

NGspice And geda Schematic

Shelcia Carolyn Samuvel

April 20, 2020

Contents

1	Introduction	1
2	Installation	1
3	Schematic	2
4	Generating the netlist	3
5	Plot	4
6	Result	5

1 Introduction

This document is to show how to use **NGspice** and **geda** to analyze circuits and produce outputs.

2 Installation

Lets install both the required software in the system before we proceed. Since the project has been done on a Linux system, It will be assumed that the user is operating a Linux OS, preferably Ubuntu.

Type the following commands in the terminal to install the packages:

```
sudo apt install ngspice  
sudo apt install geda
```

3 Schematic

The first step is to prepare a working schematic. To achieve this open **gschem**. Draw the schematic with the required components and the values as shown in figure.

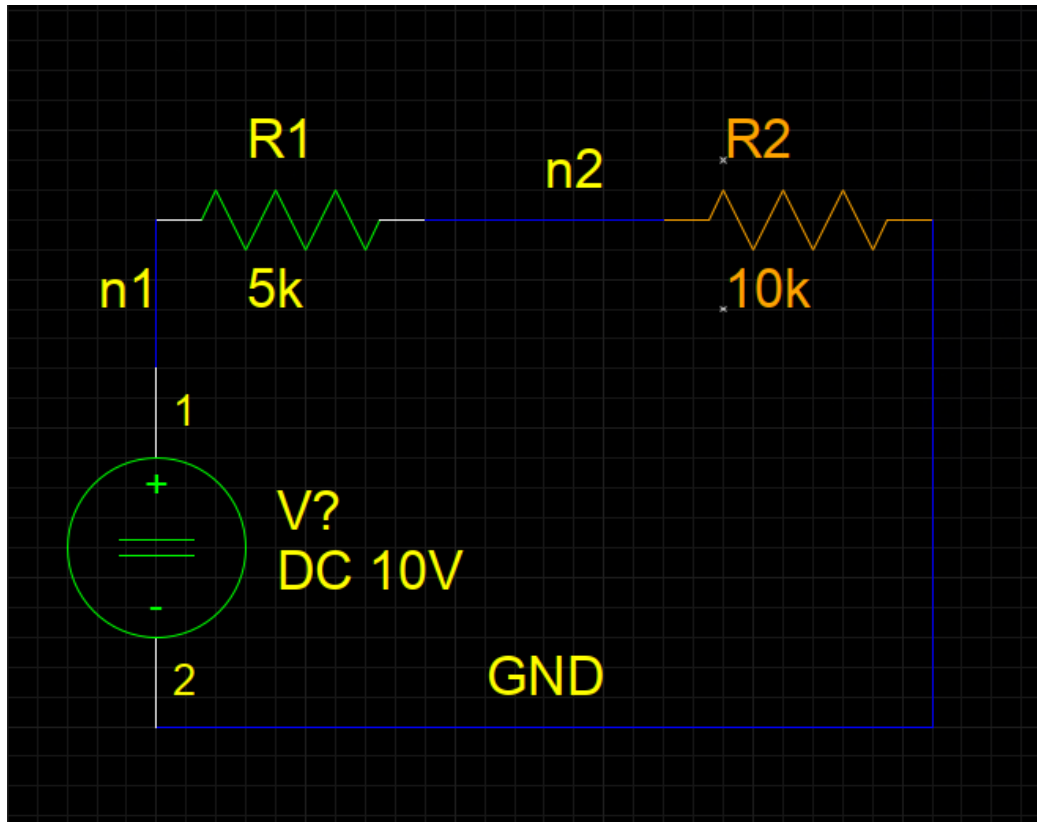


Figure 1: Schematic

4 Generating the netlist

The commands in the terminal are shown in the image below:

```
nazrul_being@nazrul:~/Documents/Spice_Works$ ls
01.sch  01.sch~
nazrul_being@nazrul:~/Documents/Spice_Works$ gnetlist -g spice -o 01.net 01.sch
Loading schematic [/home/nazrul_being/Documents/Spice_Works/01.sch]
nazrul_being@nazrul:~/Documents/Spice_Works$ cat 01.net
* Spice netlister for gnetlist
V? n1 0 DC 10V
R2 n2 0 10k
R1 n1 n2 5k
.END
nazrul_being@nazrul:~/Documents/Spice_Works$ ngspice
