Summer Fellowship Report

On

Spice Simulations using Kicad nightly builds

Submitted by

Akshay NH and Athul MS

Under the guidance of

Prof.Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

July 2, 2018

Acknowledgment

We are extremely thankful to Prof . Kannan Moudgalya for guiding and motivating us throughout the FOSSEE fellowship programme.We would also like to thank our mentors Mrs.Gloria Nandihal and Mr.Athul George for their immense support and advice.Moreover, we are grateful to our fellow friends Mudit Joshi and Ashutosh Gangwar for assisting us with their programming knowledge.Lastly,we extend our warm gratiutude to the managers and staff of FOSSEE for their co-operation and assistance.

Contents

Chapter 1

Introduction

1.1 Introduction

KiCad is an open-source software tool for the creation of electronic schematic diagrams and PCB artwork.KiCad can be considered mature enough to be used for the successful development and maintenance of complex electronic boards.KiCad does not present any board-size limitation and it can easily handle up to 32 copper layers, up to 14 technical layers and up to 4 auxiliary layers. KiCad can create all the files necessary for building printed boards, Gerber files for photo-plotters, drilling files, component location files and a lot more.Being open source (GPL licensed), KiCad represents the ideal tool for projects oriented towards the creation of electronic hardware with an open-source flavour.

Stable builds Stable releases of KiCad can be found in most distributions package managers as kicad and kicad-doc. If your distribution does not provide latest stable version, please follow the instruction for unstable builds and select and install the latest stable version.

Unstable (nightly development) builds Unstable builds are built from the most recent source code. They can sometimes have bugs that cause file corruption, generate bad gerbers, etc, but are generally stable and have the latest features.

Chapter 2

Steps to follow for better understanding of kicad nightly builds for electronic simulation

2.1 Steps to follow for better understanding of kicad nightly builds for electronic simulation

- Download Kicad Documentation from `http://docs.kicad-pcb.org/stable/en/getting_started_in_kicad.pdf`
- Go through the working of Kicad.
- Download `https://github.com/KiCad/kicad-source-mirror`
- Go to demos-¿simulation where you will find three example circuits
- Figure out the working of simulations and adding library files to components as shown in the examples.
- Post questions on the Kicad forum if you are facing any difficulties.

Chapter 3

ANALOG CIRCUITS

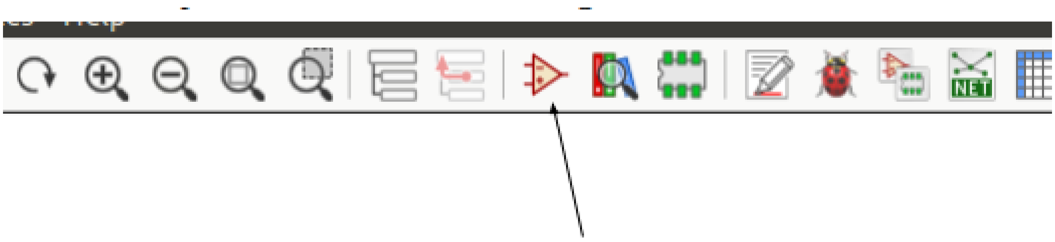
3.1 ANALOG CIRCUITS

3.1.1 Second Order low pass filter

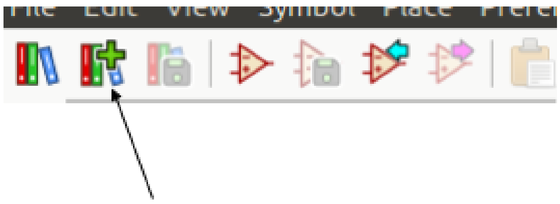
Problem Statement: Plot the input and output waveform of second order low pass filter and verify the same using AC Analysis

Solution:

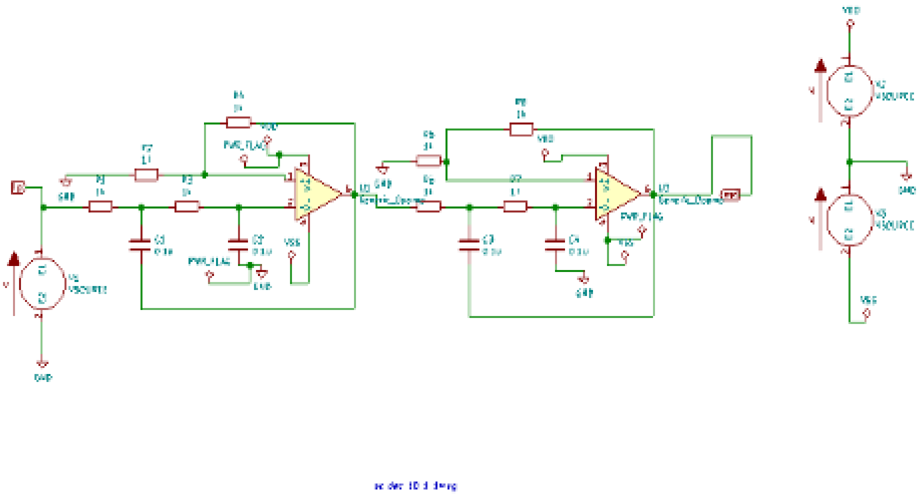
- Create a schematic in Kicad as shown in the diagram below
- Download `https://github.com/KiCad/kicad-source-mirror`
- To add generic op amp model click on create,delete and edit symbols in the top toolbox as shown in diagram.



- Next click on Add an existing library in the next window

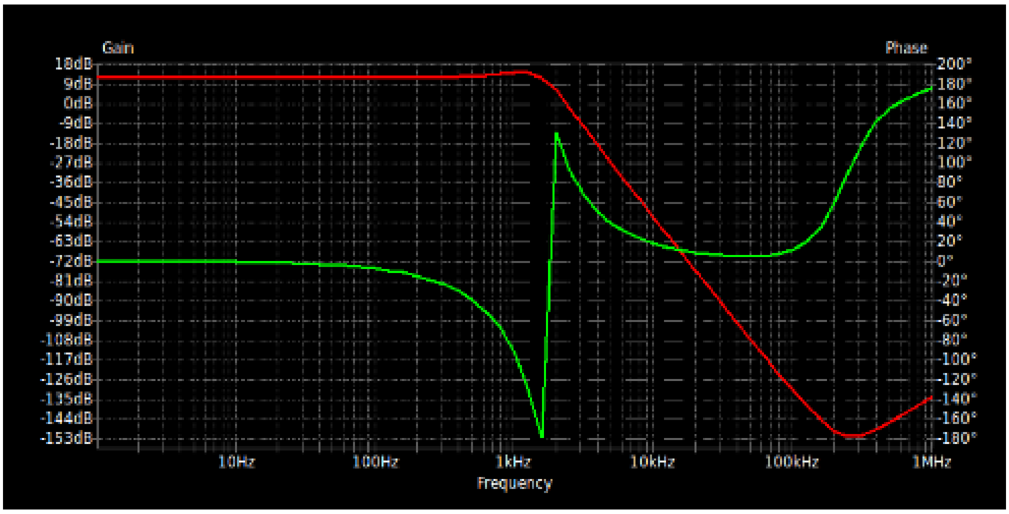


- Go to kicad-source mirror master demos →simulation →sallen_key→sallen_key_schlib.lib
- Finally generic op amp is added to your components library ,add it to your schematic and right click on modeledit properties→edit→ spice model→model.Give library as 8051.lib present in Kicad source mirror and Type as subcircuit.
- Provide ac analysis in simulation and run the schematic



