Electronic Circuit Simulation using eSim

Kavya Manohar

August 7, 2016

Copyright ©2015

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

List of Figures

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 curernt 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.

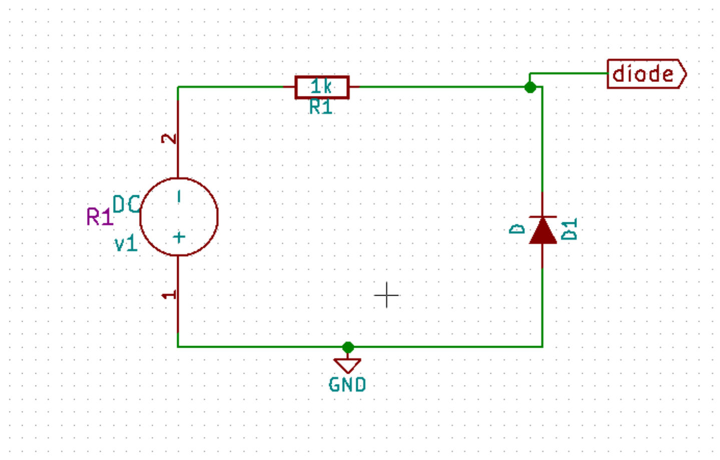

Figure 2.1: Schematic diagram for diode characteristics

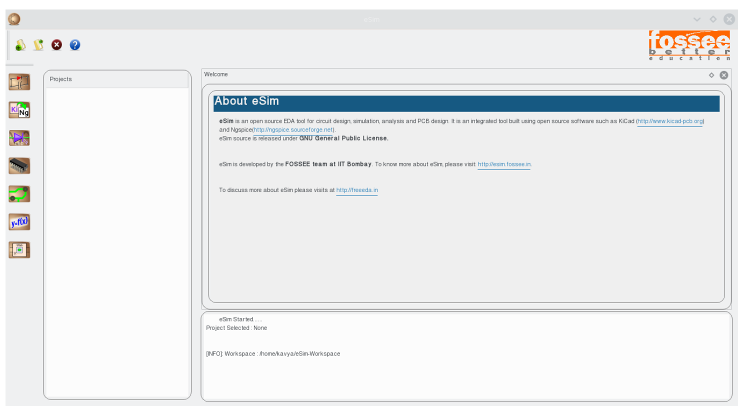

Figure 2.2: Launching eSim will take you to this window

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.

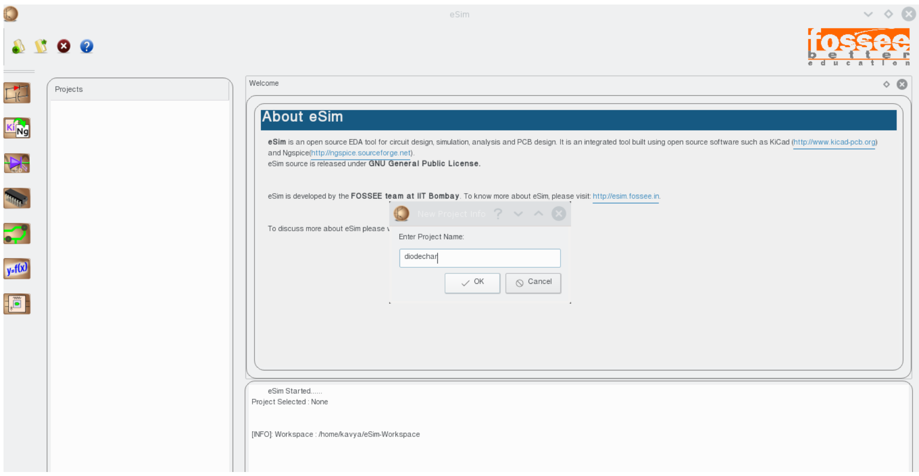

Figure 2.3: Creating new project

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.

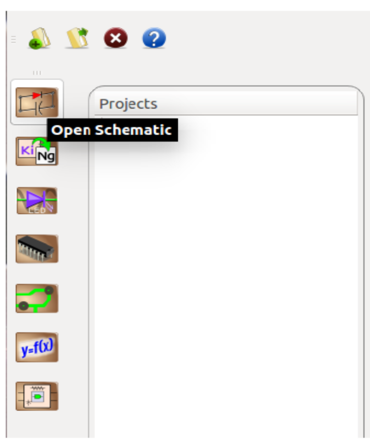

Figure 2.4: Creating new schematic diagram

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.

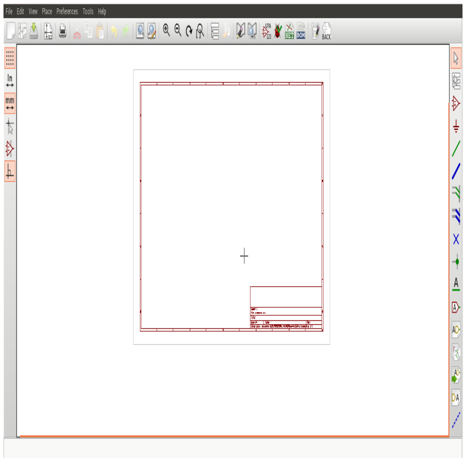

Figure 2.5: The Kicad Eeschema page

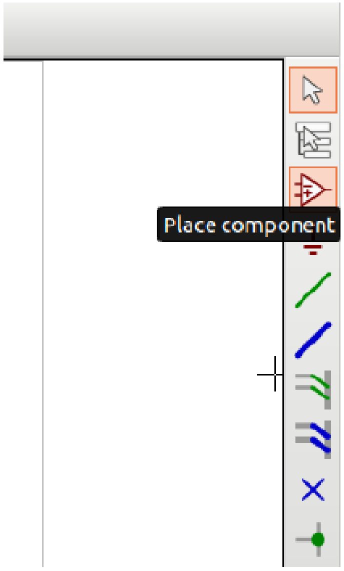

Figure 2.6: Place component icon

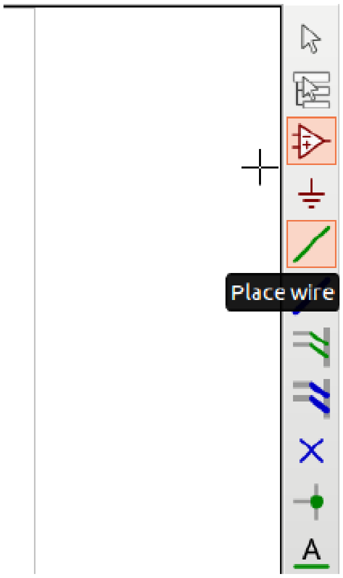

Figure 2.7: Place wire icon

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

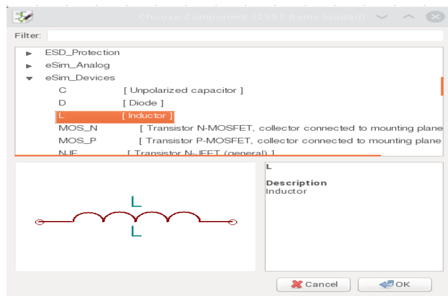

Figure 2.8: The Kicad Libraries of components

- 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.

