## Electronic Circuit Simulation using eSim

Kavya Manohar

August 7, 2016

# Copyright ©2015

This work is licensed under a Creative Commons Attribution-Share Alike 4.0 India License. See http://creativecommons.org/licenses/by-sa/4.0/ for more details.

# Acknowledgemnt

Thanks to eSim by FOSSEE team. eSim wraps up the open source electronic design automation software Kicad and ngspice into a single package.

Thanks to every collegue at Aryanet Institute of Technology, Palakkad for their support in this venture.

Thanks in advance to every reader finding this work useful. Your feed back is most welcome.

Kavya Manohar

## Preface

This is a quick guide to design and simulate electronic circuits using open source EDA tool- eSim.

eSim is an open source EDA tool for circuit design, simulation, analysis and PCB design, developed by FOSSEE team (http://fossee.in/) under MHRD based at IIT Bombay. It is an integrated tool built using open source softwares Kicad and (http://www.kicad-pcd.org) Ngspice (http://ngspice. sourcefouge.net/). It is released under GNU GPL License. It runs on Ubuntu Linux, Windows.

This guide provides solutions to specific simulation problems using eSim. Experimental procedures are explained with screen shots. This has been prepared as a part of lab migration project supported by FOSSEE aimed at migrating the labs in educational institutions to free and open source softwares. The source code of every experiment described in this guide will be released by FOSSEE under Creative Commons Attribution-ShareAlike 4.0 International License. You may use eSim manual (http://esim.fossee.in/resource/book/esimusermanual.pdf) for further reference.

Kavya Manohar

# Contents

1	INTRODUCTION TO eSim	13
2	DIODE CHARACTERISTICS	15
3	RC FILTERS	<b>25</b>
4	CLIPPING CIRCUIT	33
5	CLAMPING CIRCUIT	41
6	HALF WAVE RECTIFIERS	<b>47</b>
7	FULL WAVE RECTIFIERS	55
8	JFET CHARACTERISTICS	63
9	MOSFET CHARACTERISTICS	<b>71</b>
10	BJT COMMON EMITTER CHARACTERISTICS	79
11	BJT COMMON BASE CHARACTERISTICS	85
12	ZENER REGULATOR WITH SERIES PASS TRANSISTOR	91

CONTENTS

# List of Figures

2.1	Schematic diagram for diode characteristics	16
2.2	Launching eSim will take you to this window	16
2.3	Creating new project	17
2.4	Creating new schematic diagram	17
2.5	The Kicad Eeschema page	18
2.6	Place component icon	18
2.7	Place wire icon	18
2.8	The Kicad Libraries of components	19
2.9	Editing the value field of component R	19
2.10	Choose annotate from the toop tool bar	20
2.11	Annotation	21
2.12	Netlist Generation	21
2.13	Choose Kicad to Ngspice tool	21
2.14	Choose DC analysis type and enter the values	21
2.15	Choose the required diode model	22
2.16	The characteristics of Diode	23
3.1	Schematic diagram for passive RC high pass filter	25
3.2	Schematic diagram for passive RC low pass filter	26
3.3	Editing the value field of component R	27
3.4	Choose annotate from the toop tool bar	28
3.5	Annotation	29
3.6	Netlist Generation	29
3.7	Choose Kicad to Ngspice tool	29
3.8	Choose AC analysis type and enter the values	30
3.9	The frequency response of RC highpass filter	31
3.10	The frequency response of RC low pass filter	31
4.1	Schematic diagram for series clipper circuit	34
4.2	Schematic diagram for shunt clipper circuit	35
4.3	Annotation	36
4.4	Netlist Generation	36
		00
4.5	Choose analysis type as 'transient' and enter the values	36

4.7	The transient response of the series clipper on python plotting window	38
4.8	The transient response of the series clipper in ngspice plotting window	39
4.9	The transient response of the shunt clipper on python plotting	00
	window	39
5.1	Schematic diagram for clamper circuit	42
5.2	Annotation	43
5.3	Netlist Generation	43
5.4	Choose analysis type as 'transient' and enter the values	44
5.5	Enter the parameters of 'Sine' source	44
5.6	The transient response of the clamper on python plotting window	45
6.1	Schematic diagram for half wave rectifier with C filter	48
6.2	Schematic diagram for half wave rectifier with LC filter	49
6.3	Annotation	50
6.4	Netlist Generation	50
6.5	Choose analysis type as 'transient' and enter the values	50
6.6	Enter the parameters of 'Sine' source	51
6.7	The transient response of the half wave rectifier with capacitor filter on python plotting window	59
6.8	The transient response of the half wave rectifier with inductor	52
0.0	and capacitor filter on python plotting window	53
7.1	Schematic diagram for bridge rectifier with C filter	55
7.2	Schematic diagram for bridge rectifier with LC filter	56
7.3	Annotation	58
7.4	Netlist Generation	58
7.5	Choose analysis type as 'transient' and enter the values	58
7.6	Enter the parameters of 'Sine' source	59
7.7	The transient response of the bridge rectifier with capacitor filter	60
7.0	The transient area as a fithe heider metiling with independent	00
1.8	The transient response of the bridge rectiner with inductor and	61
	capacitor inter on python plotting window	01
8.1	Schematic diagram for JFET characteristics	64
8.2	Annotation	65
8.3	Netlist Generation	65
8.4	Choose Kicad to Ngspice tool	66
8.5	Choose DC analysis type and enter the values of V2	66
8.6	Choose DC analysis type and enter the values of V1 $\ldots$ .	67
8.7	Enter the details of fixed source V1	67
8.8	Enter the details of fixed source V2	68
8.9	Choose the required JFET model	68
8.10	The drain characteristics of JFET with gate voltage $=0V$	69

### LIST OF FIGURES

8.11 8.12	The drain characteristics of JFET with gate voltage $=1V$ The transfer characteristics of JFET with drain voltage $=3V$	69 70
91	Schematic diagram for MOSFET characteristics	72
9.2	Annotation	73
9.3	Netlist Generation	73
9.4	Choose Kicad to Ngspice tool	74
9.5	Choose DC analysis type and enter the values of V2	74
9.6	Choose DC analysis type and enter the values of V1	75
9.7	Enter the details of fixed source V1	75
9.8	Enter the details of fixed source V2	76
9.9	Choose the required MOSFET model	76
9.10	The drain characteristics of MOSFET with gate voltage $=3V$ .	77
9.11	The transfer characteristics of MOSFET with drain voltage $=10V$	78
10.1	Schematic diagram for CE output characteristics	81
10.2	Annotation	82
10.3	Netlist Generation	82
10.4	Choose Kicad to Ngspice tool	82
10.5	Choose DC analysis type and enter the values of V1 and I1 $\ldots$	83
10.6	GiveSource Details of V1 and I1	83
10.7	Choose the required NPN model	84
10.8	The output characteristics of NPN transistor	84
11.1	Schematic diagram for CE output characteristics	87
11.2	Annotation	88
11.3	Netlist Generation	88
11.4	Choose Kicad to Ngspice tool	88
11.5	Choose DC analysis type and enter the values of V1 and I1 $\ldots$	89
11.6	Give Source Details of V1 and I1	89
11.7	Choose the required NPN model	90
11.8	The output characteristics of NPN transistor	90
12.1	Launching eSim will take you to this window	92
12.2	Creating new project	92
12.3	Creating new schematic diagram	93
12.4	The Kicad Eeschema page	93
12.5	Place component icon	94
12.6	Place wire icon	94
12.7	The Kicad Libraries of components	94
12.8	Editing the value field of component R	95
12.9	Choose annotate from the toop tool bar	95
12.10	JSchematic diagram for Zener Diode Regulator	96
12.1	IAnnotation	97
12.12	Zivetlist Generation	97
12.13	3Choose Kicad to Ngspice tool	97

12.14Choose DC analysis type and enter the values	98
12.15Choose ngspice model values	98
12.16Choose the required Transistor model	99
12.17The line regulation characteristics of zener diode	100

### Chapter 1

# INTRODUCTION TO eSim

eSim is a CAD tool that helps electronic system designers to design, test and analyse their circuits. But the important feature of this tool is that it is open source and hence the user can modify the source as per his/her need. The software provides a generic, modular and extensible platform for experiment with electronic circuits. This software runs on all Ubuntu Linux distributions and some flavours of Windows. It uses Python, KiCad and Ngspice.