Circuit

Waveform



3.1.2 DIFFERENCE AMPLIFIER

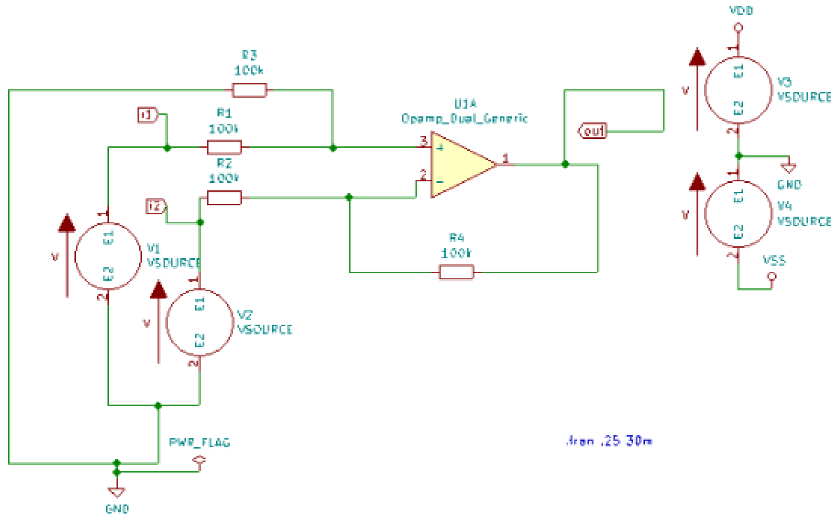
Problem Statement: Plot the input and output waveform of difference amplifier and verify the same using transient analysis

SOLUTION:

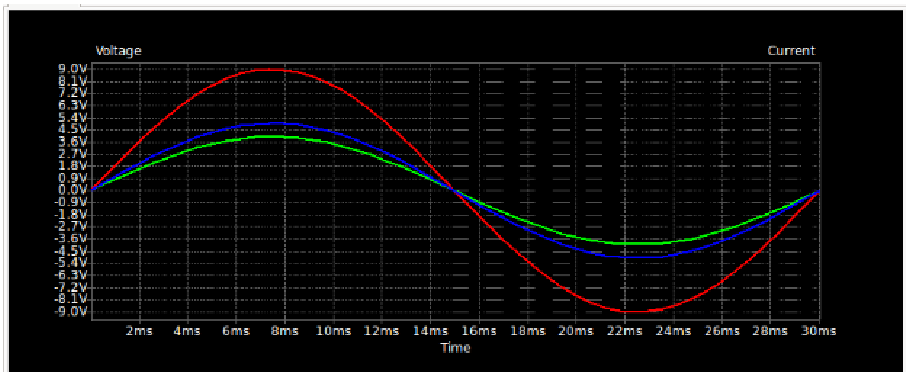
- Create a Kicad schematic as shown in the diagram.
- Use a generic op amp as opposed to any other op amp as you would need to create your own netlist for circuit to work.
- To learn how to create netlist visit `http://www.ecircuitcenter.com/Circuits/opmodel1/opmodel2.htm`
- Netlist for difference op amp

- Give transient analysis and run the simulator for waveforms.

CIRCUIT



WAVEFORM

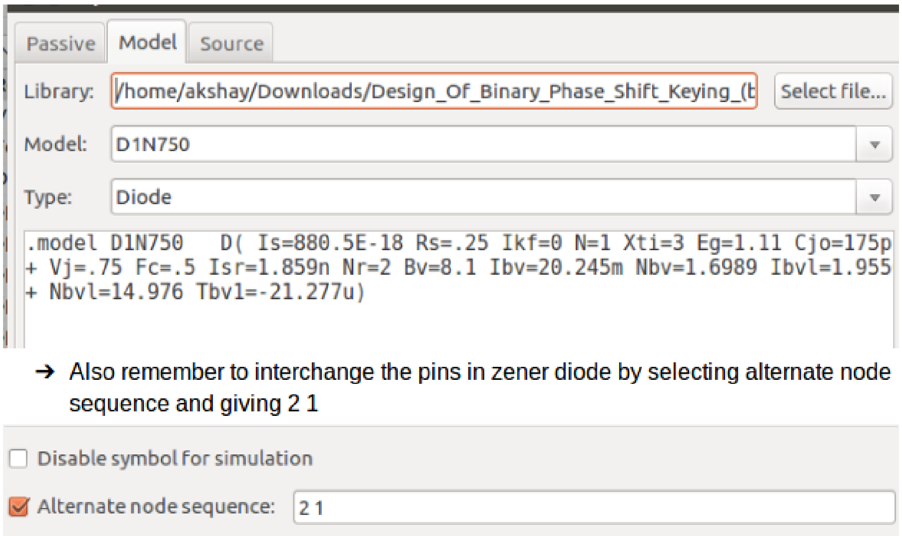


3.1.3 Voltage Regulator

Problem Statement: Plot the input and output waveform of Voltage Regulator and verify the same using transient analysis.

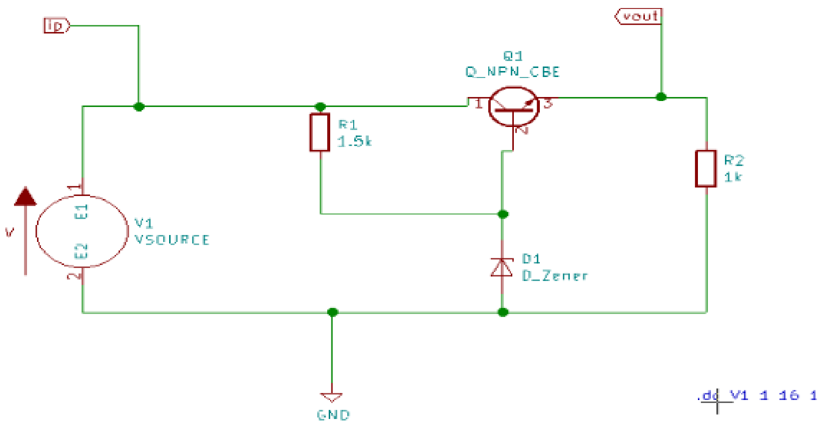
Solution:

- Create a Kicad schematic as shown in the diagram
- To use the transistor Q_NPN_CBE in your components library ,follow the steps mentioned in example 2.2.1.
- Go the Kicad-source mirror master→demos→simulation→laser_driver and add Laser_driver_schlib.lib
- To add .lib file for transistor ,right click on model→edit spice model→add fzt1049a.lib present in laser_driver and type as BJT
- For zener diode select D_Zener present in devices of components library and add netlist for the same as shown in in figure below.

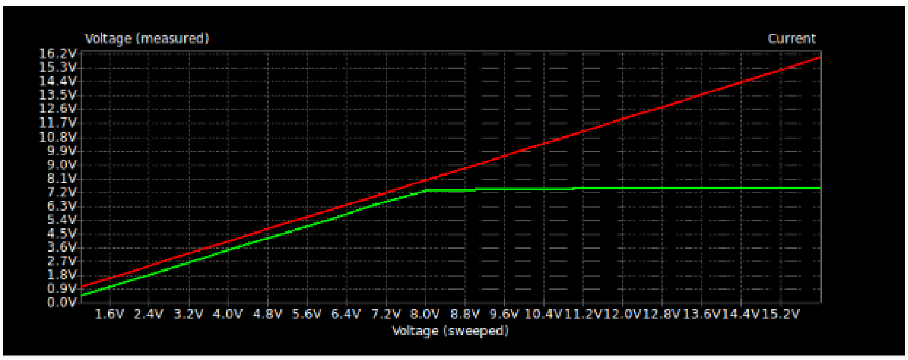


- Provide dc analysis and run the simulation

CIRCUIT



WAVEFORM



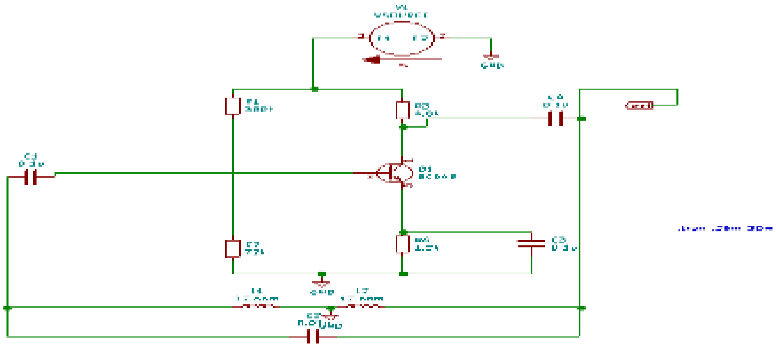
3.1.4 Hartley Oscillator

Problem Statement: Plot the output waveform of hartley oscillator and verify the same using transient analysis

