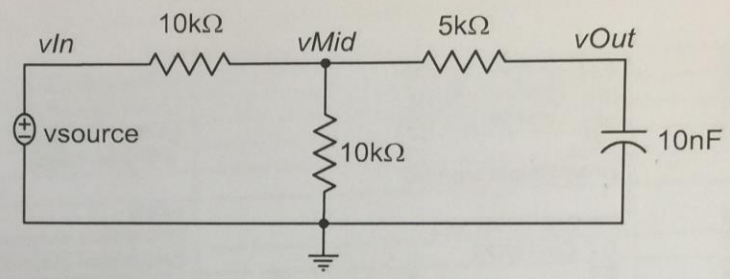
Olouwole Eteka
EE330 lab1 Report Section C

Prelab

olouwole ETEKA

Lab 1: Cadence® Custom IC design tools - Setup, Schematic capture and simulation

Pre-lab (To be completed before start of lab)



For the circuit shown above, answer the following questions:

1. What is the function of the circuit? low pass Filter
2. What is the Thévenin equivalent resistance seen by the capacitor? 10K
3. (dc analysis) If a dc voltage is applied by vsource, what would be the dc voltage at the vOut node? At the vMid node? $V_{OUT} = \frac{10K}{10K+10K} \times V_{in} = \frac{1}{2} V_{in}$ $V_{mid} = V_{out} = \frac{1}{2} V_{in}$
4. (ac analysis) What is the -3dB corner frequency of the circuit? For simulations, what start and stop frequencies would you choose in order to observe the frequency response band of interest? $F_C = \frac{1}{2\pi 10C} = 1.59 KH_2$
   0 to 16 KH2
5. (transient analysis) What is the response of the circuit to a step voltage supplied by vsource? $V_c = \frac{V_s}{2} - \left[\frac{V_s}{2} - V_i\right] \cdot e^{(-t/Rc)}$
6. (transient analysis)Assume a pulse with finite magnitude, finite rise time, finite fall time, and non-zero pulse width is to be applied to the circuit. Determine these parameters of the input pulse in order to easily observe the response of the circuit to the rising and falling edges. How long would you need to run a transient analysis to get all the information?
   Run it for one full cycle @ 10 ms For 5 periods

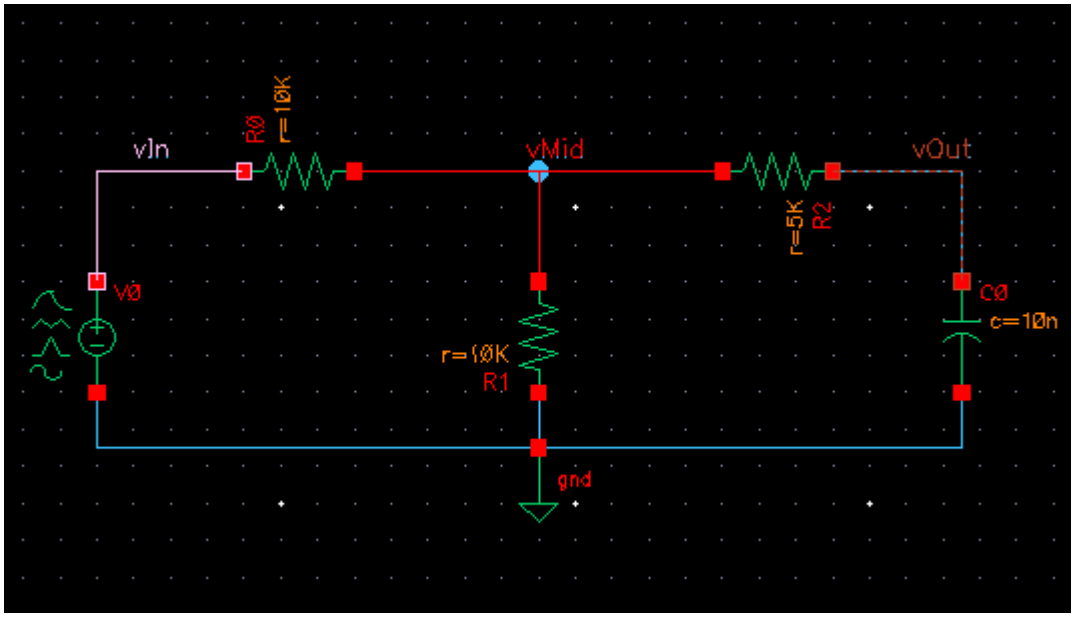
Introduction

In this lab installed the software to and familiarize ourselves with it. We run a stimulation on a simple RC circuit with DC, AC, and Transient signals as source. We learned Cadence Custom IC design tool, Virtuoso, and the basics of using Schematics Editor.

