

# ECE112 Lab Report - Linux and Spice Simulation lab

Jacob Reger

February 12, 2020

## Lab 6a - ECE112

### Here is what I did

This lab was about running the spice simulator and printing some results with latex. My results from this lab are shown in figure 1.

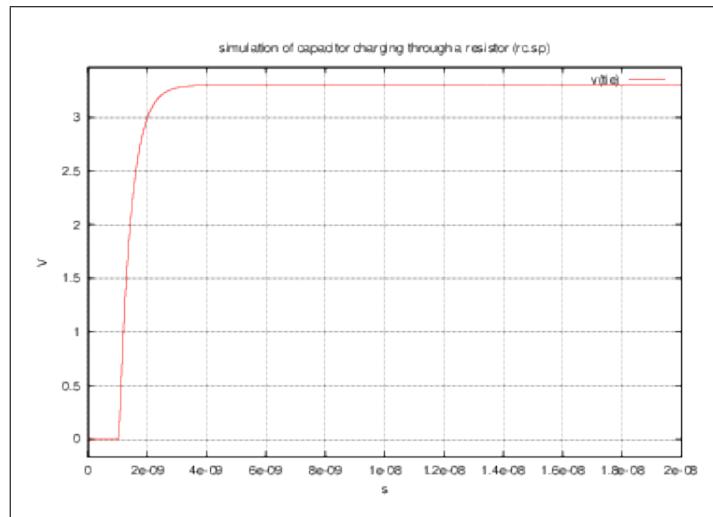


Figure 1: RC Simulation

### Spice netlist

Here is the spice input file I used:

```
Simulation of capacitor charging through a resistor (rc.sp)

vsrc vin 0 PULSE(0 3.3 1ns 100p 100p 100e-9 250e-9) ;voltage source
r1 vin tie 400 ;series resistor
*c1 tie 0 10pf ;capacitor to ground
c1 tie 0 1pf ;capacitor to ground

.control
  set hcopydevtype=postscript
  set hcopypscolor=0
  set color0=rgb:f/f/f
  set color1=rgb:0/0/0
  tran 10ps 20ns
  plot V(tie) xl 0.1ns 20ns
  gnuplot sim_output V(tie) xl 10ps 20ns
.endc
.end
```

Signed: \_\_\_\_\_