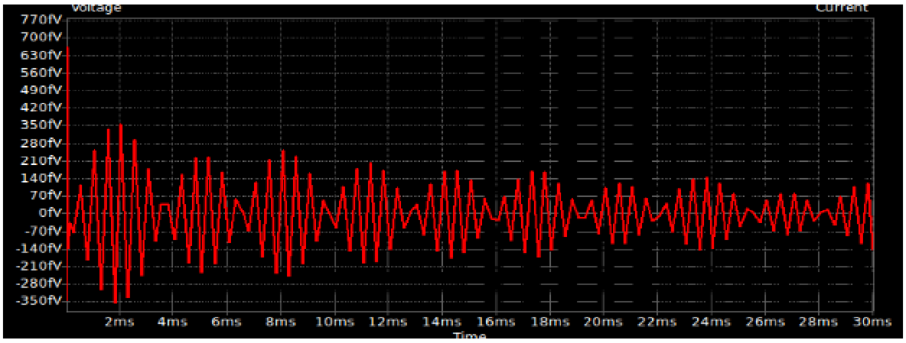
Solution:

- Create a Kicad schematic as shown in the diagram
- Use BC548 transistor present in components library and add NPN.lib present in libs folder in analog circuits in my github `https://github.com/FOSSEE/eSIm-Kicad-Simulations`
- Use transient analysis and run simulation

CIRCUIT



WAVEFORM



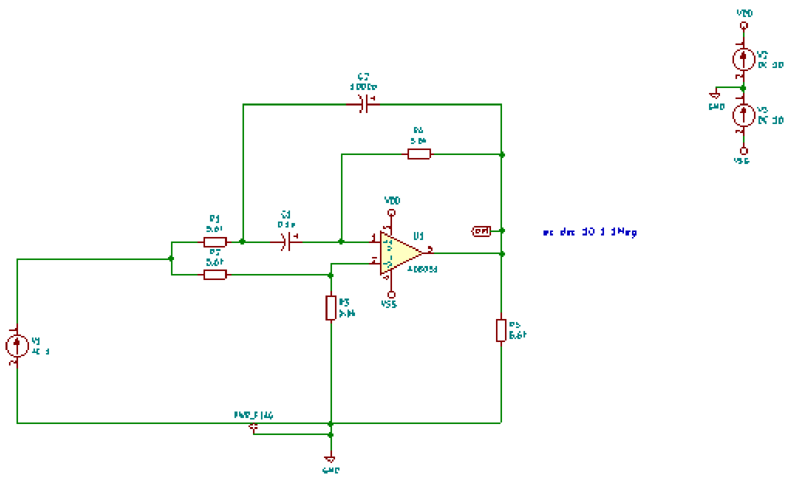
3.1.5 Notch Filter

Problem Statement: To design a notch filter that resonates at 2 kHz and verify using AC analysis in Kicad.

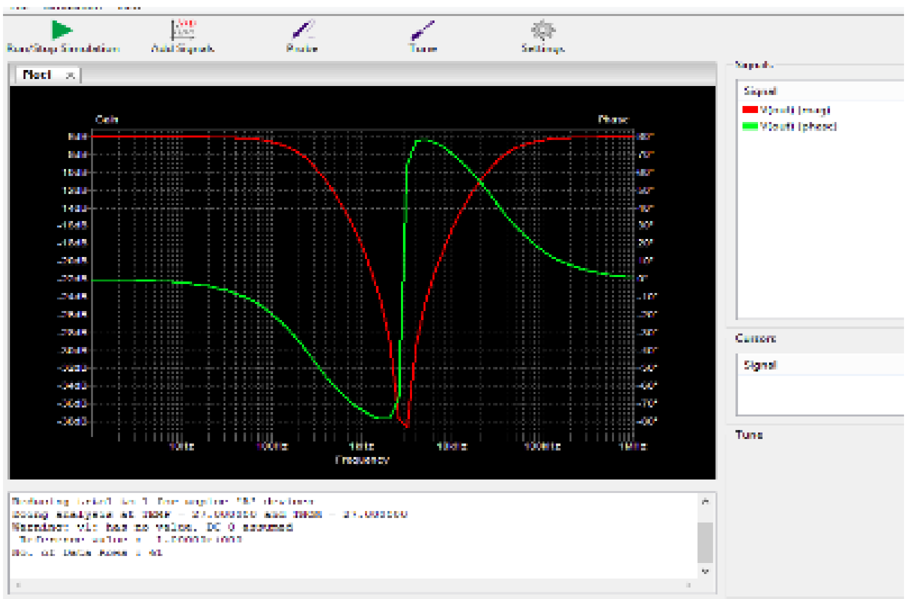
Solution:

- Create a schematic in Kicad using the circuit diagram
- Use op-amp model AD8051 in the circuit (recommended)
- Add .cir file of the AD8051 op-amp in Edit Sources → Model → Library.
- Do ac analysis from a frequency range of 10 Hz to 10 Megahertz
- Verify the resonant frequency from the simulated output plot

CIRCUIT SCHEMATIC:



WAVEFORM:

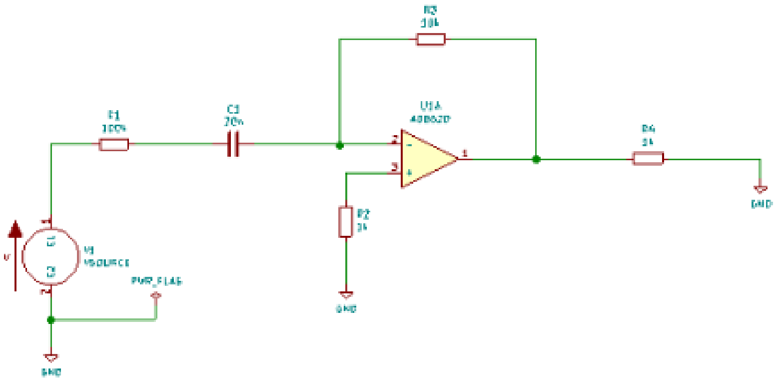


3.1.6 DIFFERENTIATOR:

Problem Statement: To realise an op-amp differentiator using Kicad.
Solution:

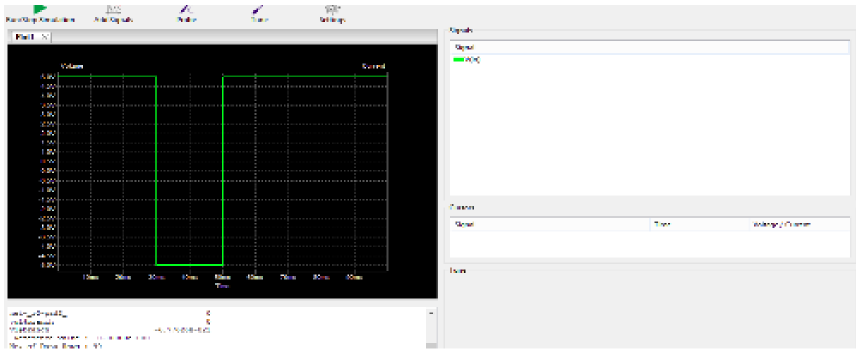
- Create a schematic in Kicad using the circuit diagram
- Use op-amp model AD8620 in the circuit.
- Do transient analysis upto 100 milliseconds with a time-step of 5ms
- Verify the simulated output plot