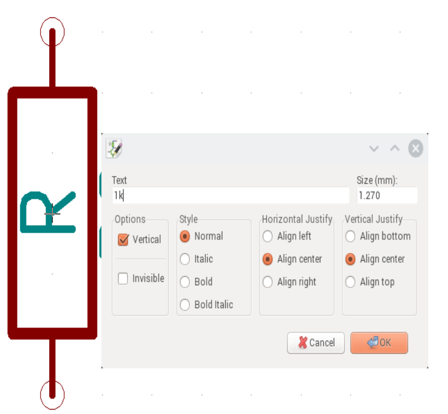

Figure 2.9: Editing the value field of component R

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.

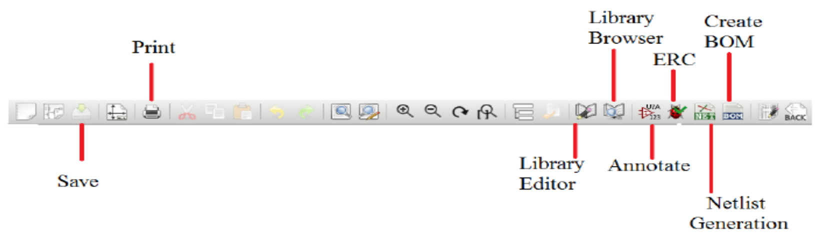

Figure 2.10: Choose annotate from the toop tool bar

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.

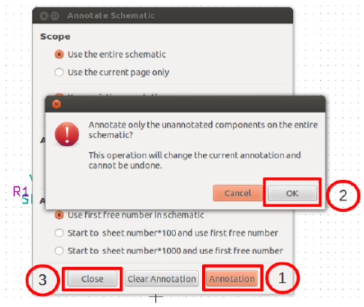

Figure 2.11: Annotation

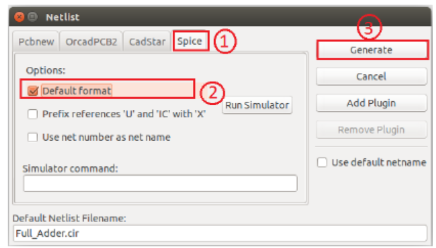

Figure 2.12: Netlist Generation

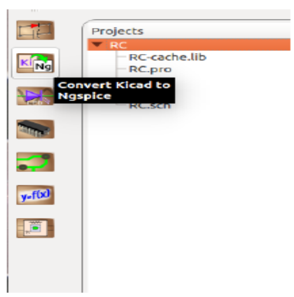

Figure 2.13: Choose Kicad to Ngspice tool

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.

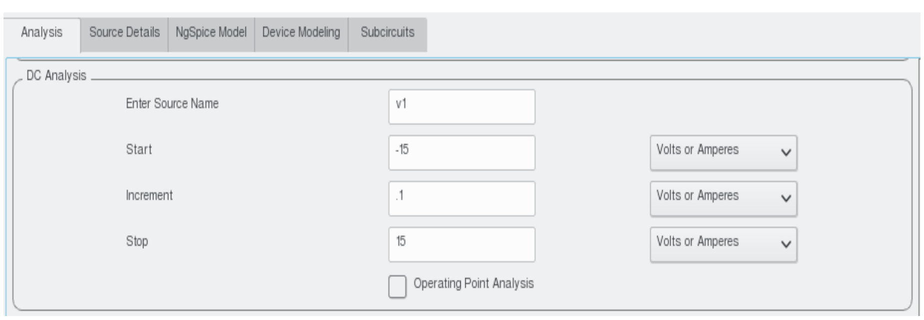

Figure 2.14: Choose DC analysis type and enter the values

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.

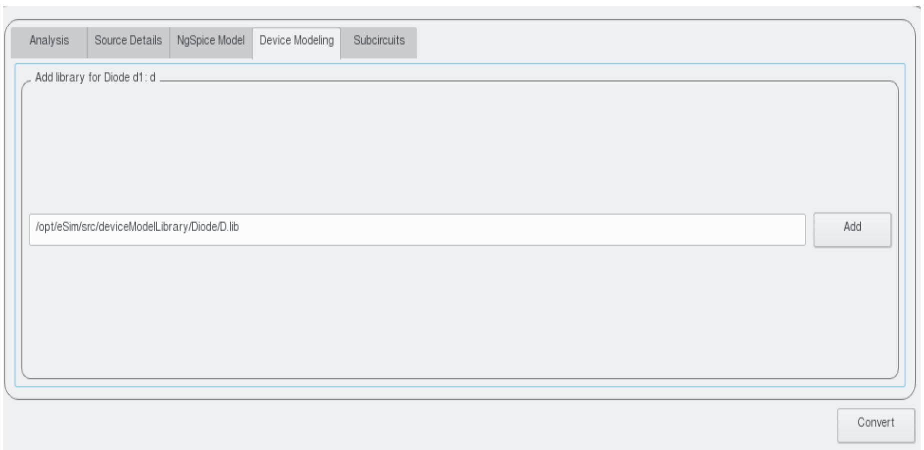

Figure 2.15: Choose the required diode model

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.