The objective behind the development of eSim is to provide an open source EDA solution for electronics and electrical engineers. The software should be capable of performing schematic creation, PCB design and circuit simulation (analog, digital and mixed signal). It should provide facilities to create new models and components. The architecture of eSim has been designed by keeping these objectives in mind [1].

### WORKFLOW OF eSim

This section describes the workflow of eSim for circuit simulation. The flow would be slightly different for PCB designing. For a generic treatment on the workflow, please refer to the user manual .

The first step in circuit simulation is to draw a schematic diagram of the circuit. In eSim it is drawn using Eeschema. The next step is to obtain a netlist file. The schematic drawing tool provides this file, which describes the electrical connections between components. The netlist generated by Schematic Editor cannot be directly used for simulation due to compatibility issues. Netlist Converter converts it into Ngspice compatible format. The type of simulation to be performed and the corresponding options are provided through a graphical

user interface (GUI). This is called KiCad to Ngspice Converter in eSim. eSim uses Ngspice for analog, digital, mixed-level/mixed-signal circuit simulation.

To summarise the simulation workflow:

- 1. Strat a new project in the eSim GUI.
- 2. Draw the schematic diagram of the circuit in Eeschema.
- 3. Convert it to netlist for simulation using Eeshema.
- 4. Come back to the eSim GUI and convert the netlist for use by Ngspice using KiCad to Ngspice Converter. Give details of the type of simulations during this step.
- 5. Simulate the circuit and plot the required variables.

Details on every step given above is avilable in the esim usermanual[1].

## Chapter 2

# DIODE CHARACTERISTICS

### AIM

To design and implement a circuit to simulate the V-I characteristics of a diode.

### DESIGN AND CIRCUIT DIAGRAM

Inorder to draw the diode characteristics, we have to use a DC source of voltage. Its value may be varied during simulation. The diode in the circuit should be associated with a 'diode model'. A current limiting resistor may also be used in series with the diode and the DC source. The resulting circuit diagram is shown in the Figure 2.1.

### PROCEDURE

### Launch eSim

Launching eSim will take you to the dialog box. It asks for the default workspace. Browse the folders and set the wokspace location. It will end up in the eSim window shown in Figure 12.1.



Figure 2.1: Schematic diagram for diode characteristics



Figure 2.2: Launching eSim will take you to this window

### Create a New Project

The new project is created by clicking the New icon on the menubar. Give the name of the project ,'Diodechar' in the pop up window as shown in Figure 12.2.

٥	eSim 🗸 🔷 🖉
â 🖞 🙆 🕼	fossee
Projects	Wetcome
	edim is developed by the FOSSEE Irean at IT Benkay To troor more about edim, please visit http://noin.fossee.n To discuss more about edim please Date Physice Name 
	elim Stankd Popul Staktist Tone (MPC) Workgaze . Annekaryaktim Workspace

Figure 2.3: Creating new project

### Create the Schematic

To create the schematic, click the very first icon of the left toolbar as shown in the Figure 12.3. This will open KiCad Eeschema.

= 🔊 🙎	S 😧
Ope King	Projects n Schematic
7	
y=f(x)	

Figure 2.4: Creating new schematic diagram

To create a schematic in KiCad, we need to place the required components. See Figure 12.4. Figure 12.5 shows the icon on the right toolbar which opens the component library. After all the required components of the simple RC circuit are placed, wiring is done using the Place Wire option as shown in the Figure 12.6. Scroll up and down for zooming in and out.



Figure 2.5: The Kicad Eeschema page



Figure 2.6: Place component icon

Figure 2.7: Place wire icon

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries. See Figure 12.7.



Figure 2.8: The Kicad Libraries of components

- Choose DC source from eSim\_Sources
- Choose R from eSim\_Devices
- Choose D from eSim\_Devices
- Choose GND from power

Select the resistor and edit its component value to 1k as shown in Figure 12.8.



Figure 2.9: Editing the value field of component R

Wire the components to get the circuit. A global label 'diode' has been added to identify that node whose voltage will be later recorded and plotted.

**Annotating the circuit:** Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the

top toolbar(See Figure 12.9 and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 12.11.



Figure 2.10: Choose annotate from the toop tool bar

Now we have the circuit diagram as shown in Figure 2.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. This is shown in Fig. 5.15. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 12.12. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of the circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 12.13. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.



cbnew OrcadPCB2 CadStar Spice 1	Generate
Options:	Cancel
Prefix references 'U' and 'IC' with 'X' Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	Use default netnam

Figure 2.12: Netlist Generation

Figure 2.11: Annotation



Figure 2.13: Choose Kicad to Ngspice tool

**Analysis:** Choose DC analysis type. Give the values of DC variables as shown in Figure 2.14. Enter the name of your DC source as on the circuit (here v1) and let its value be varied from -15V to +15V with a step of 0.1 V.

Analysis Source Details NgSpice Model Device Modeling Subci	rcuits	
_ DC Analysis		
Enter Source Name	٧١	
Start	-15	Volts or Amperes
Increment	.1	Volts or Amperes
Stop	15	Volts or Amperes
	Operating Point Analysis	

Figure 2.14: Choose DC analysis type and enter the values

Source Details: Leave this empty.

Ngspice Model: No Ngspice model to be given.

**Device Model:** The Diode is a device whose model details must be given for simulation. Let us choose the generic diode model availabe in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/Diode/D.lib. See Figure 2.15.

Analysis	Source Details NgSpice	Model Device Modeling	Subcircuits			
Add library	for Diode d1: d					
/opt/eSim/s	sc/deviceModeILbrary/Diod	eO.lib				Add
						Convert

Figure 2.15: Choose the required diode model

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. See Figure 2.15. Now you are ready to see the simulation results.

#### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the diode characteristics let us use the commands in ngspice plotting window.

We need to plot the value of voltage across the diode Vs the current through it. Since the current through the diode is same as the current through the voltage source, v1 (since both are in series connection) let us use the command:

plot i(v1) vs v(diode)

This would pop up the required characteristics fo the diode as defined in the diode model D.lib. For a differnt diode model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 2.16.



Figure 2.16: The characteristics of Diode

### RESULT

The circuit for plotting the characteristics of diode was implemented and simulated.

## Chapter 3

# **RC FILTERS**

### AIM

To design and implement circuits for passive RC highpass and lowpass filters.

### DESIGN AND CIRCUIT DIAGRAM

Inorder to plot the frequency response of RC highpass and lowpass filters use an AC source whose frequency can be varied during simulation. The AC source of voltage. It is connected across a series connection of resistor and capacitor. The corresponding circuits for RC highpass filter and lowpass filters are shown in figures 3.1 and 3.2 respectively. The cutoff frequency of the filter will be given by

$$f_c = \frac{1}{2\pi RC} \tag{3.1}$$



Figure 3.1: Schematic diagram for passive RC high pass filter



Figure 3.2: Schematic diagram for passive RC low pass filter

### PROCEDURE

The steps to plot the characteristics of RC high pass filter are explained below. Follow the same procedure to obtain the response of RC lowpass filter. Note that these are two separate projects.

#### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.

#### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as RC\_HPF for highpass filter.

### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the simple RC circuit are placed, wiring is done using the Place Wire option. The 'Place Wire' and 'Place Component' tools are available in the left toolbar. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose AC source from eSim\_Sources
- Choose R from eSim\_Devices
- Choose C from eSim\_Devices
- Choose GND from power

Select the resistor and edit its component value to 1k as shown in Figure 3.3. Also edit the value of capacitor as 1  $\mu$  F. You can just type in 1u.



Figure 3.3: Editing the value field of component R

Wire the components to get the circuit. A global label 'Input' and 'Output' has been added to identify that node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbar. See Figure 3.4 and in the subsequent dialogue boxes appearing click ok and finally close. See Figure 3.5. Now we have the circuit diagram as shown in Figure 3.1.



Figure 3.4: Choose annotate from the toop tool bar

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 3.6. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of RC circuit to NgSpice compatiable netlist click on KiCad to Ngspice icon as shown in Figure 3.7. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose AC analysis type and choose Dec scale. Dec sclae allows plotting as in a semilog graph sheet. Give the values of AC variables as shown in Figure 3.8. Enter the name of your AC source as on the circuit (here v1) and let its frequency be varied from 1Hz to 10kHz with 10 points chosen in each deacade interval of frequency.

Source Details: Set amplitude as 5 and phase shift as 0.



cbnew OrcadPCB2 CadStar Spice 1	Generate
Options:	Cancel
Default format     C     Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	Use default netnam



Figure 3.5: Annotation



Figure 3.7: Choose Kicad to Ngspice tool

**Ngspice Model:** No Ngspice model to be given.

**Device Model:** No Device model to be given.

**Subcircuits:** No subcircuits to be given. Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the frequency response characteristics let us use the commands in ngspice plotting window.

	autorigapiec-15								
7						1			
	Analysis So	ource Details	NgSpice Model	Device Modeling	Subcircuits				
Select Analysis Type									
	AC	AC				DC			
	_ AC Analysis								
	Scale		🔾 Lir	Lin Dec			O Oct		
	Start Frequency		1		Hz	~	]		
	Stop Frequency			10		Hz	~	]	
	No. of Points			10					

Figure 3.8: Choose AC analysis type and enter the values

We need to plot two graphs.

- 1. Input voltage value for different frequencies.
- 2. Output voltage value for different frequencies.

To plot these, let us use the command:

```
plot v(Input), v(Output)
```

This would plot the frequency response characteristics of input and output of the RC high pass filter. The resultant characteristics is shown in the Figure 3.9. The red indicates the Input and the blue indicates the output. The charcteristics of RC low pass filter would be as shown in Figure 3.10.

### RESULT

The circuit for plotting the frequency response of filter was implemented and simulated.



Figure 3.9: The frequency response of RC highpass filter



Figure 3.10: The frequency response of RC low pass filter

## Chapter 4

# CLIPPING CIRCUIT

### AIM

To design and implement a circuit for clipping waveforms

### DESIGN AND CIRCUIT DIAGRAM

Inorder to plot the transient response of clipping circuits use a SINE source whose amplitude, frequency, phase etc can be fixed during simulation. The SINE source is connected across a series connection of resistor and diode. The circuits for series clipper (Diode connected in series with the load) and shunt clippers (Diode connected in shunt with the load) are shown in figures 4.1 and 4.2 respectively.

### PROCEDURE

The steps to plot the transient response of a clipper are explained below.

### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.

### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as 'seriesclipper' for the circuit in figure 4.1 and 'shuntclipper' for the circuit in Figure 4.2. Follow the steps explaied below to implement either of them.



Figure 4.1: Schematic diagram for series clipper circuit

#### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the clipper circuit are placed, wiring is done using the Place Wire option. The 'Place Wire' and 'Place Component' tools are available in the left toolbar. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose SINE source from eSim\_Sources
- Choose R from eSim\_Devices
- Choose D from eSim\_Devices
- Choose GND from power

Select the resistor and edit its component value to 1k.

Wire the components to get the circuit. A global label 'in' and 'out' has been added to identify the node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.


Figure 4.2: Schematic diagram for shunt clipper circuit

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbar and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 4.3.

Now we have the circuit diagram as shown in Figure 4.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 4.4. Now the netlist is ready to be simulated.

00 Ar	innotate Schematic	
Scope		
🖲 Us	Ise the entire schematic	
0 Us	ise the current page only	
e e		
	Annotate only the unannotated components on the entire schematic?	
	This operation will change the current annotation and cannot be undone.	
R1 A	Cancel	2
0 Us	ise first free number in schematic	
::::: O St	tart to sheet number*100 and use first free number	
○ St	itart to sheet number*1000 and use first free number	
<b>3</b>	Close Clear Annotation Annotation	





Figure 4.3: Annotation

### KiCad to Ngspice conversion

To convert KiCad netlist of clipper circuit to NgSpice compatible netlist click on KiCad to Ngspice icon . Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose analysis type as 'Transient'. Give the values of time variables as shown in Figure 4.5. Enter the time to be varied from 'Start time=0 ms' to 'Stop time=20ms' with a 'Step time=1 ms'.

Transient Analysis			
Start Time	0	ms	~
Step Time	1	ms	~
Stop Time	20	ms	~
			<b>v</b>
			Convert

Figure 4.5: Choose analysis type as 'transient' and enter the values

Source Details: Set the details of 'sine' source as shown in Figure 4.6.

- Offset value(volts): 0
- Amplitude(volts): 5
- Frequency(Hz): 50

- Delay time(Seconds): 0
- Damping factor(1/seconds):0

Analysis	Source Details NgSpic	ce Model	Device Modeling	Subcircuits	
Add paramet	ers for sine source v1				
Enter offset v	alue (Volts/Amps):				0
Enter amplitu	de (Volts/Amps):				5
Enter frequen	cy(Hz):				50
Enter delay ti	me (seconds):				0
Enter dampin	g factor (1/seconds):				0

Figure 4.6: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

**Device Model:** Add thediode model available in the eSim library by browsing the folder, /opt/eSim/src/deviceModelLibrary/Diode/D.lib

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the transient response of the clipper you can use either plotting types.

**Python plotting:** This provides a graphical interface for plotting. We need to plot the value of voltage across the 'SINE' source as well as the load resistor with respect to time. We have already labled these nodes as in and out respectively The nodes will be listed on the GUI. Choose 'in' and 'out' and click on 'plot' button. See Figure 4.7.



Figure 4.7: The transient response of the series clipper on python plotting window

**Ngspice plotting:** . Time of simulation has already been set in the previous step as 0 ms to 20 ms. Use the commands in ngspice plotting window for obtaining the required plots.

plot v(out), v(in)

This would plot the transient response of input and output of the series clipper. The resultant characteristics is shown in the Figure 4.8.

Repeat the same set of procedures for implementing the shunt clipper. Use the schematic as shown in Figure 4.2. Upon using python plotting, the resultant transient analysis would look like Figure 4.9

### RESULT

The circuit for plotting the transient analysis of clipper was implemented and simulated.



Figure 4.8: The transient response of the series clipper in ng spice plotting window



Figure 4.9: The transient response of the shunt clipper on python plotting window

## Chapter 5

# CLAMPING CIRCUIT

## AIM

To design and implement circuit for clamping waveforms

## DESIGN AND CIRCUIT DIAGRAM

In order to plot the transient response of clamping circuits use a SINE source whose amplitude, frequency, phase etc can be fixed during simulation. The SINE source is connected across a series connection of a diode and a capacitor and the output is taken across the diode and the GND. The circuit for a clamper circuit is in Figure 5.1.

## PROCEDURE

The steps to plot the transint resonse of a clamper are explained below.

### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.

### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as 'clamper' for the circuit in Figure 5.1. Follow the steps explaied below to implement the circuit.