CIRCUIT

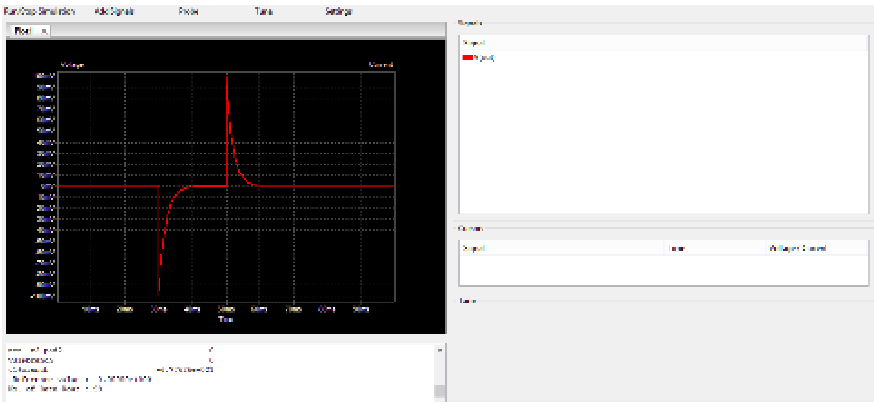


WAVEFORM

1. INPUT



2. OUTPUT



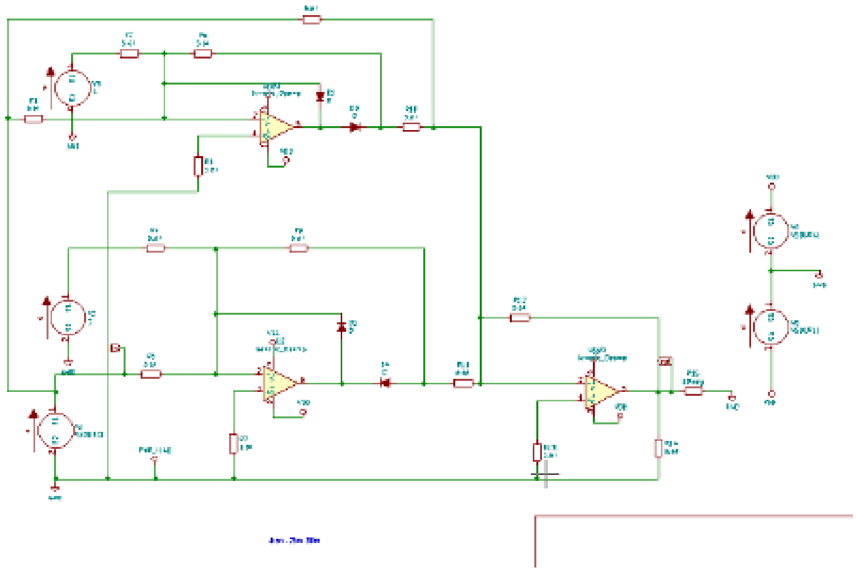
3.1.7 PRECISION CLIPPER :

Problem Statement: To design a precision clipper circuit using Kicad
Solution

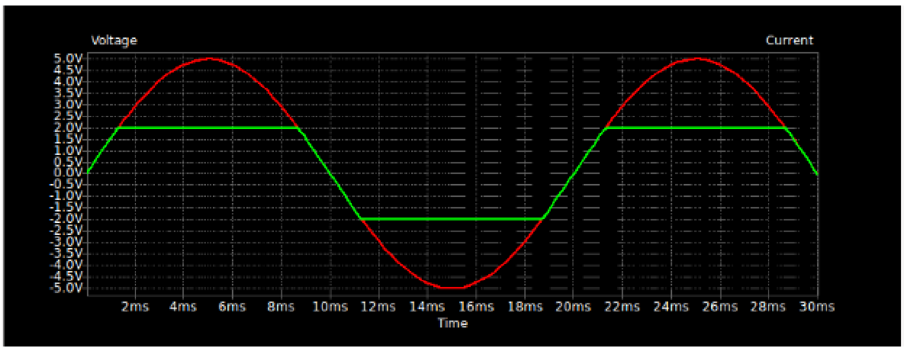
- Create a schematic in Kicad using the circuit diagram
- Use a generic op-amp model in the circuit (recommended)
- Add .cir file ad8051.cir in Edit Sources → Model → Library.

- Do transient analysis upto 100 milliseconds with a time-step of 5ms
- Note-Do not interchange pins of diode in this case
- Verify the simulated output plot

CIRCUIT



WAVEFORM

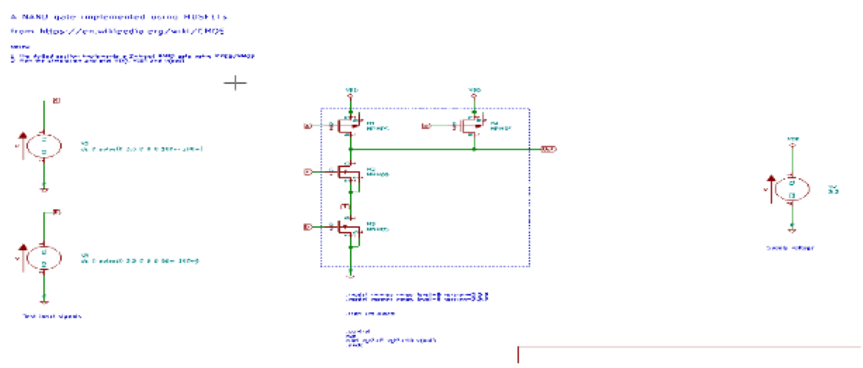


Chapter 4

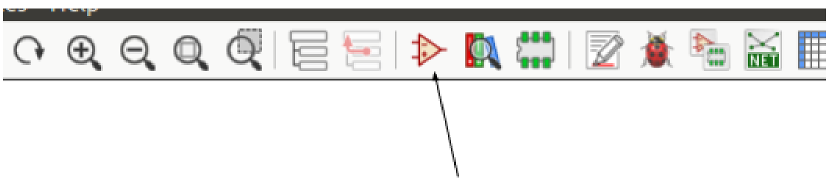
SUBCIRCUIT BUILDER METHOD

4.1 SUBCIRCUIT BUILDER METHOD

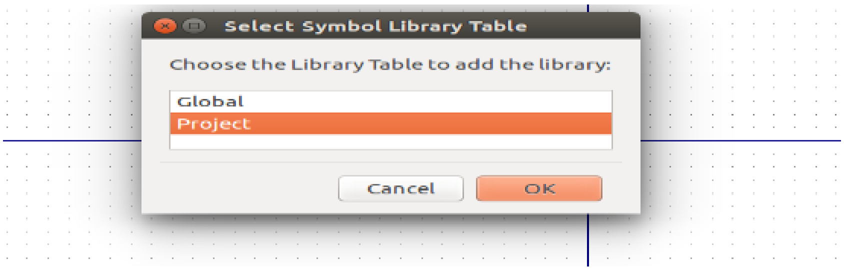
- Subcircuits are used to create large components from simple components.
- Since most Kicad components such as nand gates,flip flops etc need netlists for them to function,we have found that the subcircuit method is most successful for create netlists for circuits which we want to add
- Most digital circuits in Kicad nightly builds can be built by the method of subcircuit.
- To explain the concept of subcircuit,I will take example of creating a nand gate using cmos logic
- First create a nand gate using cmos logic and basic devices as shown in figure below.



