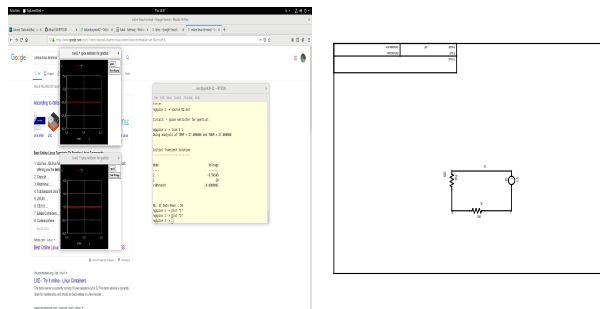# laboratorywork2

dhruvsavani510

March 2020

1   Introduction


```
* spice netlister for gnetlist
V1 0 1 10
R2 0 1 105
R1 1 1 5
.END
```

 in this project i make circuit with v1,r1,r2 and there's two nodes by using gschem
 and run circuit in ngspice.

 I lear how to plot circuit in ngspice using gschem and add all details to LaTeX.
 THANK YOU!