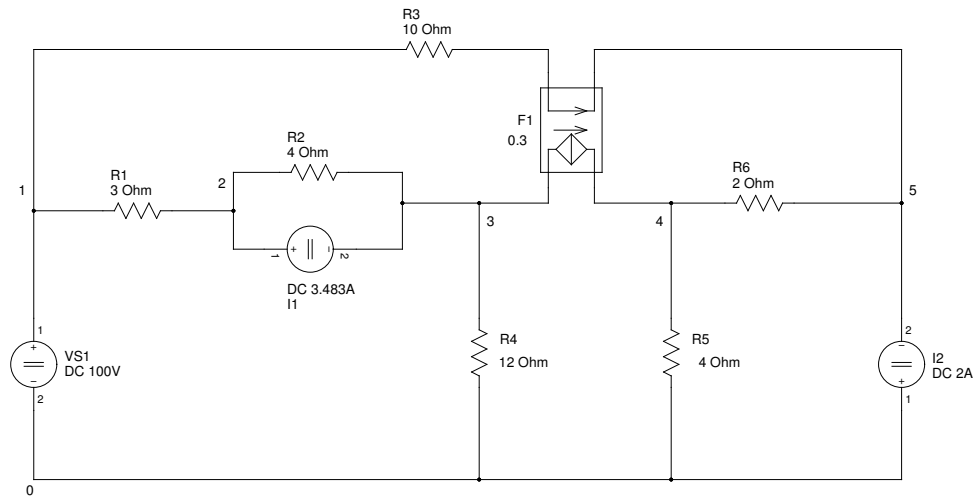


Figure 1: Circuit used for SPICE simulation. Nodes denoted by numbers with 0 as common.

The SPICE definition of the circuit is given in Appendix A. To run the simulation there are various options depending on the platform being used.

Under Windows, the closed source student version of the Orcad [1] simulator can be used. It provides a graphical user interface its use is self explanatory. Under Linux and Windows the open source Ngspice [2] simulator can be used. Shown in Figure 2 are the commands used under Linux to produce all the text based data values and plots of the simulation. Figures 3, 4, 5, and 6 show the textual output produced from running the SPICE simulation.

```
shell$ # ngspice -b <name of spice file>
shell$ ngspice -b lab1.cir
(output sent to screen)
shell$ ngspice -b lab1.cir > lab1.out
(output save to file lab1.out)
```

Figure 2: Shell commands for running the SPICE simulation in batch mode (-b) using Ngspice under Linux.

The SPICE simulation can also be used to produce graphical plot output. Using Ngspice the simulation must be re-configured to run in interactive mode instead of batch mode so that the plot directives will work (see the SPICE definition in Appendix A for more information). These plots can be displayed on the terminal or output to a file in various format (jpg, png, eps, pdf) for inclusion in to a document. Shown in Figure 7 is the graphical output of this simulation when plotted using Gnuplot [3].

eece 311 laboratory project #1			
DC transfer characteristic Sun Sep 11 11:14:25 2011			
Index	v-sweep	vs1#branch	vsens#branch
0	0.000000e+00	9.660220e-02	-6.97674e-01
1	5.000000e+00	-5.12333e-01	-4.06977e-01
2	1.000000e+01	-1.12127e+00	-1.16279e-01
3	1.500000e+01	-1.73020e+00	1.744186e-01
4	2.000000e+01	-2.33914e+00	4.651163e-01
5	2.500000e+01	-2.94807e+00	7.558140e-01
6	3.000000e+01	-3.55701e+00	1.046512e+00
7	3.500000e+01	-4.16594e+00	1.337209e+00
8	4.000000e+01	-4.77488e+00	1.627907e+00
9	4.500000e+01	-5.38381e+00	1.918605e+00
10	5.000000e+01	-5.99275e+00	2.209302e+00
11	5.500000e+01	-6.60168e+00	2.500000e+00
12	6.000000e+01	-7.21062e+00	2.790698e+00
13	6.500000e+01	-7.81955e+00	3.081395e+00
14	7.000000e+01	-8.42849e+00	3.372093e+00
15	7.500000e+01	-9.03742e+00	3.662791e+00
16	8.000000e+01	-9.64636e+00	3.953488e+00
17	8.500000e+01	-1.02553e+01	4.244186e+00
18	9.000000e+01	-1.08642e+01	4.534884e+00
19	9.500000e+01	-1.14732e+01	4.825581e+00
20	1.000000e+02	-1.20821e+01	5.116279e+00

Figure 3: Tabular current outputs resulting from voltage sweep of VS1.

eece 311 laboratory project #1
DC transfer characteristic Sun Sep 11 11:14:25 2011