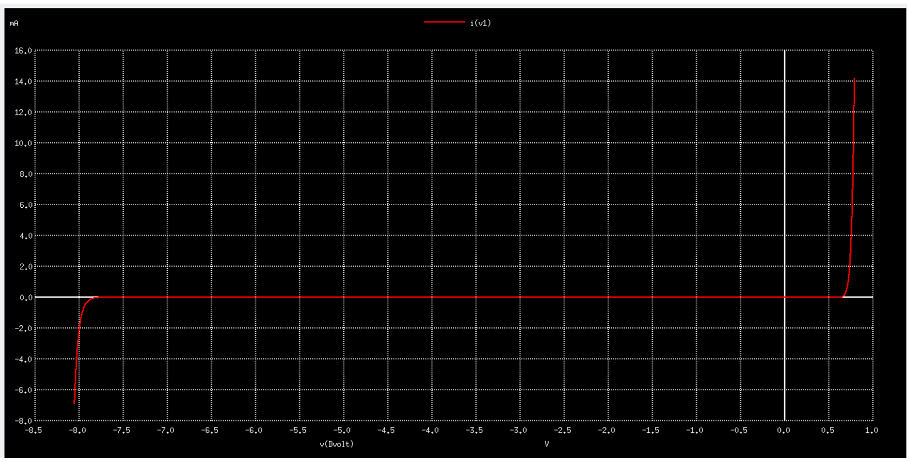

Figure 2.16: The characteristics of Diode

RESULT

The circuit for plotting the charateristics 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 durinmg 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

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

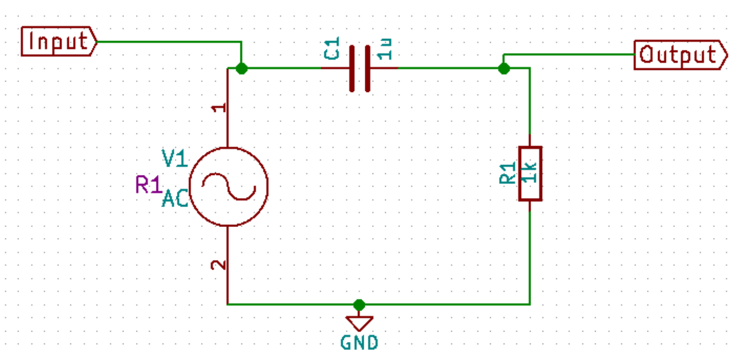

Figure 3.1: Schematic diagram for passive RC high pass filter

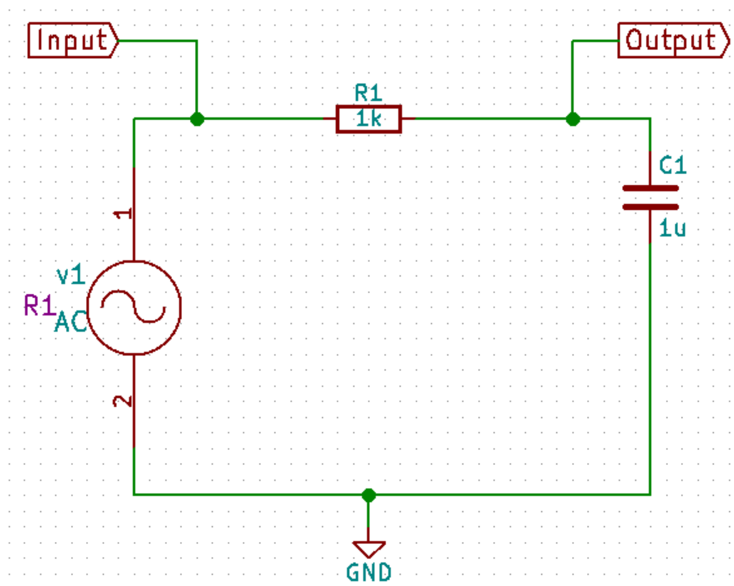

Figure 3.2: Schematic diagram for passive RC low pass filter

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 availbale 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 μ F. You can just type in 1u.

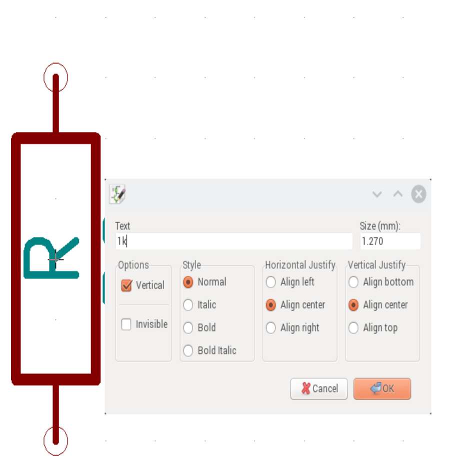

Figure 3.3: Editing the value field of component R

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.

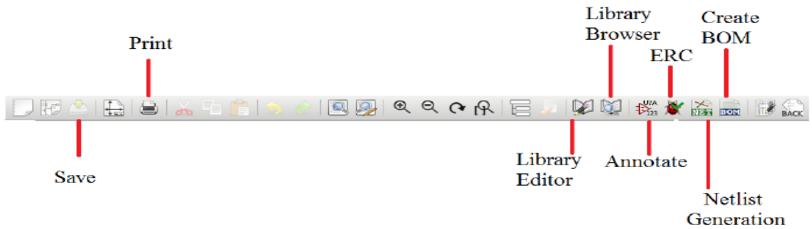

Figure 3.4: Choose annotate from the toop tool bar

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.

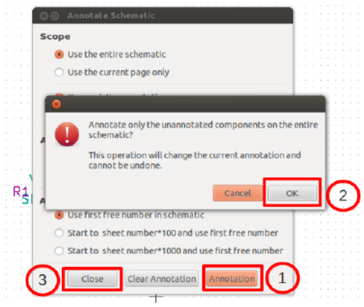

Figure 3.5: Annotation

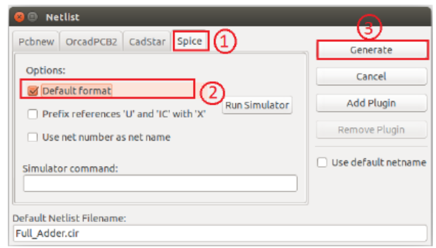

Figure 3.6: Netlist Generation

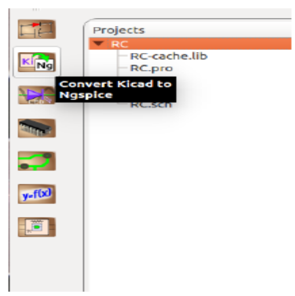

Figure 3.7: Choose Kicad to Ngspice tool

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.

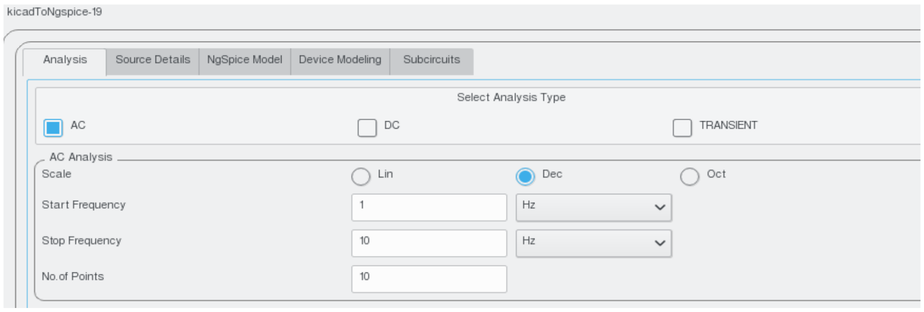

Figure 3.8: Choose AC analysis type and enter the values

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.