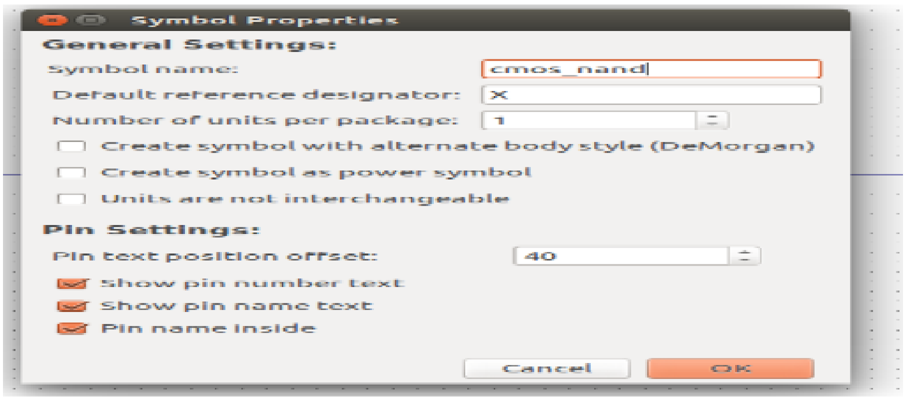
- Generate netlist and simulate.Check whether waveforms are correct.
- Now open a new project and click on create,editor delete symbol in toolbox



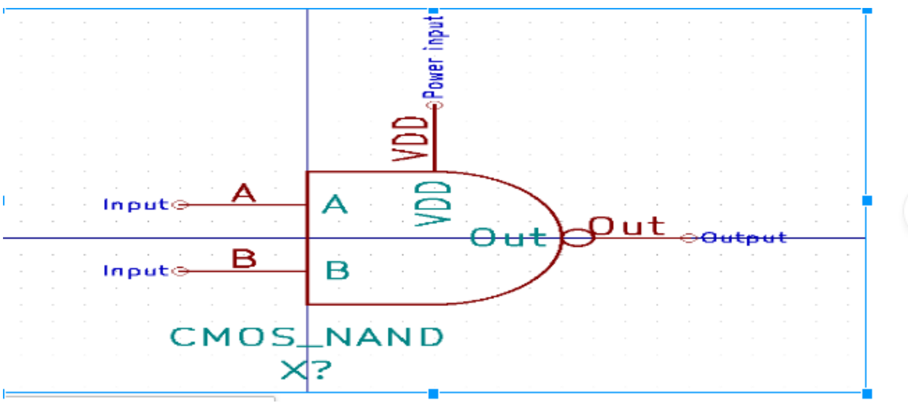
- Next click on create a new library to store all your subcircuit files
- Select symbol libray table to add symbol select project



- Click on create new symbol and select the subcircuit file you have created



- Create the new subcircuit using toolbox provided in the right as shown in figure below



- Next step is to create a subckt file for cmos_nand using the netlist generated copy the contents as highlighted in the figure below

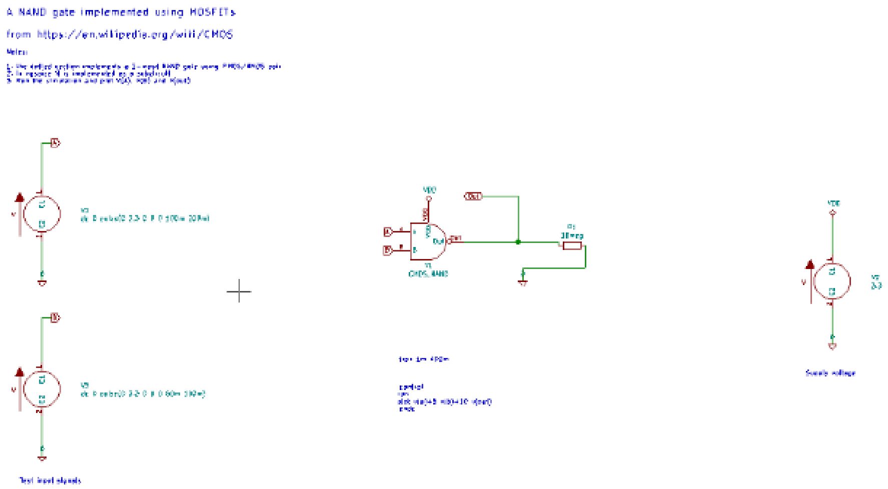
```
.title KiCad schematic
V1 A 0 dc 0 pulse(0 3.3 0 0 0 100m 200m)
V2 VDD 0 3.3
M1 Out A VDD VDD MPMOS
M2 Out A 1 1 MNMOS
M4 Out B VDD VDD MPMOS
M3 1 B 0 0 MNMOS
V3 B 0 dc 0 pulse(0 3.3 0 0 0 50m 100m)
.tran 1m 400m
.model mnmos nmos level=8 version=3.3.0
.model mpmos pmos level=8 version=3.3.0
.control
run
plot v(a)+5 v(b)+10 v(out)
.endc
.end
```

- Next create a subcircuit .lib file for the netlist generated using .subckt and adding pin numbers.Make sure pin numbers for subcircuit is same as that provided for the circuit.Use globel labels for providing pin numbers.Generated subcircuit would look as follows.

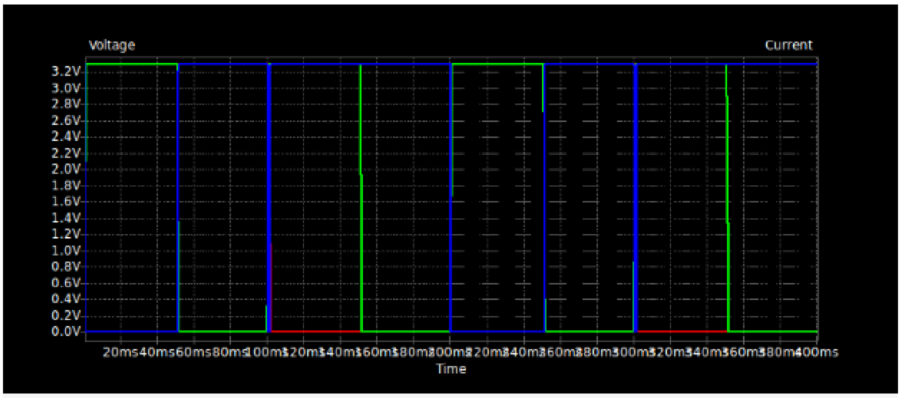
```
.SUBCKT NAND A B Out VDD
* Nodes:
MM1 Out A VDD VDD MPMOS
MM2 Out A 1 1 MNMOS
MM4 Out B VDD VDD MPMOS
MM3 1 B 0 0 MNMOS
.ENDS NAND
```

- Now you are free to use the subcircuit created by you and use it as a component for the required functionality.

CIRCUIT


A NAND gate implemented using MOSFETs
from https://en.wikipedia.org/wiki/CMOS

WAVEFORM


Voltage
Current
3.2V
3.0V
2.8V
2.6V
2.4V
2.2V
2.0V
1.8V
1.6V
1.4V
1.2V
1.0V
0.8V
0.6V
0.4V
0.2V
0.0V
Time

Chapter 5

DIGITAL CIRCUITS

5.1 DIGITAL CIRCUITS

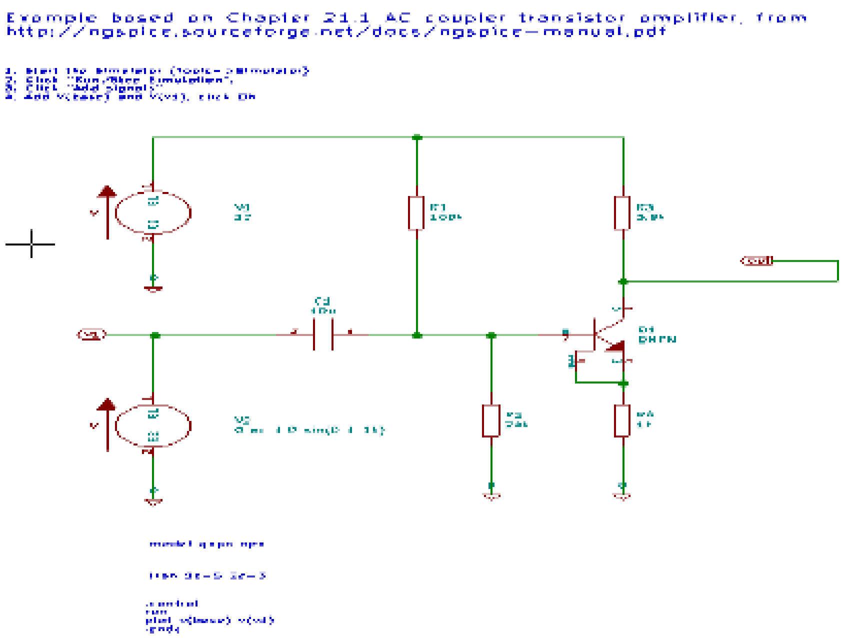
5.1.1 Transient Amplifier

Problem Statement: Plot the input and output waveforms of transient amplifier and verify the same using transient analysis

