

ngspice01

antoine.remy99

March 12 2020

1 GSchem

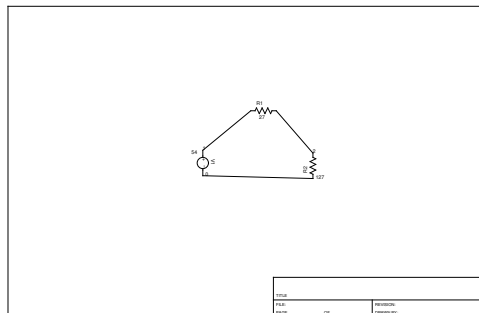
I started this project by creating a gschem document to create my circuit I used one voltage generator and 2 resistor :

-Resistor 1 as a value of 27Ω

-Resistor 2 as a value of 127Ω

-Voltage generator as a value of 54V .

And I linked all the component to obtain this :



I finally transformed the .sch into a .net by using gnetlist.

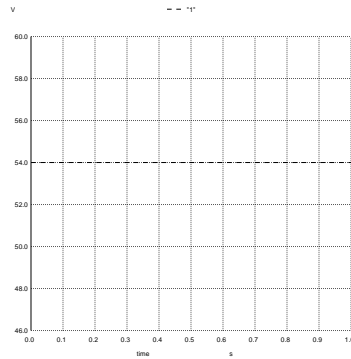
the file net file was :

```
* Spice netlister for gnetlist
V1 1 0 54
R2 0 2 127
R1 1 2 27
.END
```

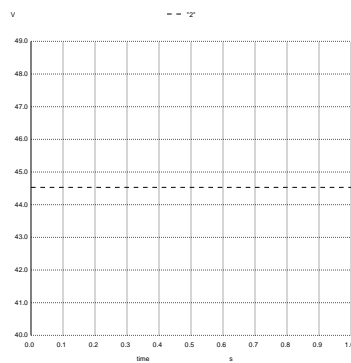
2 NGspice

After exporting the .sch into a .net using gnetlist.

I used ngspice to calculate the different voltage of all the resistors by using command : `-tran 5 1 -plot "1" -plot "2"` as show below:



Resistor 1



Resistor 2

3 Conclusion

This work allow me to learn how to create electrical circuit in GSchem and to evaluate them using NGspice.