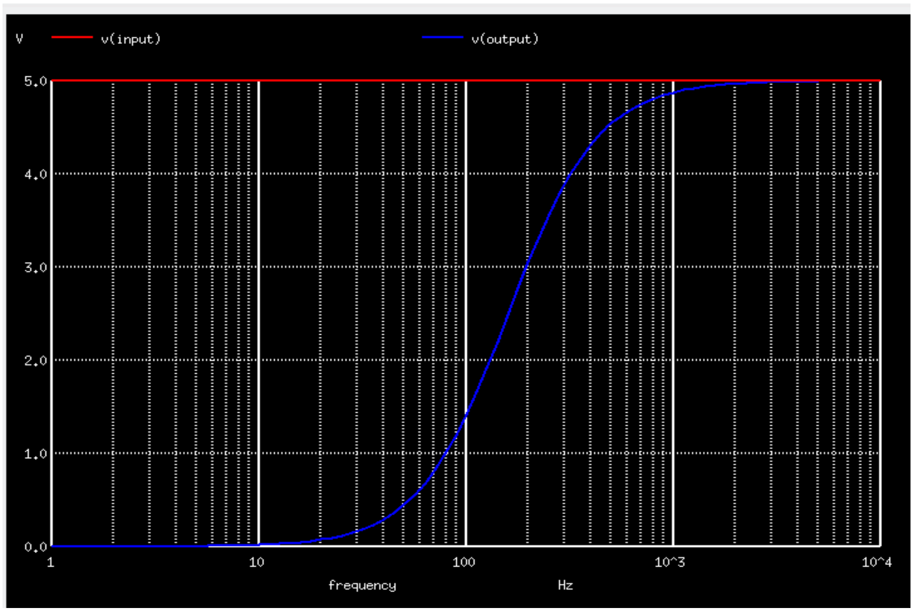

Figure 3.9: The frequency response of RC highpass filter

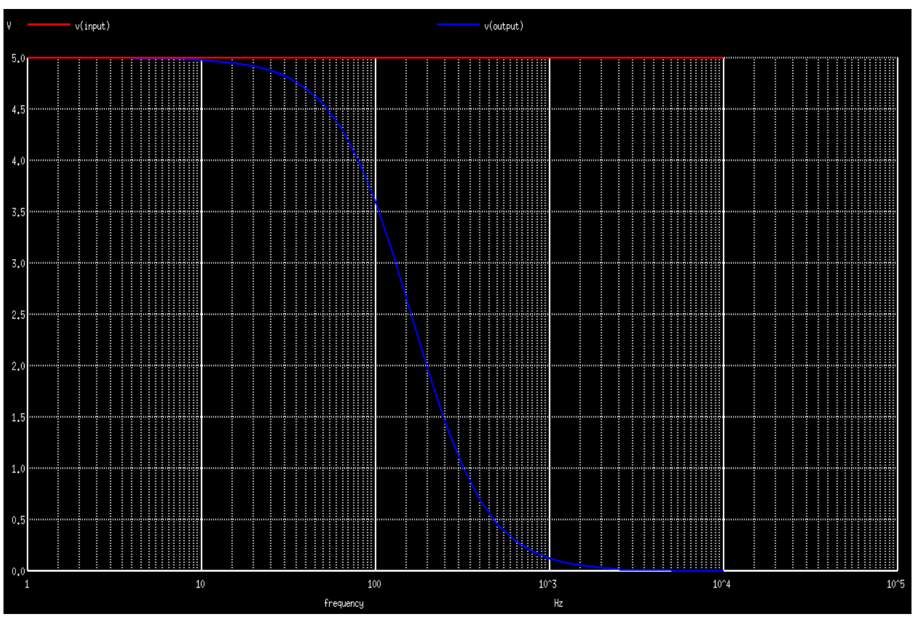

Figure 3.10: The frequency response of RC low pass filter

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.

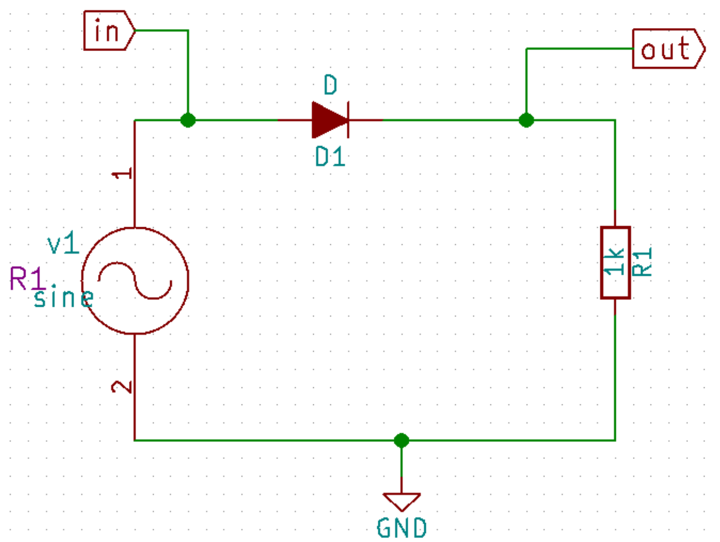

Figure 4.1: Schematic diagram for series clipper circuit

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 availbale 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.

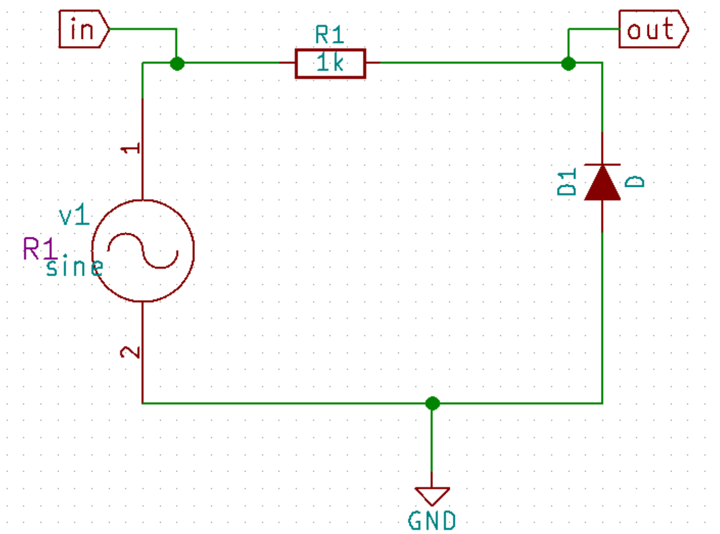

Figure 4.2: Schematic diagram for shunt clipper circuit

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.