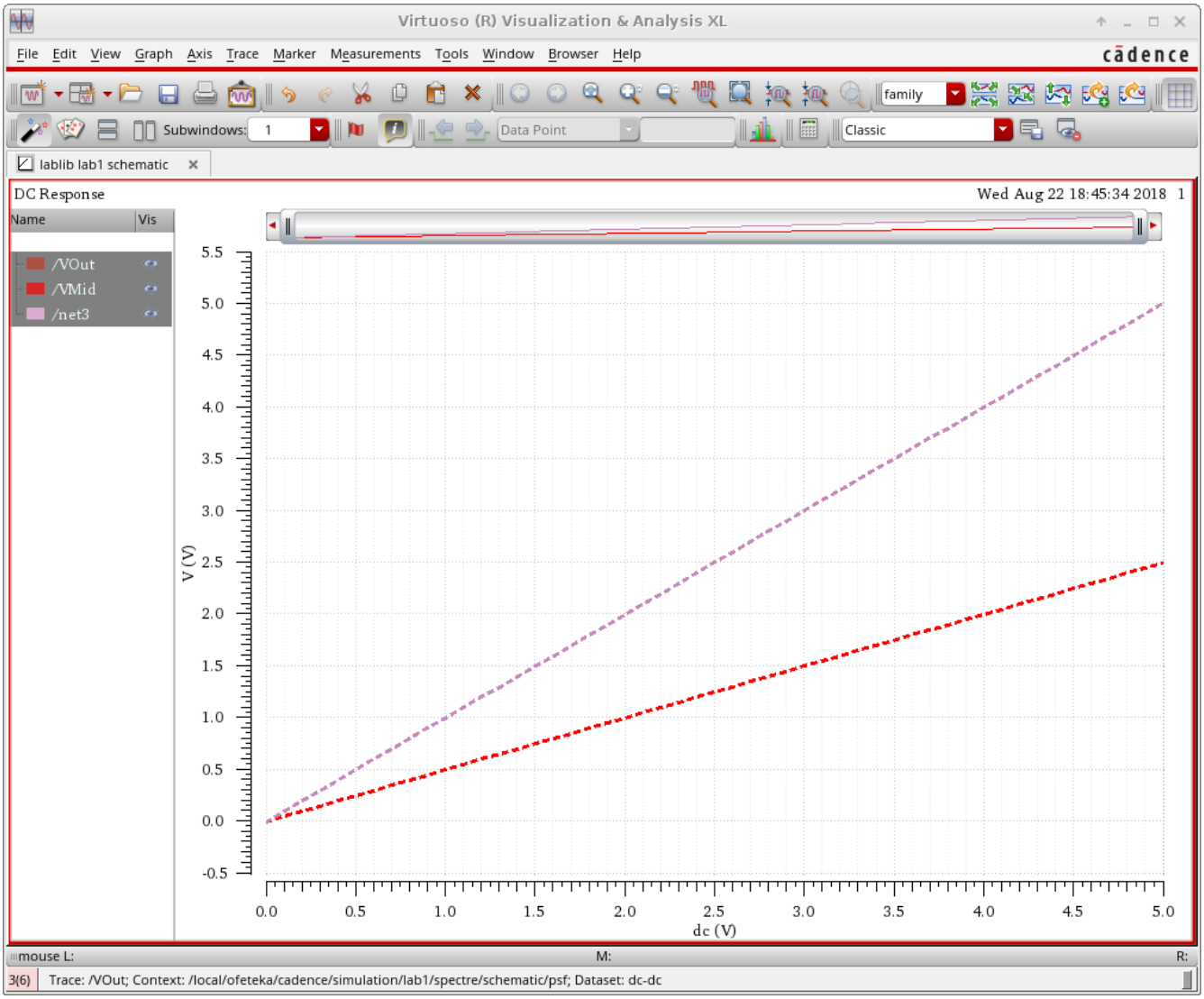
Part 1

DC analysis

Implementation of the RC circuit in Cadence after setting up the software. My circuit it composed of 3 resistors and a capacitor with a DC source voltage for my first stimulation.