Legend: + = vs1#branch
* = vsens#branch

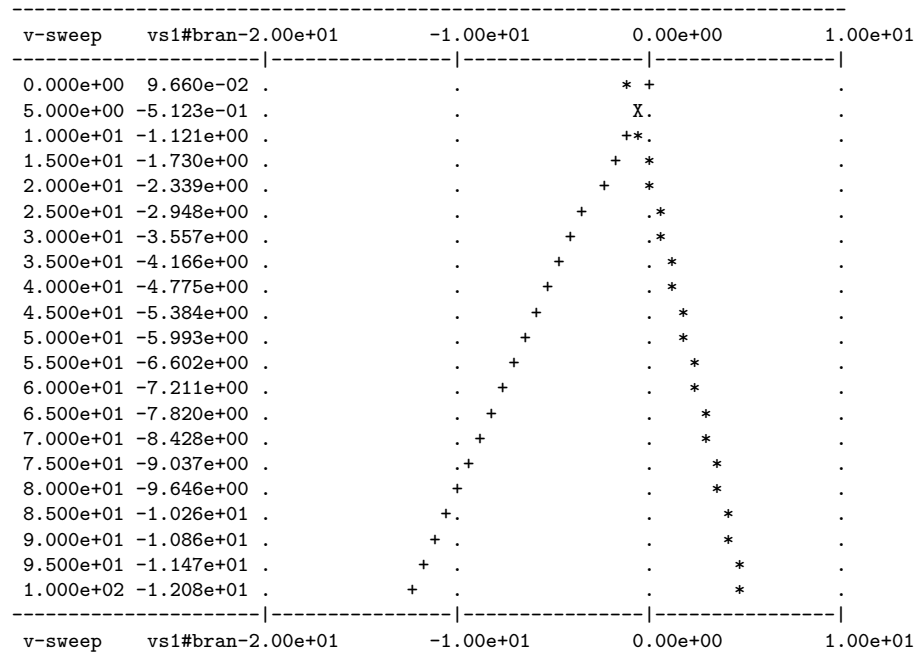


Figure 4: Textual plot of current outputs resulting from voltage sweep of VS1.

eece 311 laboratory project #1				
DC transfer characteristic Sun Sep 11 11:14:25 2011				
Index	v-sweep	v(3)	v(4)	v(5)
0	0.000000e+00	9.724494e+00	4.372093e+00	6.976744e+00
1	5.000000e+00	1.249683e+01	5.883721e+00	9.069767e+00
2	1.000000e+01	1.526917e+01	7.395349e+00	1.116279e+01
3	1.500000e+01	1.804151e+01	8.906977e+00	1.325581e+01
4	2.000000e+01	2.081385e+01	1.041860e+01	1.534884e+01
5	2.500000e+01	2.358618e+01	1.193023e+01	1.744186e+01
6	3.000000e+01	2.635852e+01	1.344186e+01	1.953488e+01
7	3.500000e+01	2.913086e+01	1.495349e+01	2.162791e+01
8	4.000000e+01	3.190320e+01	1.646512e+01	2.372093e+01
9	4.500000e+01	3.467553e+01	1.797674e+01	2.581395e+01
10	5.000000e+01	3.744787e+01	1.948837e+01	2.790698e+01
11	5.500000e+01	4.022021e+01	2.100000e+01	3.000000e+01
12	6.000000e+01	4.299255e+01	2.251163e+01	3.209302e+01
13	6.500000e+01	4.576489e+01	2.402326e+01	3.418605e+01
14	7.000000e+01	4.853722e+01	2.553488e+01	3.627907e+01
15	7.500000e+01	5.130956e+01	2.704651e+01	3.837209e+01
16	8.000000e+01	5.408190e+01	2.855814e+01	4.046512e+01
17	8.500000e+01	5.685424e+01	3.006977e+01	4.255814e+01
18	9.000000e+01	5.962658e+01	3.158140e+01	4.465116e+01
19	9.500000e+01	6.239891e+01	3.309302e+01	4.674419e+01
20	1.000000e+02	6.517125e+01	3.460465e+01	4.883721e+01

Figure 5: Tabular voltage outputs resulting from voltage sweep of VS1.

eece 311 laboratory project #1
DC transfer characteristic Sun Sep 11 11:14:25 2011

Legend: + = v(1)
* = v(3)
= = v(4)
\$ = v(5)

v-sweep	v(1)	0.00e+00	2.00e+03	4.00e+03	6.00e+03	8.00e+03	1.00e+04
0.000e+00	0.000e+00	+=\$*
5.000e+00	5.000e+00	.X\$*
1.000e+01	1.000e+01	.=X*
1.500e+01	1.500e+01	.=\$+*
2.000e+01	2.000e+01	.=\$X
2.500e+01	2.500e+01	.=\$.*+
3.000e+01	3.000e+01	.=\$.*+
3.500e+01	3.500e+01	.=\$.*+
4.000e+01	4.000e+01	.=\$.*+
4.500e+01	4.500e+01	.=\$.*+
5.000e+01	5.000e+01	.=\$.*+
5.500e+01	5.500e+01	.=\$.*+
6.000e+01	6.000e+01	.=\$.*+
6.500e+01	6.500e+01	.=\$.*+
7.000e+01	7.000e+01	.=\$.*+
7.500e+01	7.500e+01	.=\$.*+
8.000e+01	8.000e+01	.=\$.*+
8.500e+01	8.500e+01	.=\$.*+
9.000e+01	9.000e+01	.=\$.*+
9.500e+01	9.500e+01	.=\$.*+
1.000e+02	1.000e+02	.=\$.*+
v-sweep	v(1)	0.00e+00	2.00e+03	4.00e+03	6.00e+03	8.00e+03	1.00e+04

Figure 6: Textual plot of voltage outputs resulting from voltage sweep of VS1.