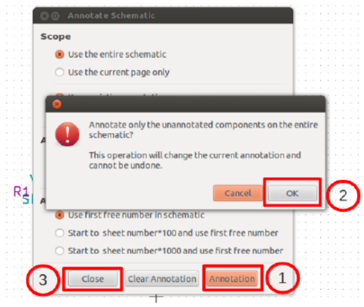

Figure 4.3: Annotation

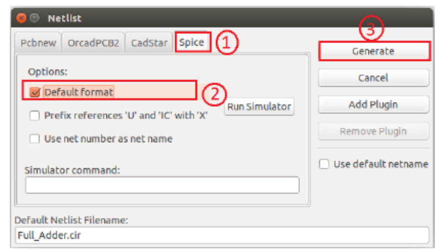

Figure 4.4: Netlist Generation

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'.

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

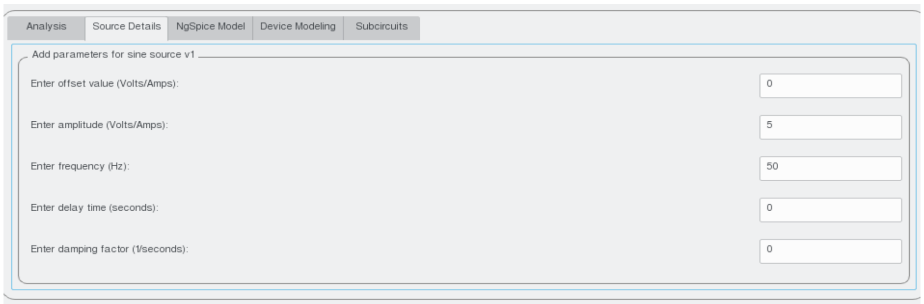

Figure 4.6: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

Device Model: Add thediode model availablein 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.

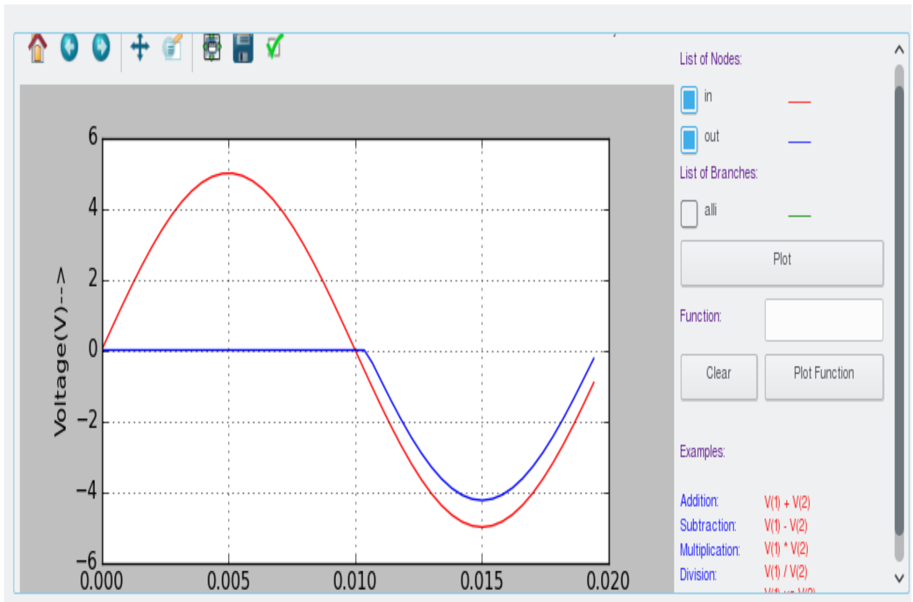

Figure 4.7: The transient response of the series clipper on python plotting window

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.

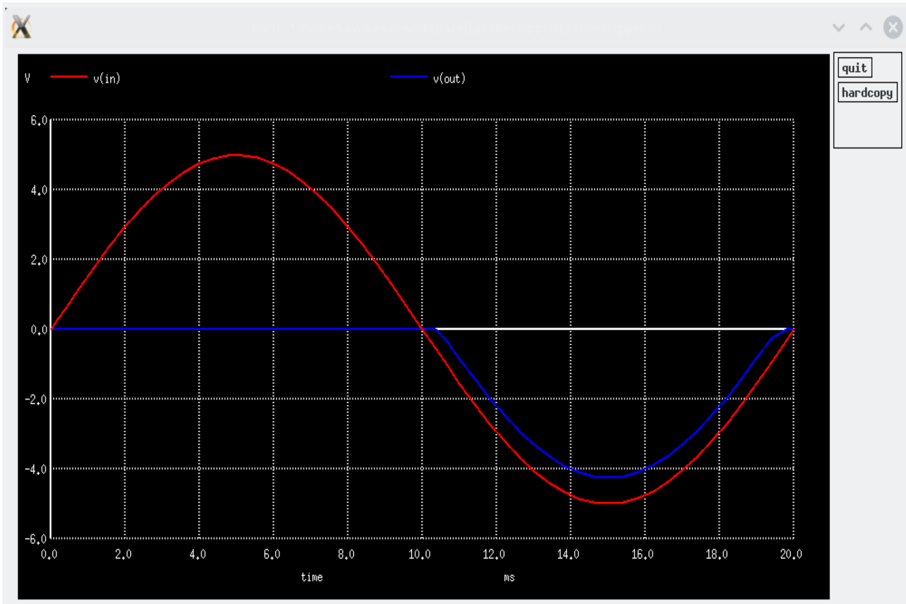

Figure 4.8: The transient response of the series clipper in ngspice plotting window