Figure 5.1: Schematic diagram for clamper circuit

### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the clipper circuit are placed, wiring is done using the Place Wire option. The 'Place Wire' and 'Place Component' tools are available in the left toolbar. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose SINE source from eSim\_Sources
- Choose C from eSim\_Devices
- Choose D from eSim\_Devices
- Choose GND from power

Select the capacitor and edit its component value to 1u.

Wire the components to get the circuit. A global label 'in' and 'out' has been added to identify the node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.



cbnew OrcadPCB2 CadStar Spice 1	Generate
Options:	Cancel
Prefix references 'U' and 'IC' with 'X' Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	🗌 Use default netnam

Figure 5.3: Netlist Generation

Figure 5.2: Annotation

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically. Choose annotate button from the top toolbar and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 5.2.

Now we have the circuit diagram as shown in Figure 5.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 5.3. Now the netlist is ready to be simulated.

### KiCad to Ngspice conversion

To convert KiCad netlist of clipper circuit to NgSpice compatible netlist click on KiCad to Ngspice icon . Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models. **Analysis:** Choose analysis type as 'Transient'. Give the values of time variables as shown in Figure 5.4. Enter the time to be varied from 'Start time=0 ms' to 'Stop time=4ms' with a 'Step time=0.01 ms'.

_ Transient Analysis			
Start Time	0	ms	<b>v</b>
Step Time	.01	ms	~
Stop Time	4	ms	~
			Convert

Figure 5.4: Choose analysis type as 'transient' and enter the values

Source Details: Set the details of 'sine' source as shown in Figure 4.6.

- Offset value(volts): 0
- Amplitude(volts): 4
- Frequency(Hz): 1k
- Delay time(Seconds): 0
- Damping factor(1/seconds):0

Analysis Source Details NgSpice Model Device Modeling Subcircuits	
Add parameters for sine source v1	
Enter offset value (Volts/Amps):	0
Enter amplitude (Volts/Amps):	4
Enter frequency (Hz):	1k
Enter delay time (seconds):	0
Enter damping factor (Viseconds):	0
	]
	Convert

Figure 5.5: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

46

**Device Model:** Add the diode model available in the eSim library by browsing the folder, /opt/eSim/src/deviceModelLibrary/Diode/D.lib

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the transient response of the clipper you can use either plotting types.

**Python plotting:** This provides a graphical interface for plotting. We need to plot the value of voltage across the 'SINE' source as well as the diode with respect to time. We have already labled these nodes as in and out respectively The nodes will be listed on the GUI. Choose 'in' and 'out' and click on 'plot' button. See Figure 5.6.



Figure 5.6: The transient response of the clamper on python plotting window

**Ngspice plotting:** . Time of simulation has already been set in the previous step. Use the commands in ngspice plotting window for obtaining the required plots.

plot v(out), v(in)

This would plot the transient response of input and output of the clamper.

## RESULT

The circuit for plotting the transient analysis of clamper was implemented and simulated.

## Chapter 6

# HALF WAVE RECTIFIERS

## AIM

To design and implement circuit for half wave rectifier with C and LC filter.

## DESIGN AND CIRCUIT DIAGRAM

Inorder to plot the transient response of half wave rectifier use a SINE source whose amplitude, frequency, phase etc can be fixed during simulation. The SINE source is connected across a series connection of a diode and a load of resistance  $1k\Omega$ . The output is filtered using a capacitor as shown in Figure 6.1 as well as an LC filter as shown in Figure 6.2. The two circuits can be implemented as separate projects.

### PROCEDURE

The steps to plot the transint resonse of a half wave rectifier with filter are explained below.

### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.



Figure 6.1: Schematic diagram for half wave rectifier with C filter

### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as 'halfwaverectifierC' for the circuit in figure 6.1. Follow the steps explaied below to implement the circuit.

### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the half wave circuit are placed, wiring is done using the Place Wire option. The 'Place Wire' and 'Place Component' tools are available in the left toolbar. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose SINE source from eSim\_Sources
- Choose C from eSim\_Devices
- Choose L from eSim\_Devices
- Choose D from eSim\_Devices



Figure 6.2: Schematic diagram for half wave rectifier with LC filter

• Choose GND from power

Select the capacitor and edit its component value to 1u and edit the component value of inductor to 5.

Wire the components to get the circuit. A global label 'in' and 'out' has been added to identify the node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.

**Annotating the circuit:** Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbar and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 6.3.

Now we have the circuit diagram as shown in Figure 6.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along

💿 💿 Annotate Schematic	
Scope	
Use the entire schematic	
<ul> <li>Use the current page only</li> </ul>	
Annotate only the unannotated components on the schematic?	e entire
This operation will change the current annotation a cannot be undone.	nd .
R1 A Cancel	ж (2)
Use first free number in schematic	
<ul> <li>Start to sheet number*100 and use first free number</li> </ul>	
<ul> <li>Start to sheet number*1000 and use first free number</li> </ul>	
3 Close Clear Annotation Annotation	)



Figure 6.4: Netlist Generation

Figure 6.3: Annotation

with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 6.4. Now the netlist is ready to be simulated.

### KiCad to Ngspice conversion

To convert KiCad netlist of half wave rectifier circuit to NgSpice compatiable netlist click on KiCad to Ngspice icon . Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose analysis type as 'Transient'. Give the values of time variables as shown in Figure 6.5. Enter the time to be varied from 'Start time=0 ms' to 'Stop time=500ms' with a 'Step time=1 ms'.

Analysis Source	Details NgSpic	e Model Device Modeling	Subcircuits			
	Enter Source Nar	ne				^
	Start				Volts or Amperes 🗸 🗸	
	Increment				Volts or Amperes 🗸 🗸	
	Stop				Volts or Amperes 🗸 🗸	
			Operating Poi	nt Analysis		
_ Transient Analysis .						
Start Time			0	ms		~
Step Time			1	ms		~
Stop Time			500	ms		~
						Convert

Figure 6.5: Choose analysis type as 'transient' and enter the values

Source Details: Set the details of 'sine' source as shown in Figure 6.6.

- Offset value(volts): 0
- Amplitude(volts): 4
- Frequency(Hz): 50
- Delay time(Seconds): 0
- Damping factor(1/seconds):0

Analysis	Source Details N	gSpice Model	Device Modeling	Subcircuits								
Add parame	ters for sine source	v1										
Enter offset	Enter offset value (Volts/Amps):											
Enter amplitu	Enter amplitude (Volts/Amps):											
Enter freque	ncy (Hz):										50	
Enter delay t	ime (seconds):										0	
Enter dampir	ng factor (1/seconds)	c									0	
												Convert

Figure 6.6: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

**Device Model:** Add thediode model available in the eSim library by browsing the folder, /opt/eSim/src/deviceModelLibrary/Diode/D.lib

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the transient response of the clipper you can use either plotting types. **Python plotting:** This provides a graphical interface for plotting. We need to plot the value of voltage across the 'SINE' source as well as the load with respect to time. We have already labled these nodes as in and out respectively The nodes will will listed on the GUI. Choose 'in' and 'out' and click on 'plot' button. See Figure 6.7.



Figure 6.7: The transient response of the half wave rectifier with capacitor filter on python plotting window

**Ngspice plotting:** . Time of simulation has already been set in the previous step. Use the commands in ngspice plotting window for obtaining the required plots.

plot v(out), v(in)

This would plot the transient response of input and output of the half wave rectifier.

Repeat the same set of steps for the circuit with LC filter shown in Figure 6.2. The corresponding output is shown in Figure 6.8. It can be seen that the ripple has largely been reduced in Figure 6.8. The change in characteristics of the ripple may be experimentally observed by varying the value of inductor as L=15H, L=0.1H etc. This is left as an exercise for the user.



Figure 6.8: The transient response of the half wave rectifier with inductor and capacitor filter on python plotting window

## RESULT

The circuit for plotting the transient analysis of half wave rectifier was implemented and simulated.