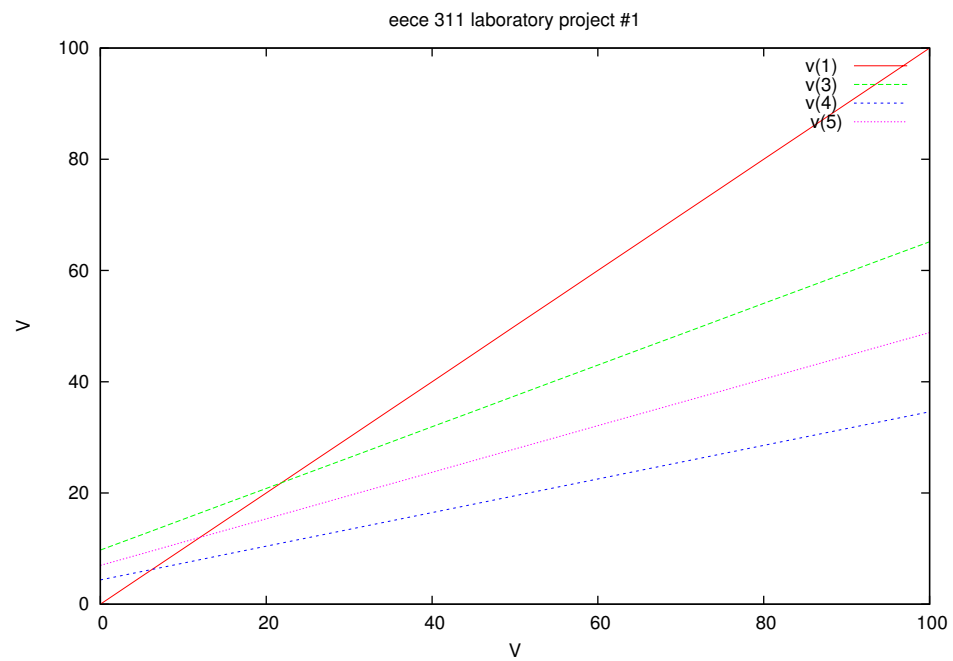


Figure 7: Graphical plot of SPICE simulation.

3 Observations

The various ways of displaying data should corroborate the expected and calculated values. It can be seen by comparing the textual plot in Figure 6 and the graphical plot in Figure 7 that the results are the same other than the fact that the vertical axis is inverted. The raw data can also be used to view the data in more detail than what is available with the plots. The textual plots are the least precise and have the largest amount of error compared to graphical plots.

4 Conclusion

This lab was a success in gaining experience with the various ways to define and analyze circuits using SPICE. A single SPICE circuit definition can be used to produce raw data values, textual plots or full graphical plots suitable for inclusion in to a document.

5 References

- [1] “Orcad, electronic design automation software.”
<http://www.cadence.com/orcad/>, 2011. Cadence Design Systems.
- [2] Ngspice, “Ngspice, spice circuit simulator.”
<http://ngspice.sourceforge.net>, 2011.
- [3] T. Williams and C. Kelley, “Gnuplot.” <http://www.gnuplot.info>, 2011.

A SPICE definition

```
EECE 311 Laboratory Project #1
* run using: ngspice -b lab1.cir
* http://www.ecst.csuchico.edu/~hma/Lab1.311.pdf

VS1 1 0 DC 100V
R1 1 2 3
R2 2 3 4
I1 2 3 DC 3.483

R3 1 6 10
VSENS 6 5 DC 0V
F1 3 4 VSENS 0.3

R4 3 0 12
R5 4 0 4
R6 4 5 2
I2 0 5 DC 2

* Uncomment the section for the mode your are using.
*****> -b (batch mode) <*****
** ngspice -b <this file>

* sweep VS1 from 0 volts to 100 volts in 5 volt increments
*.DC VS1 0V 100V 5V

*.PRINT DC I(VS1) I(VSENS)
*.PLOT DC I(VS1) I(VSENS)
*.PRINT DC I(R1) I(R2) I(R3) I(R4) I(R5) I(R6)
*.PRINT DC V(3) V(4) V(5)
*.PLOT DC V(1) V(3) V(4) V(5)
*.PROBE
*.END
*****> interactiv mode <*****
** ngspice <this file>

.CONTROL

DC VS1 0V 100V 5V

* gui plot
*PLOT V(1) V(2) V(3) V(4) V(5)
```

```
GNUPLOT lab1-plot V(1) V(3) V(4) V(5)
* gnuplot -persist lab1-plot.plt
* outputs to lab1-plot.eps

.ENDC
.END
*****
* vim:syntax=spice
```