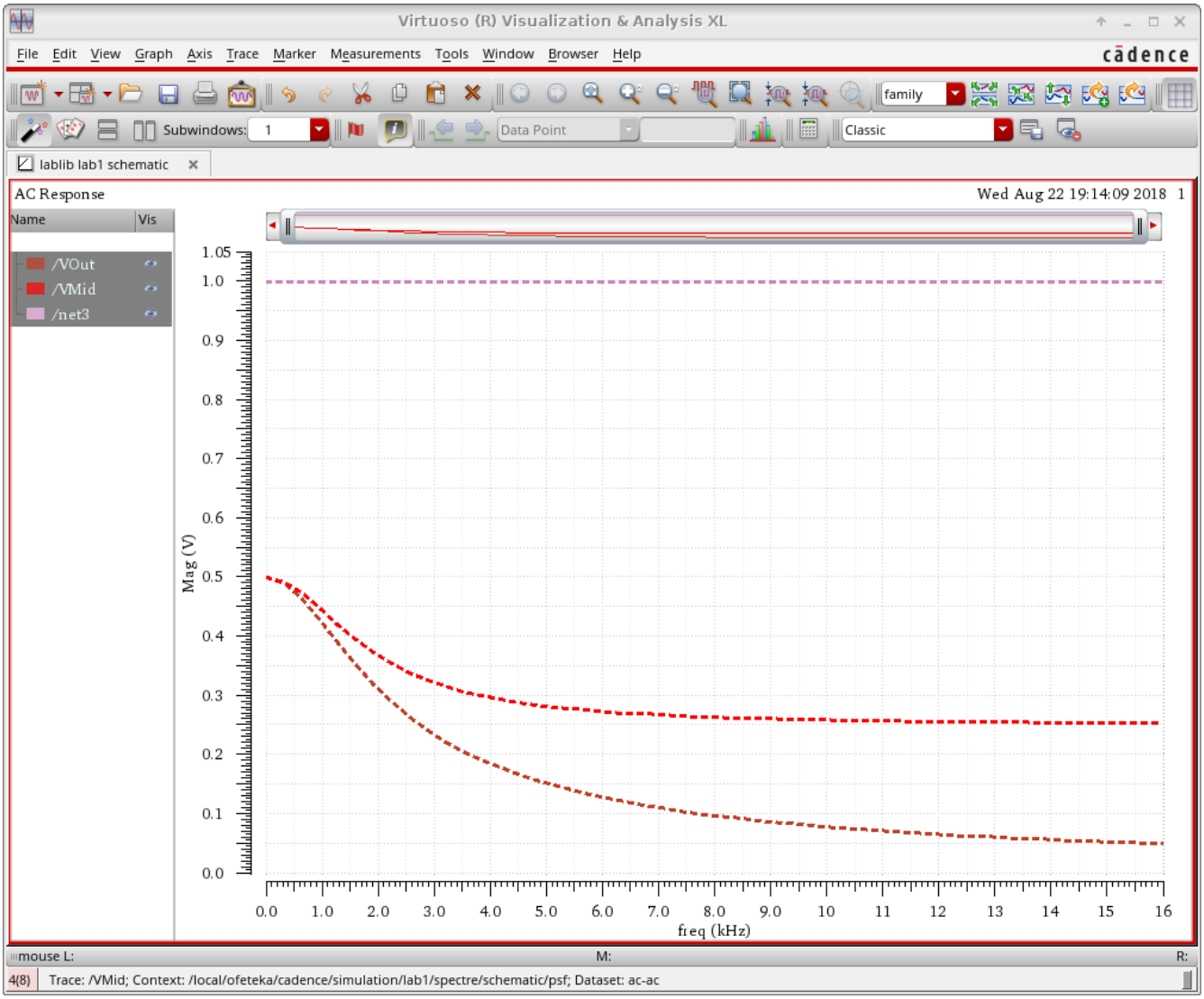


The calculations in Pre-Lab are demontrated trhough the graph below that vMid is similar to vOut and they are increasing as the input voltage increase. It shows to be true as vMid=vOut=vSource/2.
We can also observe that the slope of vIn is 2x of the slope of vMid and vOut, which still supports my calculation.