*****
** ngspice-31 : Circuit level simulation program
** The U. C. Berkeley CAD Group
** Copyright 1985-1994, Regents of the University of California.
** Please get your ngspice manual from http://ngspice.sourceforge.net/docs.html
** Please file your bug-reports at http://ngspice.sourceforge.net/bugrep.html
** Creation Date: Wed Oct  9 16:01:32 UTC 2019
*****
ngspice 1 -> source 01.net

Circuit: * spice netlister for gnetlist

ngspice 1 -> tran 0.01ms 5ms
Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

Initial Transient Solution
-----

Node                                Voltage
----                                -
n1                                  10
n2                                  6.66667
v?#branch                          -0.000666667

No. of Data Rows : 508
ngspice 1 -> plot n2, n1
ngspice 1 -> □
```

Figure 2: Generating the netlist

Under the part of **Initial Transient Solution** we have obtained some values. We plot the graph using this values to see what is happening with the circuit.

5 Plot

After performing all commands, as described in the previous section (Generating the netlist) we plot the graph. We can see we obtain a graph similar to this.

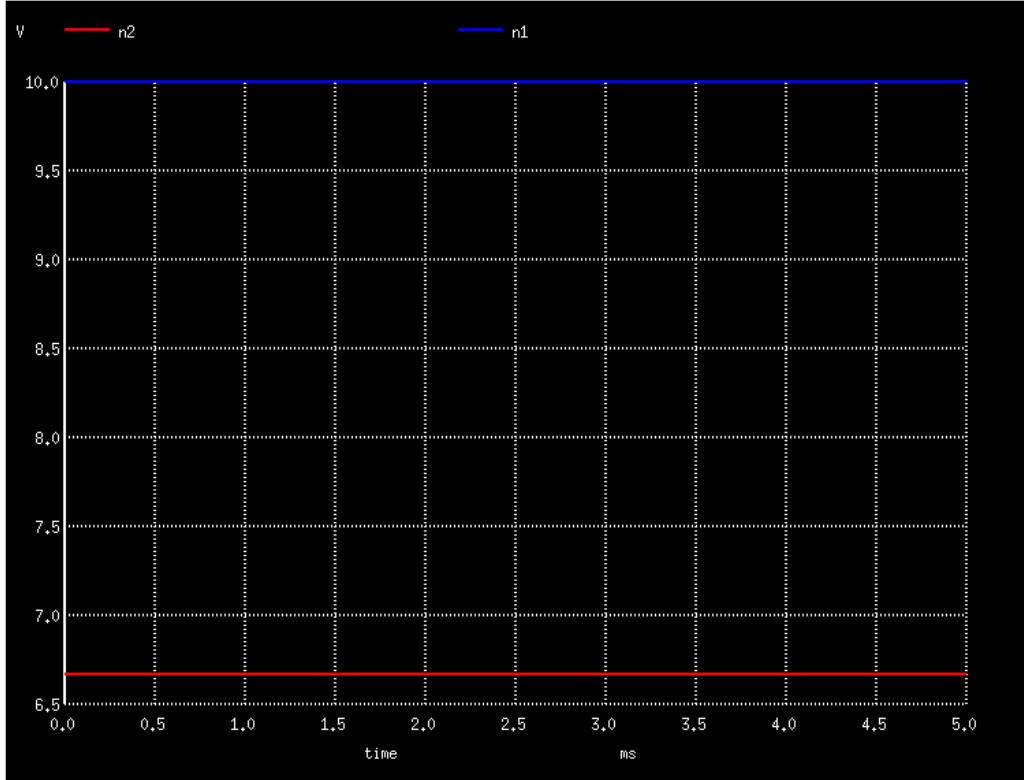


Figure 3: Plot

6 Result

This method of analyzing circuits is much easier and accurate. As we can see that the circuit is a simple voltage divider. Using this form of analysis gives us much better understanding of circuits and at the same time allows us to solve the circuits with much less effort and accurately.

Thank You!