## Chapter 7

# FULL WAVE RECTIFIERS

## AIM

To design and implement circuit for full wave rectifiers with C and LC filter.

## DESIGN AND CIRCUIT DIAGRAM

Inorder to plot the transient response of fullwave bridge rectifier use a SINE source whose amplitude, frequency, phase etc can be fixed during simulation. The SINE source is connected across a bridge combination of diodes and the output is taken across a load of resistance  $1k\Omega$ . The output is filtered using a capacitor as shown in Figure 7.1 as well as an LC filter as shown in Figure 7.2. The two circuits can be implemented as separate projects.



Figure 7.1: Schematic diagram for bridge rectifier with C filter



Figure 7.2: Schematic diagram for bridge rectifier with LC filter

### PROCEDURE

The steps to plot the transient response of a bridge rectifier with filter are explained below.

### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window.

### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as 'bridgerectifierC' for the circuit in figure 7.1. Follow the steps explaied below to implement the circuit.

### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components. After all the required components of the clipper circuit are placed, wiring is done using the Place Wire option. The 'Place Wire' and 'Place Component' tools are available in the left toolbar. Scroll up and down for zooming in and out. **Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose SINE source from eSim\_Sources
- Choose C from eSim\_Devices
- Choose L from eSim\_Devices
- Choose D from eSim\_Devices
- Choose GND from power

Select the capacitor and edit its component value to 1u and edit the component value of inductor to 15H.

Wire the components to get the circuit. A global label 'in' and 'out' has been added to identify the node whose voltage will be later recorded and plotted. Global label is added from the right toolbar of Eeschema.

**Annotating the circuit:** Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbar and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 7.3.

Now we have the circuit diagram as shown in Figure 7.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving. See Figure 7.4. Now the netlist is ready to be simulated.

1111	) 🕘 Annotate Schematic	
s	icope	
	Use the entire schematic	
1111	<ul> <li>Use the current page only</li> </ul>	
	8	
	Annotate only the unannotated components on the schematic?	entire
	This operation will change the current annotation as cannot be undone.	nd
R1	Cancel	K (2)
1111	Ose first free number in schematic	
1111	<ul> <li>Start to sheet number*100 and use first free number</li> </ul>	
	$\bigcirc$ Start to sheet number*1000 and use first free number	
3		)





Figure 7.3: Annotation

### KiCad to Ngspice conversion

To convert KiCad netlist of bridge rectifier circuit to NgSpice compatible netlist click on KiCad to Ngspice icon . Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose analysis type as 'Transient'. Give the values of time variables as shown in Figure 7.5. Enter the time to be varied from 'Start time=0 ms' to 'Stop time=500ms' with a 'Step time=1 ms'.

alysis Source Details NgSpice Model Device Model	ng Subcircuits		
Enter Source Name			^
Start		Volts or Amperes 🗸 🗸	
Increment		Volts or Amperes 🗸	
Stop		Volts or Amperes 🗸 🗸	
	Operating Point Analysis		
ansient Analysis			
art Time	0 ms		~)
p Time	1 ms		~
ıp Time	500 ms		~

Figure 7.5: Choose analysis type as 'transient' and enter the values

Source Details: Set the details of 'sine' source as shown in Figure 7.6.

- Offset value(volts): 0
- Amplitude(volts): 4
- Frequency(Hz): 50

- Delay time(Seconds): 0
- Damping factor(1/seconds):0

Analysis	Source Details	NgSpice Model	Device Modeling	Subcircuits						
Add param	eters for sine sour	ce v1								
Enter offse	t value (Volts/Amps	):							0	
Enter ampli	tude (Volts/Amps):								4	
Enter frequ	ency (Hz):								50	
Enter delay	time (seconds):								0	
Enter damp	ing factor (1/secon	ds):							0	
										Convert

Figure 7.6: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

**Device Model:** Add thediode model available in the eSim library by browsing the folder, /opt/eSim/src/deviceModelLibrary/Diode/D.lib

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the transient response of the clipper you can use either plotting types.

**Python plotting:** This provides a graphical interface for plotting. We need to plot the value of voltage across the 'SINE' source as well as the load with respect to time. We have already labled these nodes as in and out respectively The nodes will will listed on the GUI. Choose 'in' and 'out' and click on 'plot' button. See Figure 7.7.



Figure 7.7: The transient response of the bridge rectifier with capacitor filter on python plotting window

**Ngspice plotting:** . Time of simulation has already been set in the previous step. Use the commands in ngspice plotting window for obtaining the required plots.

plot v(out), v(in)

This would plot the transient response of input and output of the bridge rectifier.

Repeat the same set of steps for the circuit with LC filter shown in Figure 7.2. The corresponding output is shown in Figure 7.8. It can be seen that the ripple has largely been reduced in Figure 7.8. The change in characteristics of the ripple may be experimentally observed by varying the value of inductor as L=5H, L=0.1H etc. This is left as an exercise for the user.

## RESULT

The circuit for plotting the transient analysis of clamper was implemented and simulated.



Figure 7.8: The transient response of the bridge rectifier with inductor and capacitor filter on python plotting window

## Chapter 8

# JFET CHARACTERISTICS

### AIM

To design and implement a circuit for simulating the drain and transfer characteristics of a JFET.

### DESIGN AND CIRCUIT DIAGRAM

Inorder to draw the JFET characteristics, we have to use a DC source of voltage which may be varied during simulation. The JFET in the circuit should be associated with a coresponding 'JFET model' during simulations. The resulting circuit diagram is shown in the Figure 8.1.

Drain charcteristics is a plot between the drain current and drain to source voltage keeping the gate voltage constant. Transfer charcteristics is a plot between the drain current and gate to source voltage keeping the drain voltage constant.

### PROCEDURE

#### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window

### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window.



Figure 8.1: Schematic diagram for JFET characteristics

#### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components.

Clicking on the icon on the right toolbar opens the component library. After all the required components of the circuit are placed, wiring is done using the Place Wire option. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose DC sources from eSim\_Sources
- Choose NJF from eSim\_Devices
- Choose GND from power

Wire the components to get the circuit. A global labels 'GATE' and 'DRAIN' hav been added to identify those nodes whose voltage will be later recorded and plotted.



	(3)
conew OrcadPCB2 CadStar Spice (1)	Generate
Options:	Cancel
Prefix references 'U' and 'IC' with 'X'  Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	🗌 Use default netnam
fault Netlist Filename:	

Figure 8.3: Netlist Generation

Figure 8.2: Annotation

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbarand in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 8.2.

Now we have the circuit diagram as shown in Figure 8.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 8.3. Now the netlist is ready to be simulated.

### KiCad to Ngspice conversion

To convert KiCad netlist of JFET circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 8.4. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.



Figure 8.4: Choose Kicad to Ngspice tool

**Analysis:** Choose DC analysis type. On the same netlist you can simulate the drain characteristics as well as transfer charcteristics. Choose the values of two DC sources, V1 and V2 in the netlist properly as described below. Follow the procedures for drain characteristics first. After obtaining the required plots do the procedures for the transfer characteristics and obtain the required charcteristics curves.

- Drain Charcteristics: Give the values of DC variables as shown in Figure 8.5. Enter the name of your DC source V2 and let its value be varied from 0V to 30V with a step of 0.1 V.
- Transfer Charcteristics: Give the values of DC variables as shown in Figure 8.6. Enter the name of your DC source V1 and let its value be varied from 0V to 4V with a step of 0.1 V.

v2	
0	Volts or Amperes 🗸 🗸
.1	Volts or Amperes 🛛 🗸
30	Volts or Amperes 🗸
	v2 0 .1 30

Figure 8.5: Choose DC analysis type and enter the values of V2

### Source Details:

• Drain Charcteristics: Give the value of DC variables as shown in Figure 8.7. Leave the column of V2 blank. Give the value of V1 as 0V, which

lo.of Points		
DC Analysis		
Enter Source Name	vl	
Start	0	Volts or Amperes
Increment	.1	Volts or Amperes 🗸 🗸
Stop	4	Volts or Amperes 🗸 🗸

Figure 8.6: Choose DC analysis type and enter the values of V1

is the gate voltage. (You may repeat the experiment by varying the gate voltage as V1=1V, V1=2V etc.)