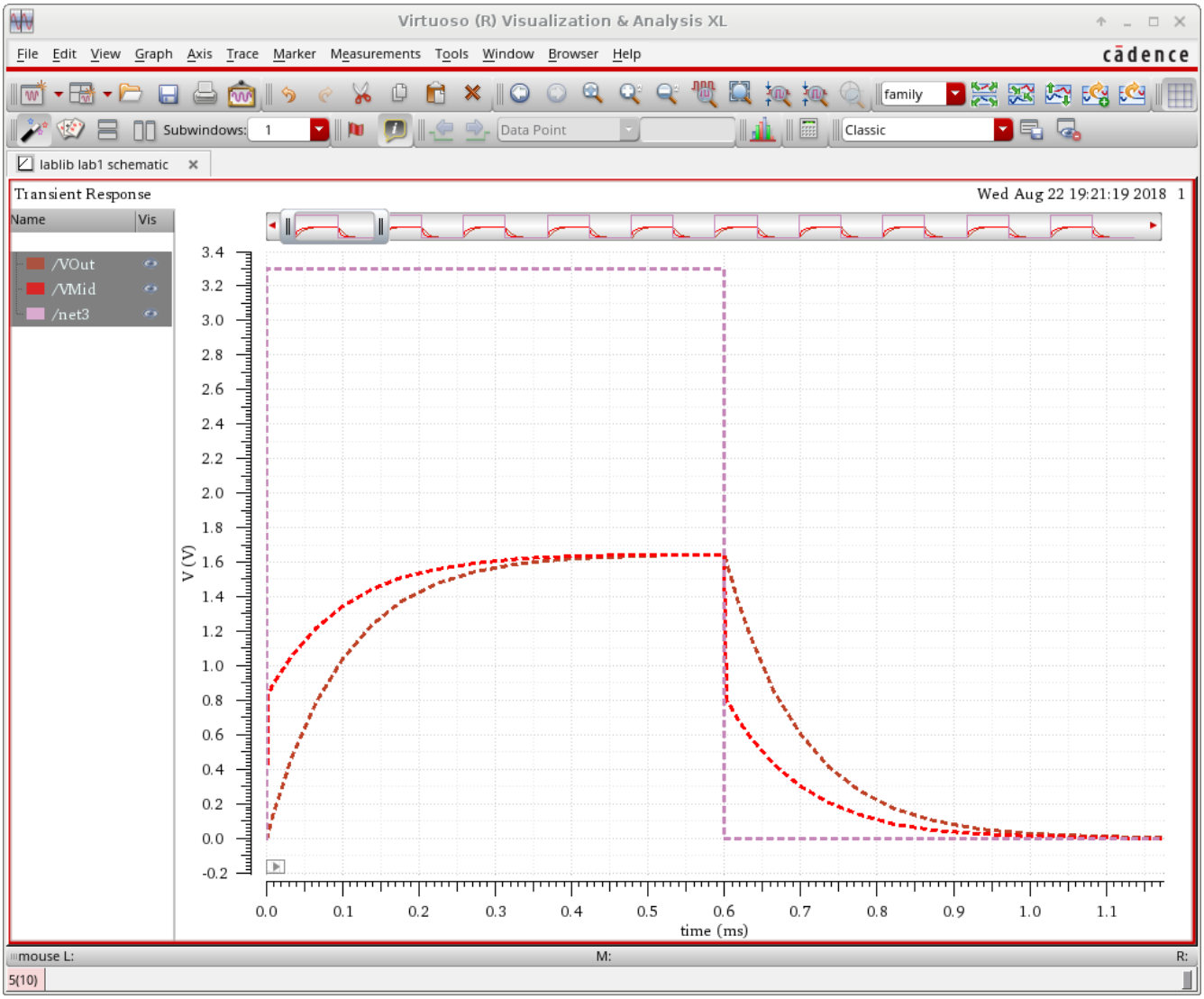
AC analysis

The second part was to run an AC analysis. I sweep the frequency from 1Hz to 1MHz. As we observe from the graph, vOut and vMid maintain at 0.5v for frequency <100Hz. Then, the both start reduces exponentially. vMid begins to maintain at 0.25v at about 10kHz while vOut begins to maintain at 0 starting 100kHz. The result was much of what I was expecting. In the prelab, I found that 1.59kHz for -3dB corner frequency. According to the graph, the maximum magnitude is 0.5v therefore (0.707*0.5v = 0.3535v) is the -3dB magnitude and magnitude 0.3535v lies between 1kHz and 10kHz which I not far from the answer.



