

RTR108 - Circuit Simulation Report

Bruno Bevilacqua Nascimento

March 2020

1 Introduction

In this report, it will be explained how to do circuit simulations using *gschem* and *ngspice* software.

2 Body

2.1 Creating circuit using *gschem*

First, we need to open *gschem* software for creating a circuit. The circuit chosen in this report is a **High-Pass Filter** Circuit composed by the following components:

- Voltage Source 10V (1kHz);
- Capacitor 3,3nF;
- Resistor 1000 Ω .

The *Figure 1* shows the schematic drawing of the circuit.

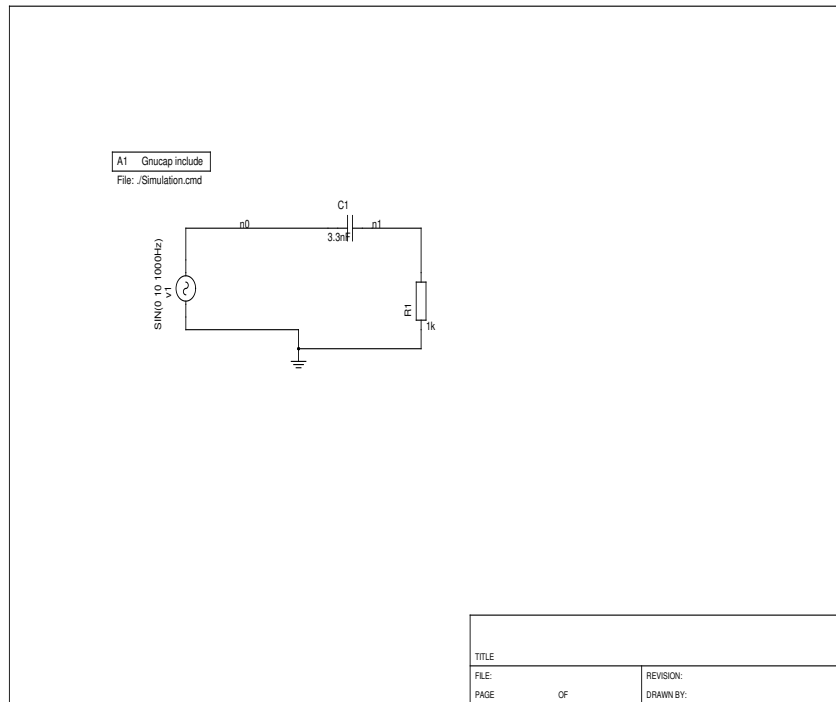


Figure 1: High-Pass Filter Circuit

2.2 Generating *netlist*

In order to generate your *netlist*, execute the command below.

```
## Command to generate gnetlist
getlist -g spice-sdb -o outputFile.net circuitSimulated.sch
```

The final result for file *.net* can be seen above:

```
* gnetlist -g spice-sdb -o testCircuit.net testCircuit.sch
*****
* Spice file generated by gnetlist                                     *
* spice-sdb version 4.28.2007 by SDB --                               *
```

```
* provides advanced spice netlisting capability.  *
* Documentation at http://www.brorson.com/gEDA/SPICE/ *
*****
***** Begin SPICE netlist of main design *****
v1 0 n0 SIN(0 10 1000Hz)
.INCLUDE ./Simulation.cmd
R1 0 n1 1k
C1 n0 n1 3.3nF
.end
```

2.3 Simulate circuit using *ngspice*

In order to simulate circuit, we are going to use *ngspice*. Execute *ngspice* command in order to start the software and begin simulation.

```
ngspice outputFile.net
```

After this, you can execute *run* command and see the result.

```
Activities Terminal Thu 09:37
user@epk428-4:~$ cd /RTX208/circuitTests
user@epk428-4:~$ ls
Desktop Documents Downloads Music Pictures Public RTX208 Templates Videos 'VirtualBox VMs'
user@epk428-4:~$ cd RTX208/
user@epk428-4:~$ ls
circuitTests diaries exercises.phyton git-upload initial_exercises list_exercises README.md
user@epk428-4:~$ cd circuitTests/
user@epk428-4:~$ cd circuitTests$ ngspice
*****
** ngspice-27 : 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: Tue Dec 26 17:18:28 UTC 2017
*****
ngspice 1 -> source testCircuit.net

Circuit: * gnetlist -g spice-sdb -o testcircuit.net testcircuit.sch

ngspice 1 -> run
Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

Warning: v1: no DC value, transient time 0 value used

No. of Data Rows : 1

Initial Transient Solution
-----
Node          Voltage
-----
n0             0
n1             0
v1#branch      0

No. of Data Rows : 512
ngspice 1 ->
```

Figure 2: *Ngspice* final results

With the defined nodes, we can plot the behaviour of our function typing the following command:

```
plot n0 n1 v1#branch
```

The software will generate the following graph.