Analysis Source Detail	s NgSpice Model	Device Modeling	Subcircuits		
Add parameters for DC	source vl				
Enter value(Volts/Amps	):				0
Add parameters for DC	source v2				
Enter value(Volts/Amps	):				

Figure 8.7: Enter the details of fixed source V1

• Transfer Charcteristics: Give the value of DC variables as shown in Figure 8.8. Leave the column of V1 blank. Give the value of V2 as 3V, which is the drain voltage.

Ngspice Model: No Ngspice model to be given.

**Device Model:** The JFET is a device whose model details must be given for simulation. Let us choose the generic N-chnnel JFET model availabe in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/JFET/NJF.lib. See Figure 8.9.

Subcircuits: No subcircuits to be given.

Analysis	Source Details	NgSpice Model	Device Modeling	Subcircuits				
Add par	ameters for DC sc	ource v1						7
Enter va	lue(Volts/Amps):							
Add par	ameters for DC sc	ource v2				 		
Enter va	lue(Volts/Amps):						5	



Analysis	Source Details	NgSpice Model	Device Modeling	Subcircuits			
_ Add libra	ry for JFET j1 : njf						
/opt/eSi	m/src/deviceMode	lLibrary/JFET/NJF.lil	)				Add
					 	 	)

Figure 8.9: Choose the required JFET model

Once these details are provided click on convert button. Now you are ready to see the simulation results.

### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the JFET characteristics let us use the commands in ngspice plotting window. We need to plot the drain charcteristics as well as transfer characteristics.

**Drain Characteritics:** In the ngspice plotting window, type the following command:

plot -i(v2) vs v(drain)

This would pop up the drain characteristics of the JFET as defined in the JFET model NJF.lib. For a different device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 8.10 and 8.11.



Figure 8.10: The drain characteristics of JFET with gate voltage =0V



Figure 8.11: The drain characteristics of JFET with gate voltage =1V

**Transfer Characteritics:** In the ngspice plotting window, type the following command:

plot -i(v2) vs v(gate)

This would pop up the transfer characteristics of the JFET as defined in the JFET model NJF.lib. For a different device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 8.12.



Figure 8.12: The transfer characteristics of JFET with drain voltage =3V

## RESULT

The circuit for plotting the characteristics of JFET was implemented and simulated.
## Chapter 9

# MOSFET CHARACTERISTICS

#### AIM

To design and implement a circuit for simulating the drain and transfer characteristics of a MOSFET.

#### DESIGN AND CIRCUIT DIAGRAM

Inorder to draw the MOSFET characteristics, we have to use a DC source of voltage which may be varied during simulation. The MOSFET in the circuit should be associated with a coresponding 'MOSFET model' during simulations. The resulting circuit diagram is shown in the Figure 9.1.

Drain charcteristics is a plot between the drain current and drain to source voltage keeping the gate voltage constant. Transfer charcteristics is a plot between the drain current and gate to source voltage keeping the drain voltage constant.

#### PROCEDURE

#### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window

#### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window.



Figure 9.1: Schematic diagram for MOSFET characteristics

#### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components.

Clicking on the icon on the right toolbar opens the component library. After all the required components of the circuit are placed, wiring is done using the Place Wire option. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose DC sources from eSim\_Sources
- Choose resistors from eSim\_Devices
- Choose MOS\_N from eSim\_Devices
- Choose GND from power

Wire the components to get the circuit. A global labels 'GATE' and 'DRAIN' hav been added to identify those nodes whose voltage will be later recorded and plotted.



	(3)
conew OrcadPCB2 CadStar Spice (1)	Generate
Options:	Cancel
Prefix references 'U' and 'IC' with 'X' Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	🗌 Use default netnam
fault Netlist Filename:	
Il Adder.cir	

Figure 9.3: Netlist Generation

Figure 9.2: Annotation

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbarand in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 9.2.

Now we have the circuit diagram as shown in Figure 9.1.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 9.3. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of MOSFET circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 9.4. 8Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.



Figure 9.4: Choose Kicad to Ngspice tool

**Analysis:** Choose DC analysis type. On the same netlist you can simulate the drain characteristics as well as transfer charcteristics. Choose the values of two DC sources, V1 and V2 in the netlist properly as described below. Follow the procedures for drain characteristics first. After obtaining the required plots do the procedures for the transfer characteristics and obtain the required charcteristics curves.

- Drain Charcteristics: Give the values of DC variables as shown in Figure 9.5. Enter the name of your DC source V2 and let its value be varied from 0V to 20V with a step of 0.1 V.
- Transfer Charcteristics: Give the values of DC variables as shown in Figure 9.6. Enter the name of your DC source V1 and let its value be varied from 0V to 4V with a step of 0.1 V.

Analysis	Source Details NgSpice Mon	lel Device Modeling	Subcircuits		
_ DC Analysi	is				
	Enter Source Name		V2		
Start		0	Volts or Amperes	~	
Increment		.1	Volts or Amperes	~	
Stop		20	Volts or Amperes	~	
			Operating Poi	t Analysis	

Figure 9.5: Choose DC analysis type and enter the values of V2

lo.of Points		
DC Analysis		
Enter Source Name	vl	
Start	0	Volts or Amperes
Increment	.1	Volts or Amperes 🗸 🗸
Stop	4	Volts or Amperes 🗸

Figure 9.6: Choose DC analysis type and enter the values of V1

#### Source Details:

• Drain Charcteristics: Give the value of DC variables as shown in Figure 9.7. Leave the column of V2 blank. Give the value of V1 as 3V, which is the gate voltage. (You may repeat the experiment by varying the gate voltage as V1=4V, V1=6V etc.)

ſ	Analysis	Source Details	NgSpice Model	Device Modeling	Subcircuits	
	Add param	eters for DC sourc	ce v1			
	Enter value(	Volts/Amps):				3
	Add param	eters for DC sourc	:e v2			
	Enter value(	Volts/Amps):				

Figure 9.7: Enter the details of fixed source V1

• Transfer Charcteristics: Give the value of DC variables as shown in Figure 9.8. Leave the column of V1 blank. Give the value of V2 as 10V, which is the drain voltage.

Ngspice Model: No Ngspice model to be given.

**Device Model:** The MOSFET is a device whose model details must be given for simulation. Let us choose the generic N-chnnel MOSFET model availabe in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/MOS/NMOS-5um.lib. See Figure 9.9.

Analysis	Source Details	NgSpice Model	Device Modeling	Subcircuits	;				
Add para	ameters for DC sc	ource v1							]
Enter va	lue(Volts/Amps):								
_ Add para	ameters for DC sc	ource v2				 	 		
Enter va	lue(Volts/Amps):							5	
									7



Analysis Source Details NgSpice Model D	evice Modeling Subcircuits
Add library for MOSFET m1: mos_n	
	/opt/eSim/src/deviceModeLibrary/MOS/NMOS/Sum lib
Enter width of MOSFET m1(default=100u):	
Enter length of MOSFET m1(default=100u):	
Enter multiplicative factor of MOSFET m1(default=1):	

Figure 9.9: Choose the required MOSFET model

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

#### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the MOSFET characteristics let us use the commands in ngspice plotting window. We need to plot the drain charcteristics as well as transfer characteristics. **Drain Characteritics:** In the ngspice plotting window, type the following command:

plot -i(v2) vs v(drain)

This would pop up the drain characteristics of the MOSFET as defined in the MOSFET model NMOS-5um.lib. For a different device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.10.