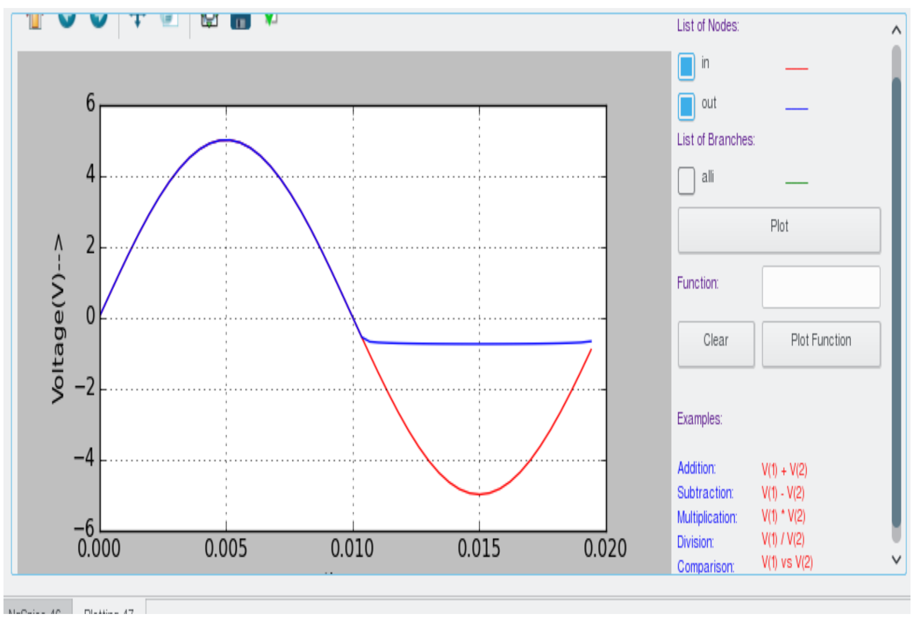

Figure 4.9: The transient response of the shunt clipper on python plotting window

Chapter 5

CLAMPING CIRCUIT

AIM

To design and implement circuit for clamping waveforms

DESIGN AND CIRCUIT DIAGRAM

Inorder 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.

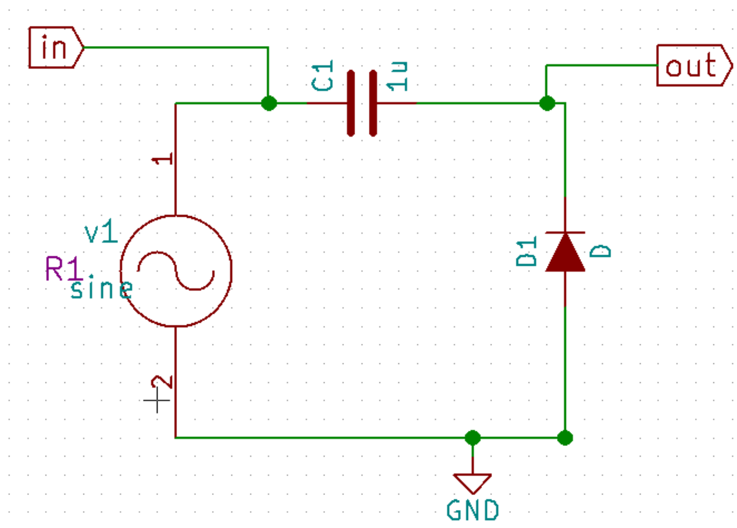

Figure 5.1: Schematic diagram for clamper circuit

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 availbale 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.

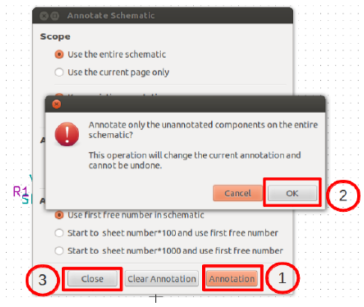

Figure 5.2: Annotation

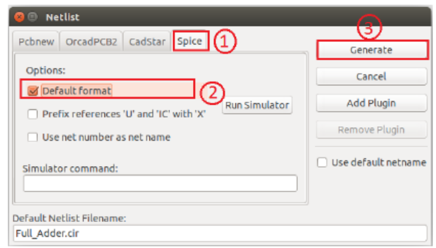

Figure 5.3: Netlist Generation

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'.

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

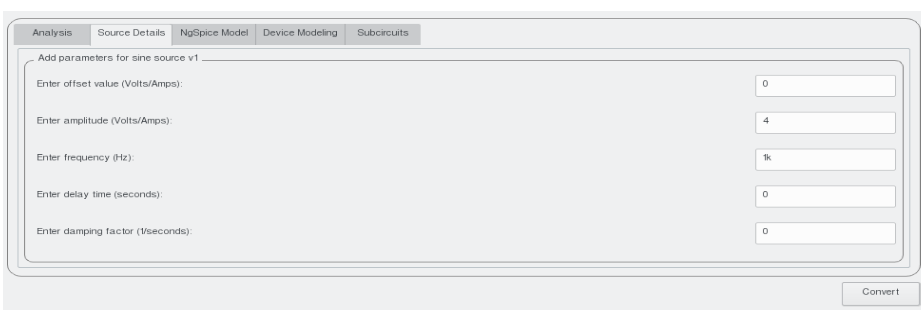

Figure 5.5: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

Device Model: Add the diode model availablein 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.

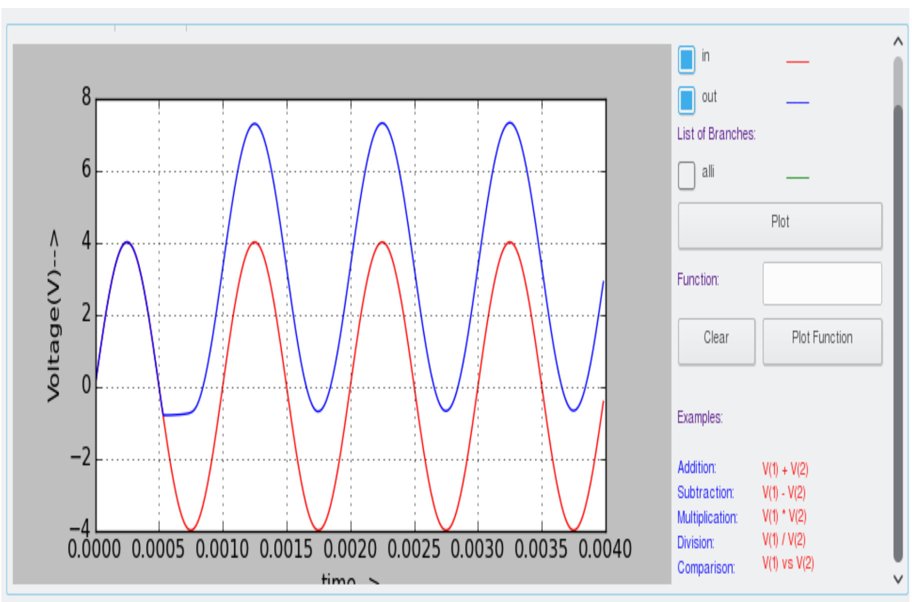

Figure 5.6: The transient response of the clamper on python plotting window

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 1kΩ. 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.