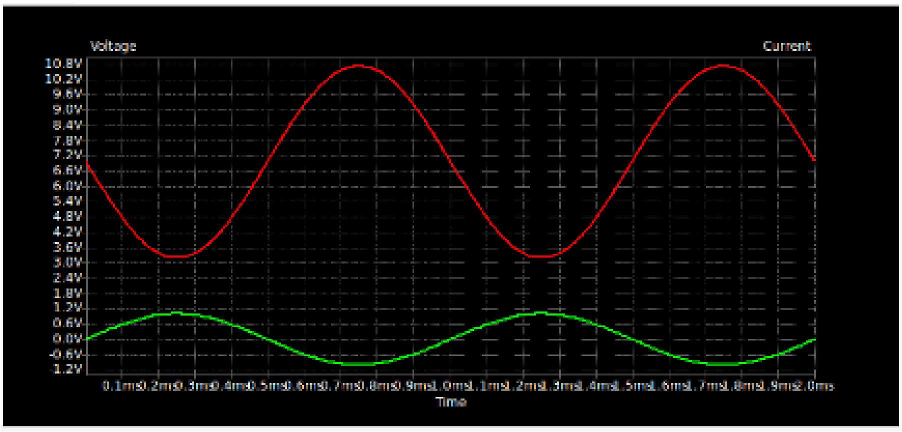
SOLUTION:

- Create a circuit as shown in the diagram.
- Use basic components and there is no need for subcircuit in this circuit.
- Give transient analysis and simulate the circuit

CIRCUIT:



WAVEFORM



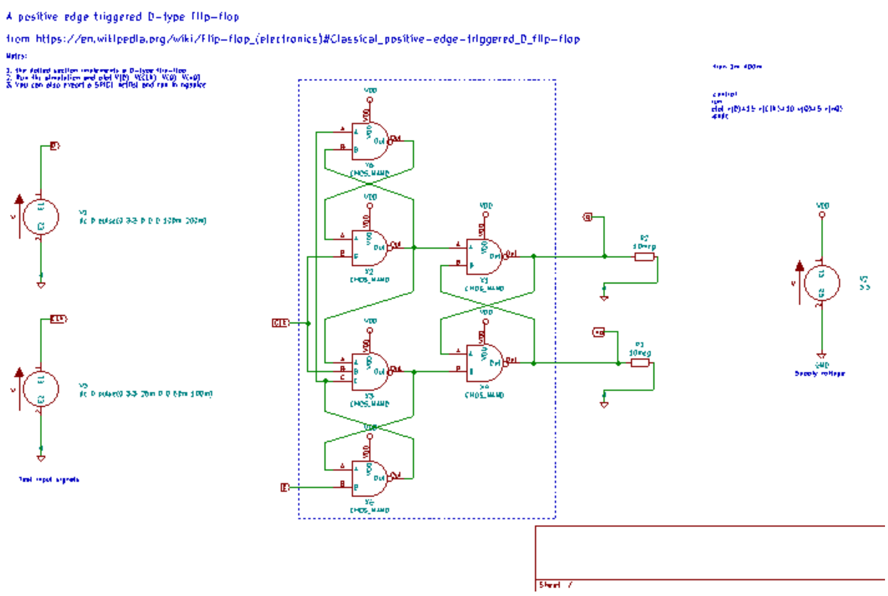
5.1.2 D Flip Flop

Problem Statement: Plot the input and output waveforms of D Flip Flop and verify the same using Transient Analysis

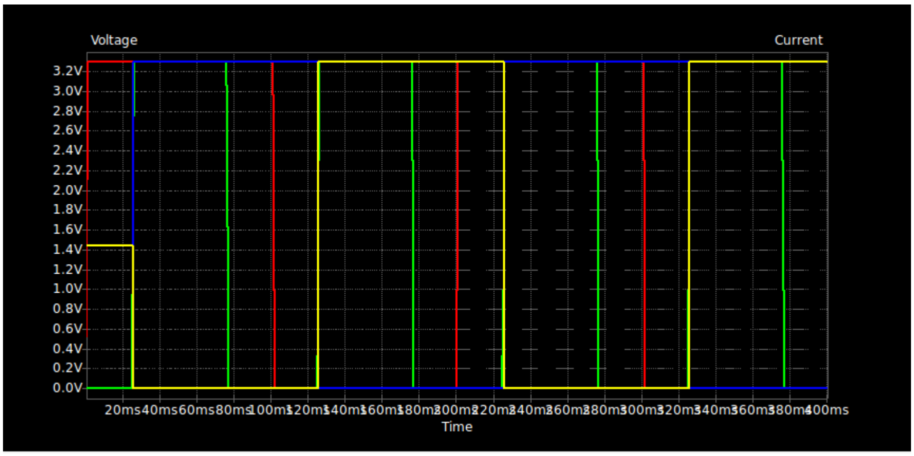
Solution:

- Create a schematic in Kicad as shown in the diagram
- Use the subcircuit Builder method as mentioned in section 4.1.1 to build cmos_nand gates and add it to your component library
- Make sure to use correct pins in building subcircuit
- Add library file to each and every nand gate
- Recommended to use pwl for clock instead of pulse.
- Provide transient analysis and run simulation

Circuit



Waveform



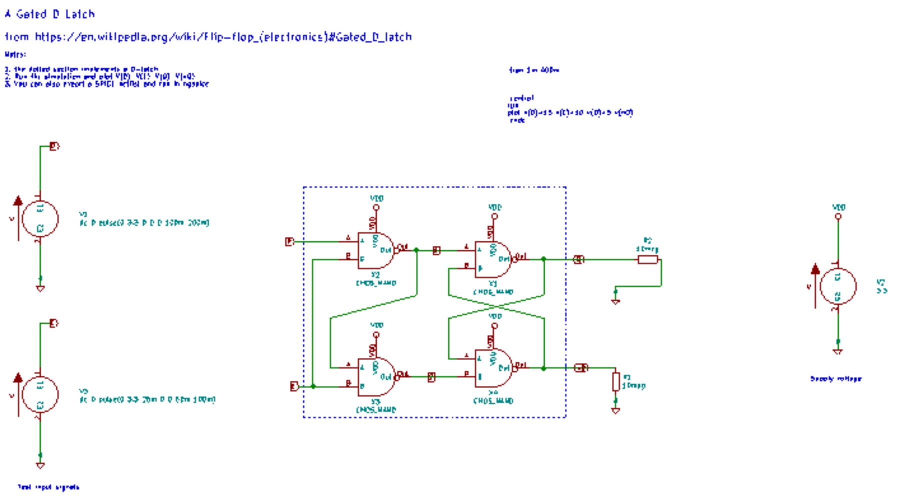
5.1.3 D Latch

Problem Statement: Plot the input and output waveform of D Latch and verify the same using transient analysis

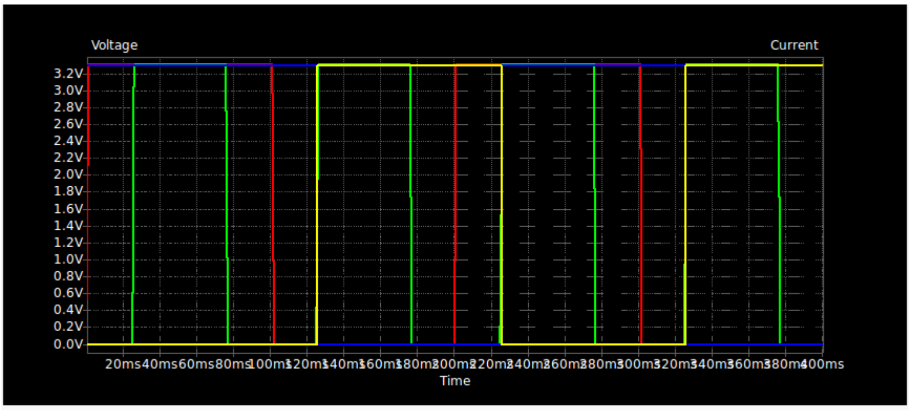
Solution:

- Create a schematic for D Latch as shown in the diagram.
- Use the subcircuit builder method to build cmos nand gate and add it to you components library
- Make sure pins are correct and provide correct sources.
- Give transient analysis and run the simulation

Circuit



Waveform



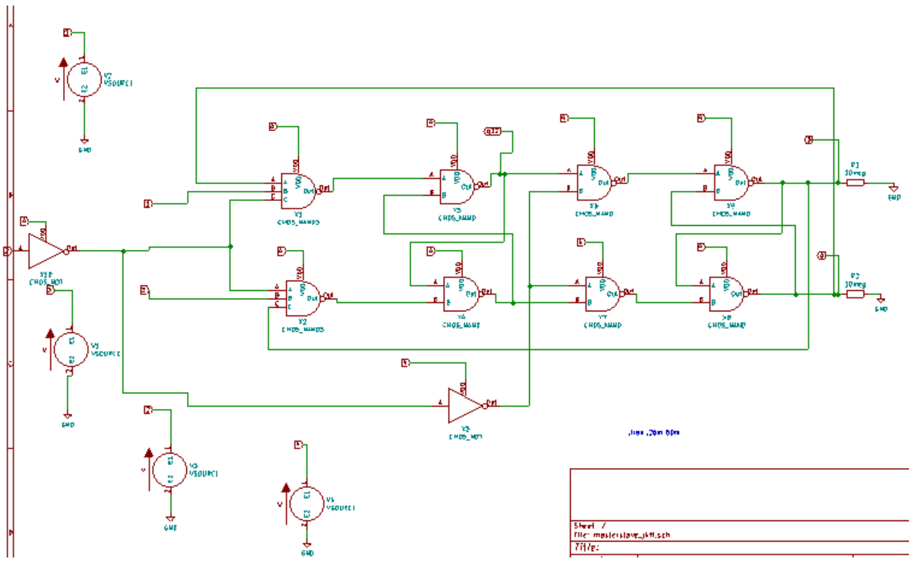
5.1.4 Masterslave JK Flip Flop

Problem Statement: Plot the input and output waveforms of mastersave JK flip flop and verify the same using transient analysis

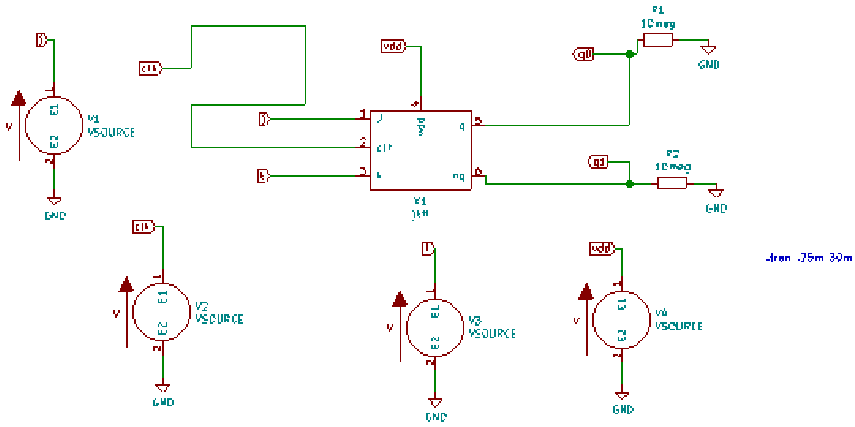
Solution

- Create the schematic as shown in the diagram
- Use subcircuit builder method to buid and add cmos_nand gate.
- Make sure the numbering of pins is correct
- Note that I have made a positive edge triggered JK Flip Flop by adding not gate in the beginning
- Provide transient analysis and run the simulation

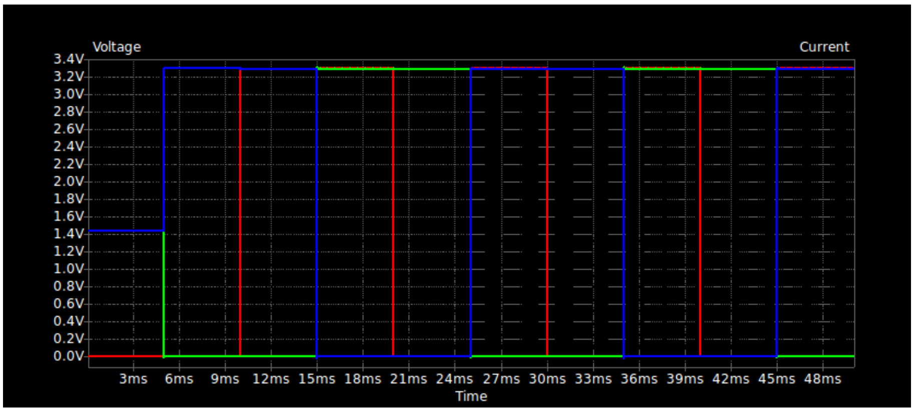
Circuit



Subcircuit



Waveform



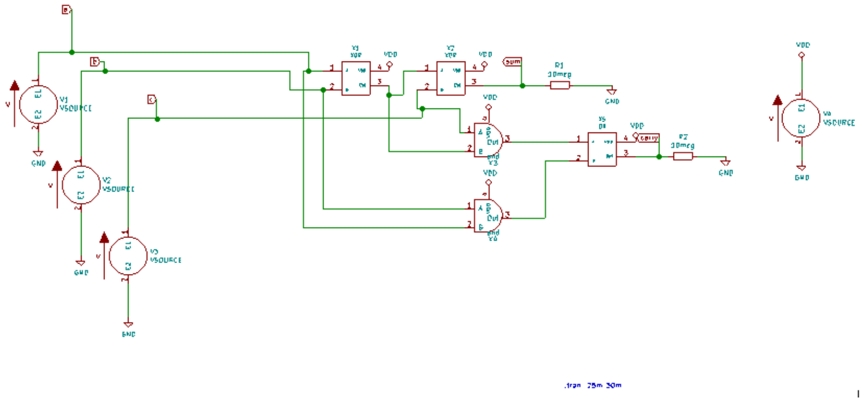
5.1.5 Full Adder

Problem Statement: Plot the input and out waveforms for Full adder and verify the same using traansient analysis

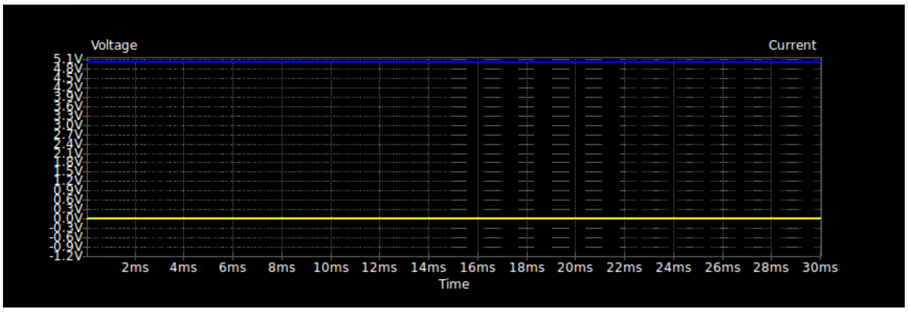
Solution:

- Create the schematic as shown in the diagram
- You must first create subcircuit for and gate and xor gate using subcircuit builder method
- Note the output might have little fluctuations,you must ignore this.
- Provide transient analysis and run the simulation.