Figure 9.10: The drain characteristics of MOSFET with gate voltage =3V

**Transfer Characteritics:** In the ngspice plotting window, type the following command:

plot -i(v2) vs v(gate)

This would pop up the transfer characteristics of the MOSFET as defined in the MOSFET model NMOS-5um.lib. For a different device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.11.

### RESULT

The circuit for plotting the characteristics of MOSFET was implemented and simulated.



Figure 9.11: The transfer characteristics of MOSFET with drain voltage  ${=}10\mathrm{V}$ 

## Chapter 10

# BJT COMMON EMITTER CHARACTERISTICS

#### AIM

To design and implement a circuit for simulating the output characteristics of a NPN Bipolar Junction Transistor.

#### DESIGN AND CIRCUIT DIAGRAM

In common emitter configuration the emitter terminal is grounded. Input characteristics is the plot between the base current  $i_b$  and base -emitter volatage  $V_{be}$ , keeping the collector voltage constant. Output characteristics is the plot between the collector current  $I_c$  and the collector - emitter volatage  $V_{ce}$ 

Inorder to draw the BJT CE output characteristics, we have to use a DC source of current at the base terminal which may be kept constant during simulation. Different plots can be obtained by keeping the base current at a different constant value. The BJT in the circuit should be associated with a coresponding 'NPN BJT model' during simulations. The resulting circuit diagram is shown in the Figure 10.1.

The output characteristics is a plot between collector current and collectoremitter voltage while keeping the base current constant.

#### PROCEDURE

#### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window

#### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window.

#### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components.

Clicking on the icon on the right toolbar opens the component library. After all the required components of the circuit are placed, wiring is done using the Place Wire option. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose DC sources from eSim\_Sources
- Choose resistors from eSim\_Devices
- Choose NPN from eSim\_Devices
- Choose GND from power
- Choose plot\_i2 from eSim\_Plots

Wire the components to get the circuit. A global labels 'ib' and 'vce' have been added to identify the nodes.

Now we have the circuit diagram as shown in Figure 10.1.

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbarand in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 10.2.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button



Figure 10.1: Schematic diagram for CE output characteristics

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 10.3. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of the circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 10.4. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose DC analysis type. Choose the values of two DC sources, V1 and I1 in the netlist properly as described below. Change V1 from 0V to 5V at an interval of 0.05V. I1 is to be changed from 0 mA to 5 mA at an increment of 1 mA. See Figure 10.5

**Source:** Give the details of source as in Figure 10.6.

Ngspice Model: No Ngspice model to be given.











Figure 10.4: Choose Kicad to Ngspice tool

**Device Model:** The NPN transistor is a device whose model details must be given for simulation. Let us choose the generic NPN model availabe in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/Transistor/NPN.lib. See Figure 10.7.

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

#### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. Since we have used plot components, the required output characteristic plots will automatically pop-up as shown in Figure. 10.8

Analysis Source Details NgSpice Model Device Modeling	Subcircuits	
Enter Source 1	v1	
Start	0	Volts or Amperes
Increment	0.05	Volts or Amperes 🗸 🗸
Stop	5	Volts or Amperes 🗸 🗸
Enter Source 2	it	
Start	0	mV or mA 🗸 🗸
Increment	1	mV or mA 🗸 🗸
Stop	5	mV or mA 🗸 🗸

Figure 10.5: Choose DC analysis type and enter the values of V1 and I1  $\,$ 

Analysis So	urce Details NgSpice	Model Device Modeling	Subcircuits						
Add parameters	s for DC source v1								5
Enter value(Volt	s/Amps):						0		
Add parameters	s for DC source i1				 	 		 	51
Enter value(Volt	s/Amps):						20m		
				 	 	 	 	 Conve	ert

Figure 10.6: GiveSource Details of V1 and I1

### RESULT

The circuit for plotting the common emitter characteristics of NPN transistor was implemented and simulated.



Figure 10.7: Choose the required NPN model



Figure 10.8: The output characteristics of NPN transistor

## Chapter 11

# BJT COMMON BASE CHARACTERISTICS

#### AIM

To design and implement a circuit for simulating the output characteristics of a NPN Bipolar Junction Transistor in common base configuration.

#### DESIGN AND CIRCUIT DIAGRAM

In common base configuration the base terminal is grounded. Input characteristics is the plot between the emitter current  $i_b$  and collector-base volatage  $V_{cb}$ , keeping the collector voltage constant. Output characteristics is the plot between the collector current  $I_c$  and the collector-base volatage  $V_{be}$ 

Inorder to draw the BJT CB output characteristics, we have to use a DC source of current at the base terminal which may be kept constant during simulation. Different plots can be obtained by keeping the emitter current at a different constant value. The BJT in the circuit should be associated with a coresponding 'NPN BJT model' during simulations. The resulting circuit diagram is shown in the Figure 11.1.

The output characteristics is a plot between collector current and collectorbase voltage while keeping the emitter current constant.

#### PROCEDURE

#### Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the wokspace location. It will finally end up in the eSim window

#### Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window.

#### Create the Schematic

To create the schematic, click the very first icon of the left toolbar. This will open KiCad Eeschema.

To create a schematic in KiCad, we need to place the required components.

Clicking on the icon on the right toolbar opens the component library. After all the required components of the circuit are placed, wiring is done using the Place Wire option. Scroll up and down for zooming in and out.

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries.

- Choose DC sources from eSim\_Sources
- Choose resistors from eSim\_Devices
- Choose NPN from eSim\_Devices
- Choose GND from power
- Choose plot\_i2 from eSim\_Plots

Wire the components to get the circuit. A global labels 'ie' and 'vcb' have been added to identify the nodes.

Now we have the circuit diagram as shown in Figure 11.1.

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbarand in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 11.2.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.
- 3. Choose the required libraries from 'user/share/kicad/library' and click OK button

88



Figure 11.1: Schematic diagram for CE output characteristics

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a .cir file. Do not change the directory while saving.See Figure 11.3. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of the circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 11.4. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

**Analysis:** Choose DC analysis type. Choose the values of two DC sources, V1 and I1 in the netlist properly as described below. Change V1 from -1V to 5V at an interval of 0.02V. I1 is to be changed from -1 mA to 5 mA at an increment of 1 mA. See Figure 11.5

**Source:** Give the details of source as in Figure 11.6.

Ngspice Model: No Ngspice model to be given.











Figure 11.4: Choose Kicad to Ngspice tool

**Device Model:** The NPN transistor is a device whose model details must be given for simulation. Let us choose the generic NPN model availabe in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/Transistor/NPN.lib. See Figure 11.7.

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. Now you are ready to see the simulation results.

#### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. Since we have used plot components, the required output characteristic plots will automatically pop-up as shown in Figure. 11.8

Analysis Source Details NgSpice Model Device Modeling	Subcircuits	
Enter Source 1	v1	
Start	-1	Volts or Amperes
Increment	0.02	Volts or Amperes
Stop	5	Volts or Amperes
Enter Source 2	it	
Start	-1	mV or mA 🗸 🗸
Increment	1	mV or mA 🗸 🗸
Stop	5	mV or mA 🗸 🗸

Figure 11.5: Choose DC analysis type and enter the values of V1 and I1

Add parameters for DC source v1       I2         Enter value(Volts/Amps):       I2         Add parameters for DC source i1       I2         Enter value(Volts/Amps):       20m	Analysis Source	e Details Ng	Spice Model	Device Modeling	Subcircuits				
Enter value(Volts/Amps):  Add parameters for DC source i1  Enter value(Volts/Amps):  20m	Add parameters fo	r DC source v1							
Add parameters for DC source II	Enter value(Volts/A	mps):						12	
Enter value(Volts/Amps): 20m	Add parameters fo	r DC source i1							$\equiv$
	Enter value(Volts/A	mps):						20m	

Figure 11.6: Give Source Details of V1 and I1

### RESULT

The circuit for plotting the common base characteristics of NPN transistor was implemented and simulated.