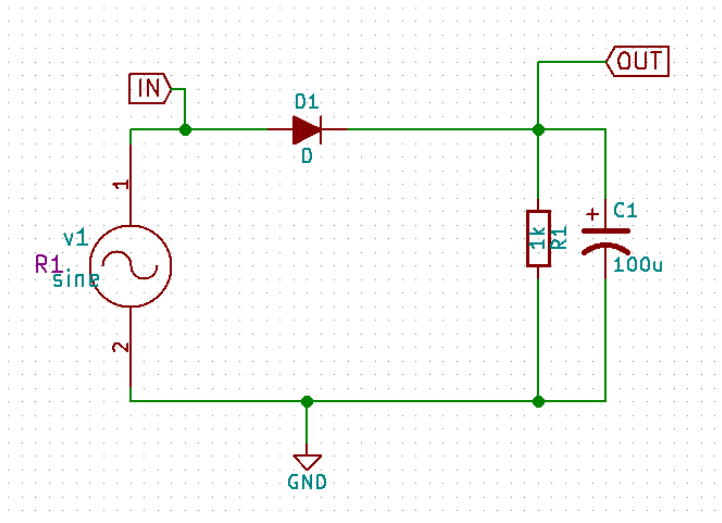

Figure 6.1: Schematic diagram for half wave rectifier with C filter

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 availbale 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

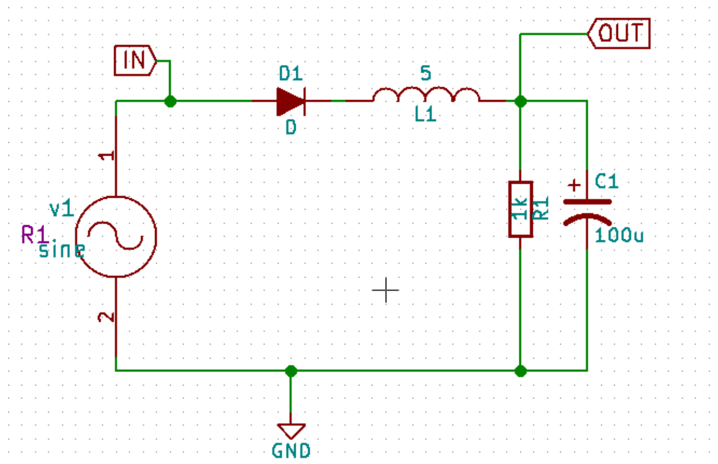

Figure 6.2: Schematic diagram for half wave rectifier with LC filter

- 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

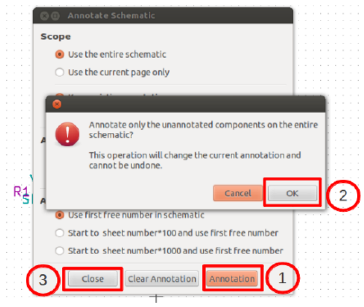

Figure 6.3: Annotation

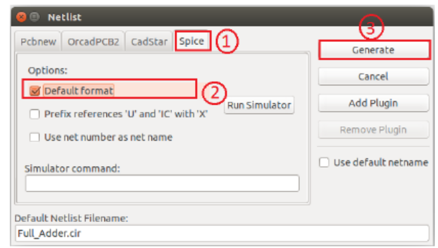

Figure 6.4: Netlist Generation

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'.

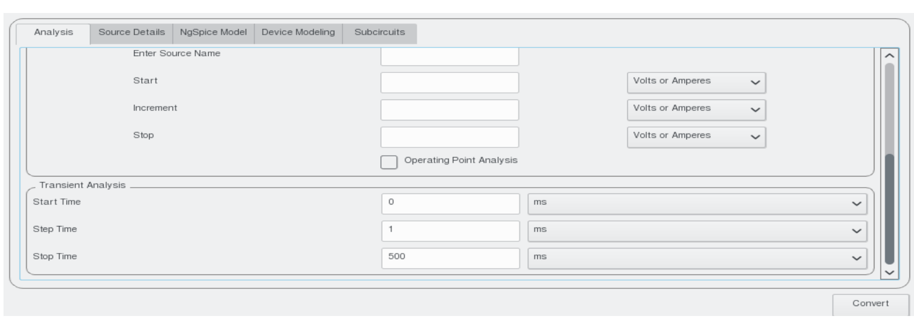

Figure 6.5: Choose analysis type as 'transient' and enter the values

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

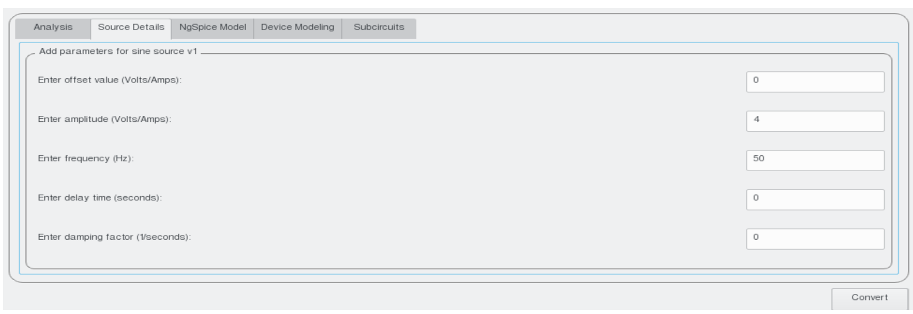

Figure 6.6: Enter the parameters of 'Sine' source

Ngspice Model: No Ngspice model to be given.

Device Model: Add thediode model availablein 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.