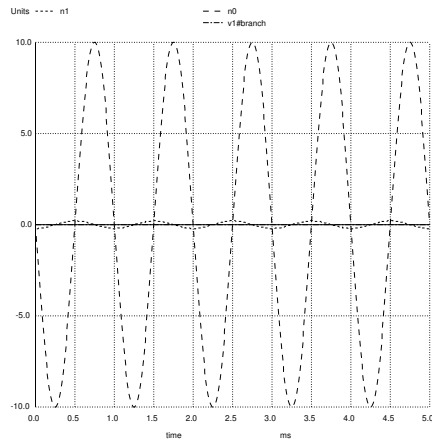


Figure 3: Complete graph of the circuit simulate

In order to see the entry signal you can execute the following command.

```
plot n0
```

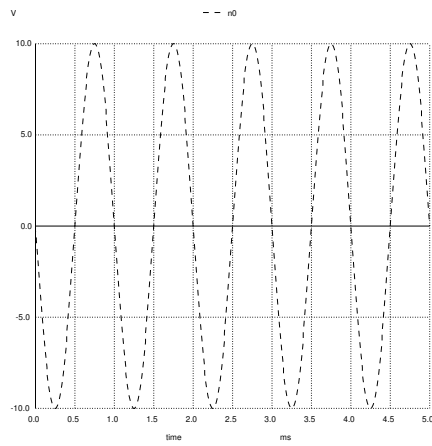


Figure 4: Entry signal $n0$

In order to see the final signal you can execute the following command.

```
plot n1
```

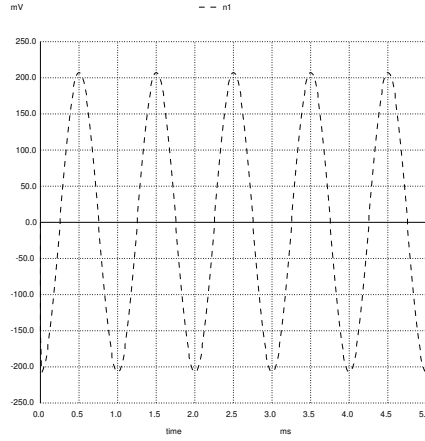


Figure 5: Final signal $n1$

3 Conclusion

In this report it was shown how to simulate a circuit. The chosen circuit was **High-Pass Filter** composed by a capacitor, resistor and a voltage source. In the final result of circuit simulation we can see the difference between the two graphs (*Figure 4* and *Figure 5*). The entry signal has an amplitude of 10V, same of the voltage source. Otherwise, final signal has an amplitude of 200mV. In order to see the same amplitude we need to increase frequency. For example, if we increase the frequency to 100kHz, entry signal and final signal will overlap like the graph below.

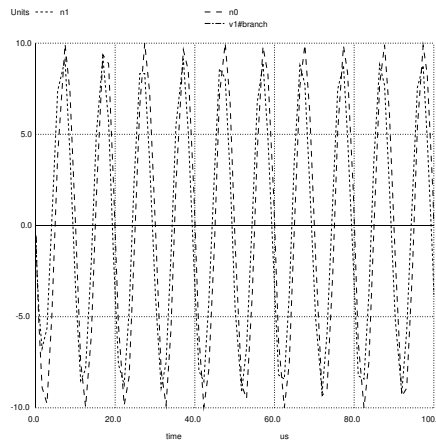


Figure 6: Entry signal and final signal overlapping for a frequency of 100kHz

Both of signals have amplitude of 10V. So, in this report we could see a simulation of a **High-Pass Filter** and know how to simulate using *gschem* and *ngspice*. Basically, in order to generate your circuit you will have to follow the step-by-step below.

1. Draw Circuit using *gschem*;
2. Generate netlist using *gnetlist*;
3. Simulate circuit using *ngspice*;
4. Analyze your results.