Figure 11.7: Choose the required NPN model



Figure 11.8: The output characteristics of NPN transistor

## Chapter 12

# ZENER REGULATOR WITH SERIES PASS TRANSISTOR

### AIM

To design and implement a zener diode regulator with series pass transistor and to plod the line regulation characteristics.

#### DESIGN AND CIRCUIT DIAGRAM

Zener diode maintains a constant voltage across its terminals when reverse biased and the applied volatage is above the reverse breakdown voltage of the diode.

The circuit diagram for implementing a series pass transistor zener diode regulator is shown in Figure 12.10.

#### PROCEDURE

#### Launch eSim

Launching eSim will take you to the dialog box. It asks for the default workspace. Browse the folders and set the wokspace location. It will end up in the eSim window shown in Figure 12.1.

94CHAPTER 12. ZENER REGULATOR WITH SERIES PASS TRANSISTOR

٩		
<ul><li>2 2 4</li></ul>	r J	OSSEE
Projects	Wekcree	<ul> <li>C</li> </ul>
	About esim About esim and an uper source table for random production and/real and/real and/real and real and real and real and and real of real and and real and after source of remaindering "BMC defined in their Learnes. The decision from about of learness and its better in the decision of the decision o	er e scal på og

Figure 12.1: Launching eSim will take you to this window

#### Create a New Project

The new project is created by clicking the New icon on the menubar. Give the name of the project ,'ZenerRegulator' in the pop up window as shown in Figure.12.2.

٩	eSim 🗸 🔶 😪
<b>9 9 2</b> 4	tossee.
Projects	Welcome O 🕲
Kíng .	About eSim eSin a new source IDA kof for creat design; sinulation, analysis and PCB design. It is an integrated tool built using spen source software such as KCad (http://www.kcad.pcb.org) and Appendity.
	eSim source is released under GNU General Public License.
	eSim is developed by the FOSSEE team at IIT Bombay. To know more about eSim please visit http://esim tossee in
	To discuss more about eSim please v Enter Project Name:
yafti	dodeharj
	eSim Started Project Salected 1kine
	(HFC) Workspace . AnnekaryslafSim Workspace

Figure 12.2: Creating new project

#### Create the Schematic

To create the schematic, click the very first icon of the left tool bar as shown in the Figure 12.3 . This will open KiCad Eeschema.



Figure 12.3: Creating new schematic diagram

To create a schematic in KiCad, we need to place the required components. See Figure 12.4. Figure 12.5 shows the icon on the right toolbar which opens the component library. After all the required components of the simple RC circuit are placed, wiring is done using the Place Wire option as shown in the Figure 12.6. Scroll up and down for zooming in and out.



Figure 12.4: The Kicad Eeschema page



Figure 12.5: Place component icon

Figure 12.6: Place wire icon

**Placing the Components:** Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries. See Figure 12.7.



Figure 12.7: The Kicad Libraries of components

- Choose DC source from eSim\_Sources
- Choose R from eSim\_Devices
- Choose zener from eSim\_Devices

- Choose NPN from eSim\_Devices
- Choose plot\_v1 from eSim\_Plot
- Choose GND from power

Select the resistor and edit its component value to 1k as shown in Figure 12.8.



Figure 12.8: Editing the value field of component R

Wire the components to get the circuit. A global label 'in' and 'out' has been added to identify that node whose voltage will be later recorded and plotted.

Annotating the circuit: Once the schematic diagram is completed, annotate it so that the 'question marks' associated with the components are converted to meaningful numbers automatically.For that choose annotate button from the top toolbar(See Figure 12.9 and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 12.11.



Figure 12.10: Schematic diagram for Zener Diode Regulator



Figure 12.9: Choose annotate from the toop tool bar

Now we have the circuit diagram as shown in Figure 12.10.

**Note:** If some libraries are found missing, you can add them from the 'Preferences' menu by following the procedure:

- 1. Choose 'Component Libraries' from Preferences menu.
- 2. Click on the Add button on the top right side of the window.



cbnew OrcadPCB2 CadStar Spice 1	Generate
Options:	Cancel
Prefix references 'U' and 'IC' with 'X'  Run Simulator	Add Plugin
Use net number as net name	Remove Plugin
Simulator command:	Use default netname
Gault Marklink Cilemanna	

Figure 12.12: Netlist Generation

Figure 12.11: Annotation

3. Choose the required libraries from 'user/share/kicad/library' and click OK button

#### **Create Netlist**

To simulate the circuit that has been created in the previous section, we need to generate its netlist. Netlist is a list of components in the schematic along with their connection information. To do so, click on the Generate netlist tool from the top toolbar. Click on spice from the window that opens up. Check the option Default Format. Then click on Generate. Save the netlist. This will be a \*.cir file. Do not change the directory while saving.See Figure 12.12. Now the netlist is ready to be simulated.

#### KiCad to Ngspice conversion

To convert KiCad netlist of the circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 12.13. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

	Projects
	▼ RC
KINg	- RC-cache.lib - RC.pro
	pice
Contract of the local division of the local	
y-f(x)	

Figure 12.13: Choose Kicad to Ngspice tool

#### 100CHAPTER 12. ZENER REGULATOR WITH SERIES PASS TRANSISTOR

**Analysis:** Choose DC analysis type. Give the values of DC variables as shown in Figure 12.14. Enter the name of your DC source as on the circuit (here v1) and let its value be varied from 6V to +15V with a step of 1 V.

, DC Analysis					
	Enter Source 1	M			
	Start	6	Volts or Amperes		
	Increment	1	Volts or Amperes		
	Stop	15	Volts or Amperes		

Figure 12.14: Choose DC analysis type and enter the values

Source Details: Leave this empty.

**Ngspice Model:** Ngspice model of zener diode will be loaded. You can see the default values of various zener parameters there. You can change those if required. In this example the breakdown voltage has been set as 8V. See Figure.12.15

Analysis Source Details NgSpice Model	Device Modeling Subcircuits			
Add Parameters for Zener Diode ut				
Enter Saturation Current (default=1.0e-12)				
Enter Forward Emission Coefficient (default=1.0)				
Enter Breakdown Voltage (default=5.6)	8			
Enter Breakdown Current (default=2.0e-2)				
Enter Switch for Limiting (default=FALSE)				

Figure 12.15: Choose ngspice model values

**Device Model:** The NPN Transistor is a device whose model details must be given for simulation. Let us choose the generic BJT model available in the eSim model library. Browse it from /opt/eSim/src/deviceModelLibrary/Transistor/NPN.lib. See Figure 12.16.



Figure 12.16: Choose the required Transistor model

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. See Figure 12.16. Now you are ready to see the simulation results.

#### Simulate

To run Ngspice simulation click the simulation icon in the left tool bar. It will open up two windows - ngspice plotting window and python plotting window. Inorder to plot the voltages at input and output let us use the commands in ngspice plotting window.

Since we have used eSim\_plot components at in and out, simulate button click will automatically plot the voltages.

To plot the value of voltages on a signle plot window type the following command

plot v(in), v(out)

This would pop up the required characteristics fo the diode as defined in the diode model D.lib. For a differnt diode model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 12.17.



Figure 12.17: The line regulation characteristics of zener diode

### RESULT

The circuit for plotting the characteristics of zener regulator was implemented and simulated.

# Bibliography

[1] User Manual of eSim by FOSSEE, IIT Bombay.(http://esim.fossee.in/ resource/book/esimusermanual.pdf