Circuit



Waveform



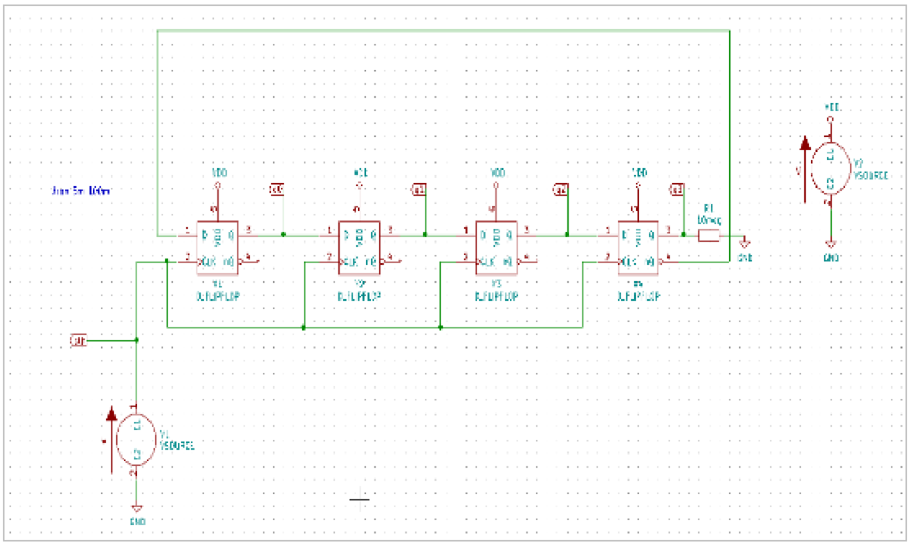
5.1.6 Johnson Counter

Problem Statement Plot the input and output waveform of johnson counter and verify the same using transient analysis

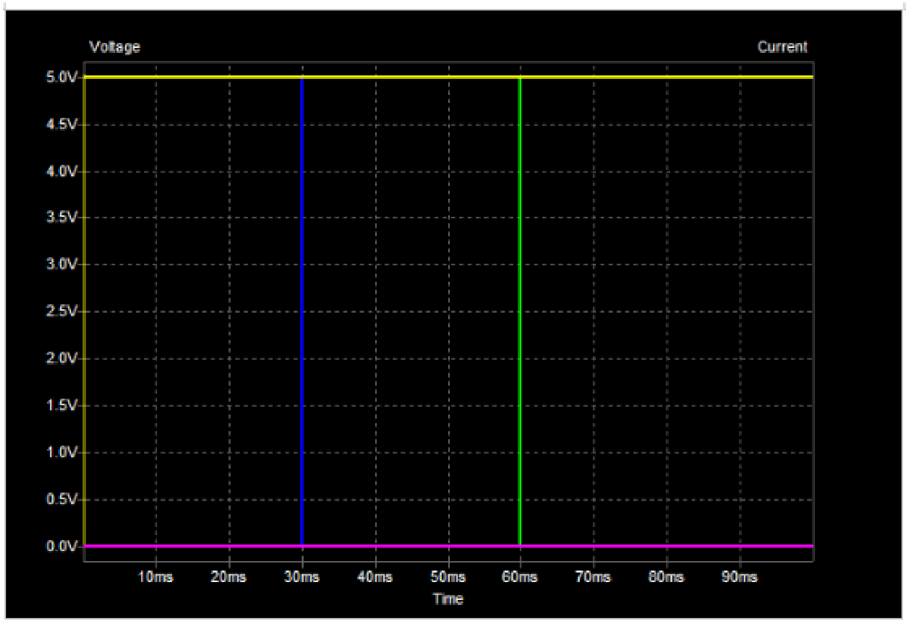
Solution

- Create a schematic in Kicad using the circuit diagram
- Use sub circuit of d-flip flop
- Add .lib file of the flip flop in Edit Sources → Model →Library.
- Do transient analysis upto 100 milliseconds with a time-step of 5ms
- Verify the counter value with the no. of clock pulses

Circuit



Waveform


Voltage
Current
5.0V
4.5V
4.0V
3.5V
3.0V
2.5V
2.0V
1.5V
1.0V
0.5V
0.0V
10ms
20ms
30ms
40ms
50ms
60ms
70ms
80ms
90ms
Time

Chapter 6

UPLOADED PROJECTS

6.1 UPLOADED PROJECTS

- All projects given in analog and digital circuits and more has been uploaded in `https://github.com/FOSSEE/eSIm-Kicad-Simulations`
- Kindly go through these examples for better understanding of kicad simulations,adding and creating netlists and building subcircuits
- Libs folder in both analog and digital circuits contains the list of netlists for each model used in the simulations
- In digital circuits sim_logic.lib contains the list of subckts made.You can add these files to your component library for using in your digital circuits.
- The library file sim_model.lib contains the list of netlists for digital circuits.

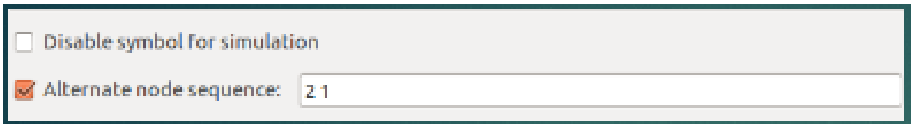
Chapter 7

Problems faced and their solutions

7.1 Problems faced and their solutions

Problem:Circuits consisting of diodes were not giving correct waveforms.

Solution:Since diode is a basic component there is no need of a .lib file added to it.However we must interchange the position of the pins for the diode to work correctly.For Zener diode we must both interchange position and add netlist



Problem:Some circuits involving transistors,op amps,zener diodes were not producing correct waveforms.

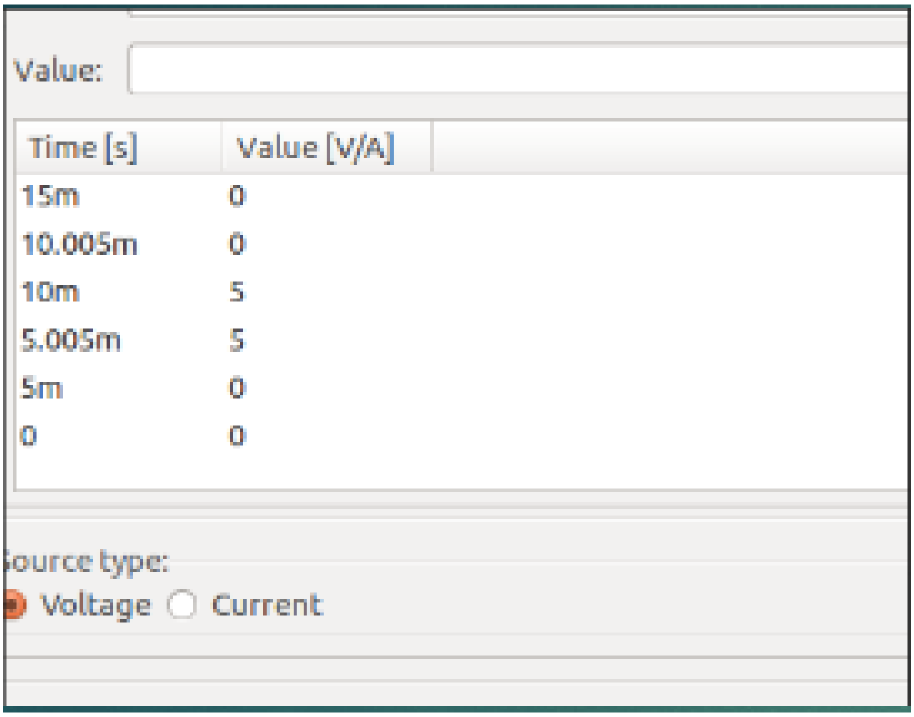
Solution:Add the corresponding netlist for each of these devices.You may have to build the netlist if it is not available or use subcircuit builder method.

Problem:We were not aware on how to give value for pwl sources

Solution:We have to add a delay whenever there is a transition

Problem:Digital circuits were not working properly

Solution:Use subcircuit builder method


Value:
Time [s]
Value [V/A]
15m 0
10.005m 0
10m 5
5.005m 5
5m 0
0 0
Source type:
Voltage
Current

Chapter 8

Conclusion

8.1 Conclusion

- We have simulated analog and digital circuits using Kicad nightly builds and have uploaded on Github
- We have also made subciruits and libraries for many components.

Reference

- http://docs.kicad-pcb.org/stable/en/getting_started_in_kicad.pdf
- https://github.com/KiCad/kicad-source-mirror
- https://github.com/FOSSEE/eSIm-Kicad-Simulations
- https://forum.kicad.info/
- http://www.ecircuitcenter.com/Basics.htm
- http://www.ecircuitcenter.com/SPICEsummary.htm