

# Working With Ngspice And Geda Schematic

Nazrul

April 1, 2020

## Contents

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

# 1 Introduction

The purpose of this document is to show how to use ngspice and geda in parallel to analyze circuits and produce desired 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 in this command and follow along the terminal to install the packages.

```
sudo apt install ngspice
```

```
sudo apt install geda
```

*Please note that this commands has to be run as a root user to install it properly!*

### 3 Drawing the Schematic

The first part of the method is to have a working schematic. To achieve this open gschem. Draw this schematic and add all the values as shown in figure.

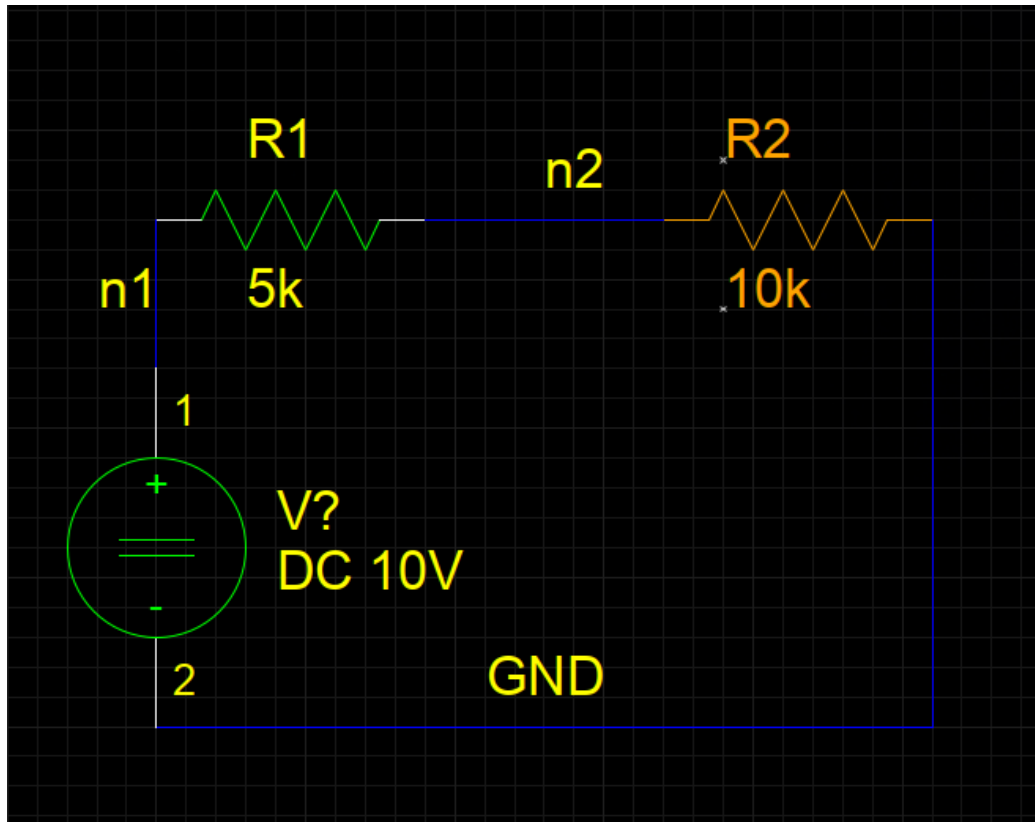


Figure 1: Schematic

## 4 Generating the netlist!

Follow the commands in the terminal from 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: Schematic

under the part Initial Transient Solution in the figure we can see that we have some values. We will plot the graph using this values to better visualize what is happening in the circuit.

## 5 Plot

After performing all commands described in the previous section. we can see we obtain a graph similar to this.

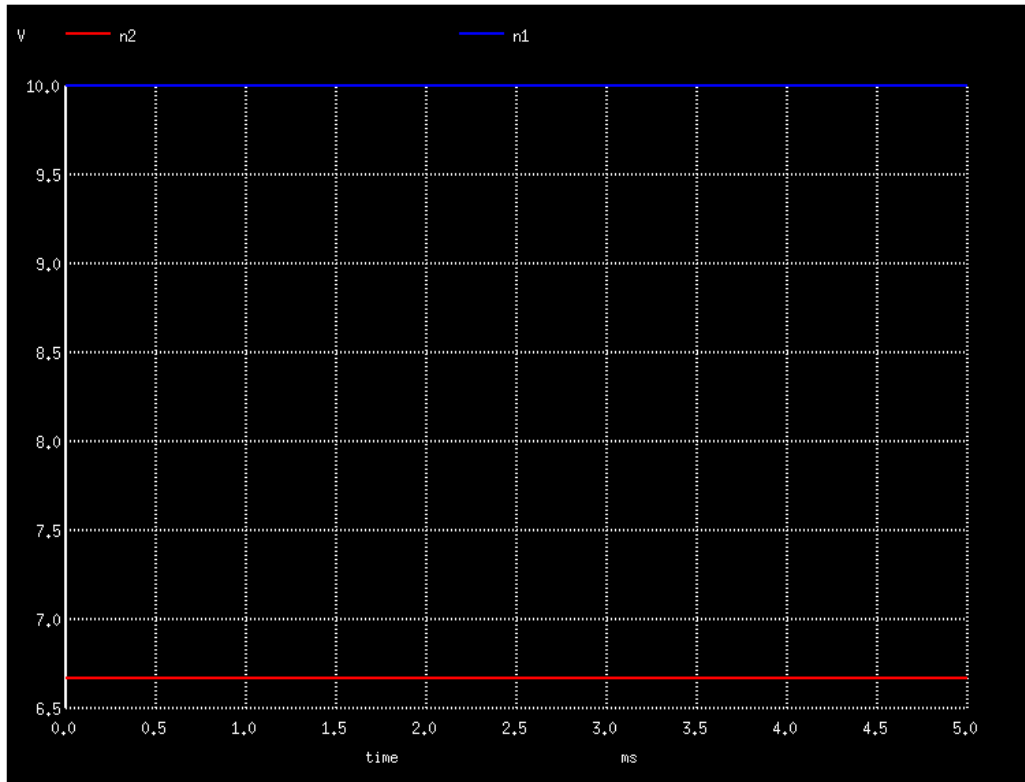


Figure 3: Schematic

## 6 Result

As we can see that the circuit is a simple voltage divider. but using this form of analysis gives us much better understanding of whats happening in the circuit. and at the same time allows for solving the curcuit with much less effort and accuretlly.

**Thankyou for Reading!**