TRANSIENT analysis

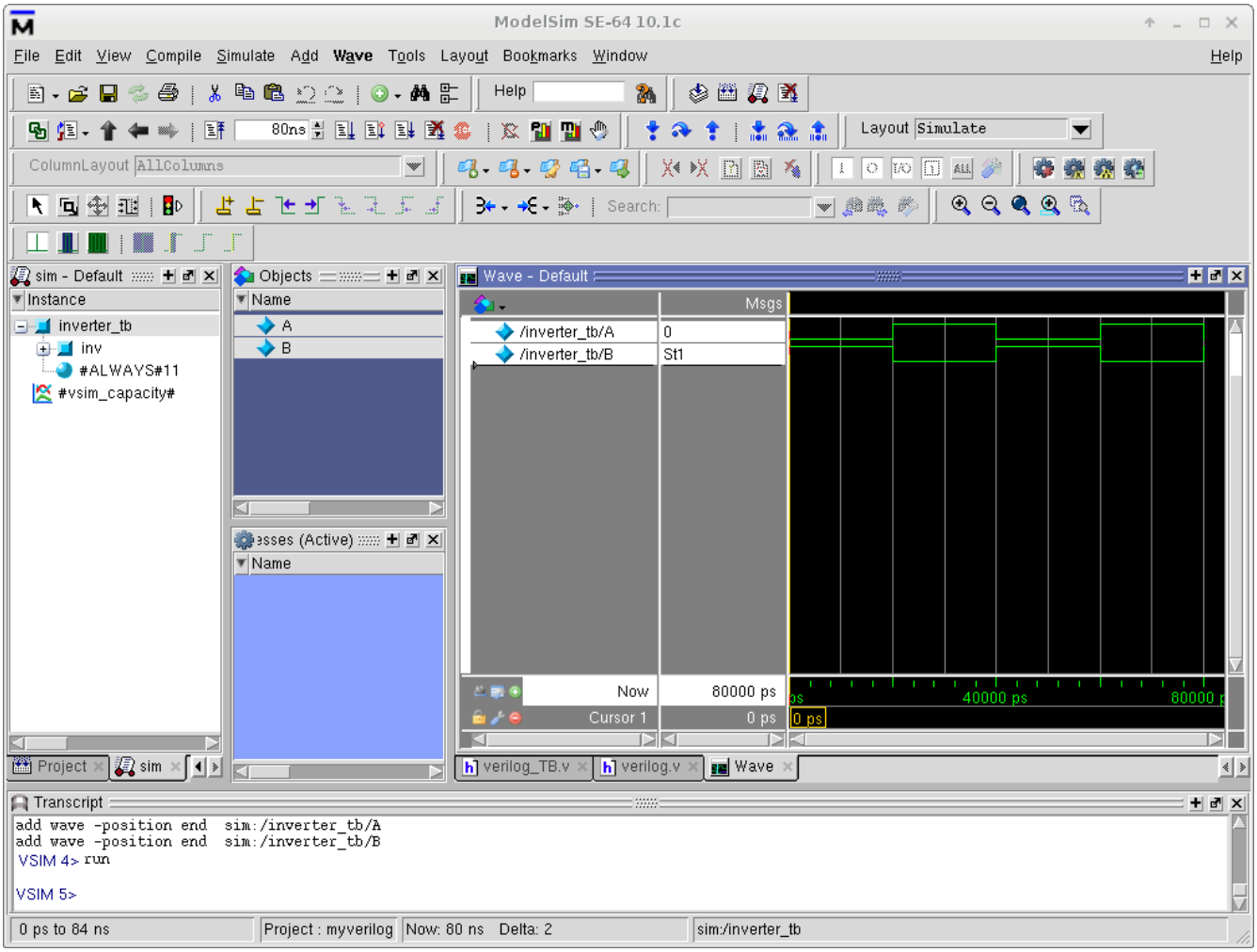
we know that the unit step response from the capacitors determine the level of our transient. From our graph below we can see the whole discharging and charging effect of the capacitor and the full period. The capacitor start charging until 0.6ms where is start discharging to 1.1 and will repeat the cycle. The graph shows the input (step input) VS vMid VS vOut and the all follow the same pattern.



part2

Inverter

We implemented an inverter using the Modelsim stimulation. The Verilog code for an inverter and the it stimulation code (testbench module) was provided I used to create “inverter.v” and “inverter_tb.v”. from there I create a simulation waveform window and simulate the design with 80ns as the length of time I would like to simulate.



Conclusion

in this lab, I've learned the basics on setting up Cadence and ModelSim on a Linux operated computer. Linux operating system was unfamiliar to me but learning the commands as they will help me achieve most objectives by just typing out commands in the terminals. Also I learned that Cadence is a very convenient software for engineers to reduce time needed to do the calculations step-by-step or having a physical circuit which also save a lot in costs.