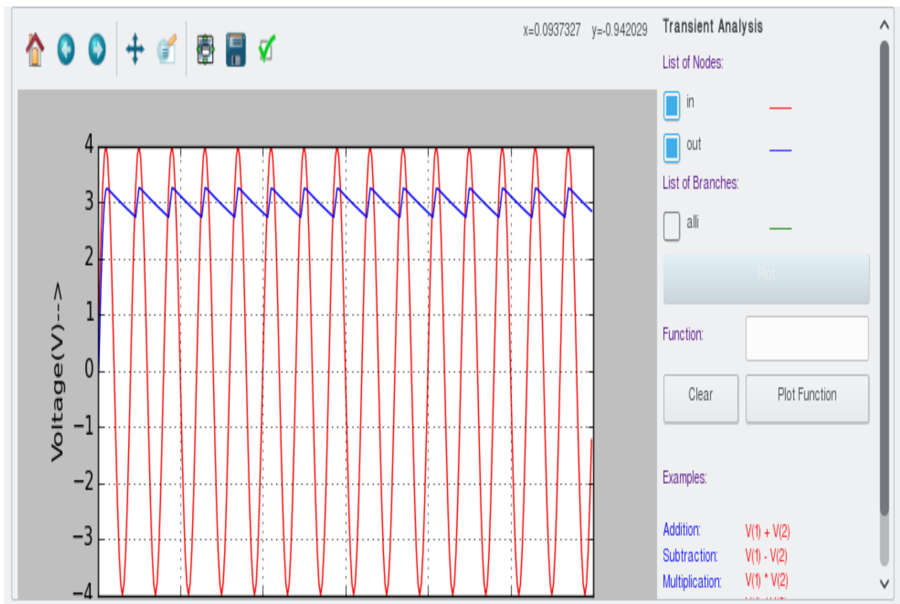

Figure 6.7: The transient response of the half wave rectifier with capacitor filter on python plotting window

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.

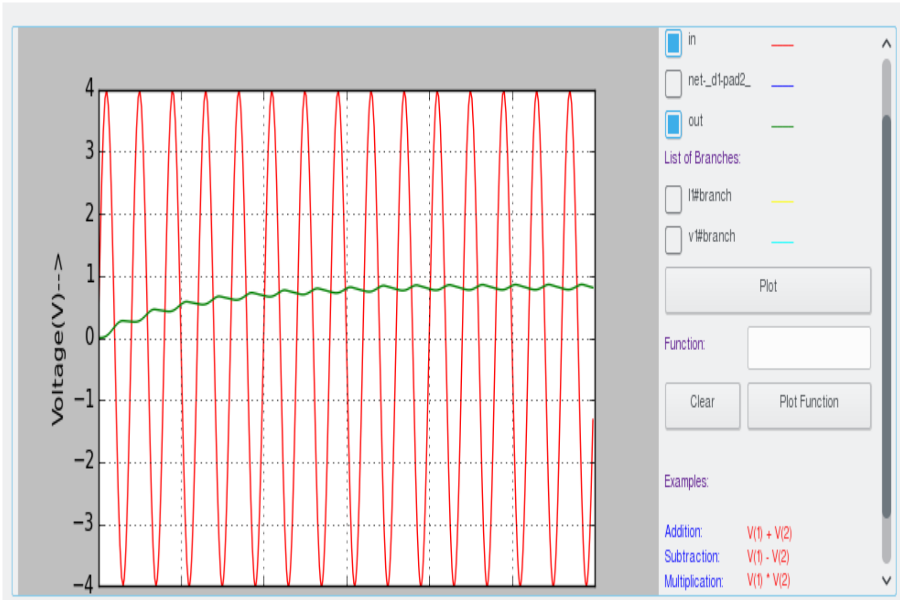

Figure 6.8: The transient response of the half wave rectifier with inductor and capacitor filter on python plotting window

.

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 1kΩ. 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.

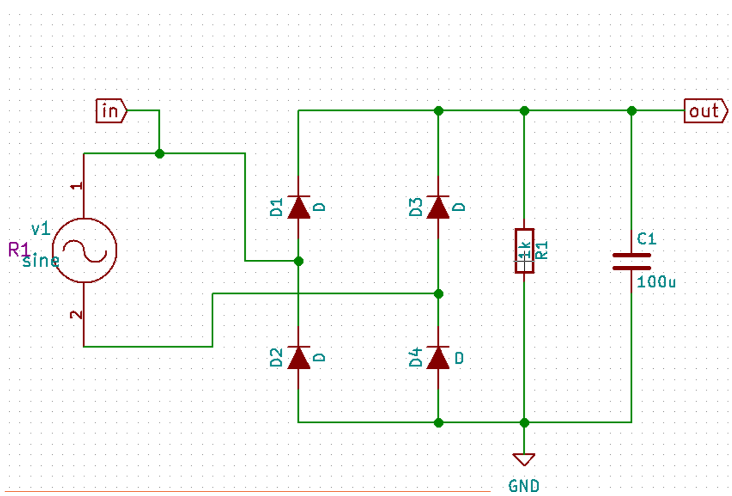

Figure 7.1: Schematic diagram for bridge rectifier with C filter

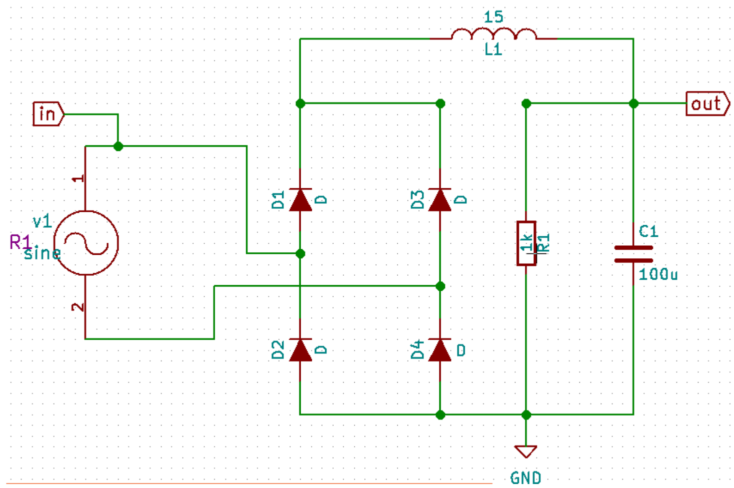

Figure 7.2: Schematic diagram for bridge rectifier with LC filter

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 availbale 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.

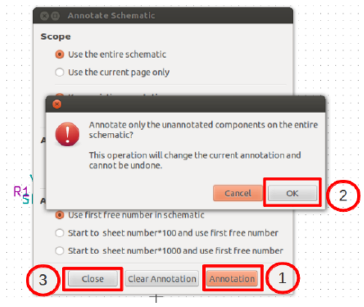

Figure 7.3: Annotation

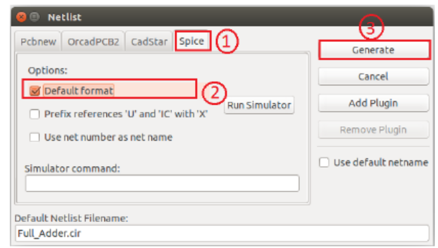

Figure 7.4: Netlist Generation

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'.

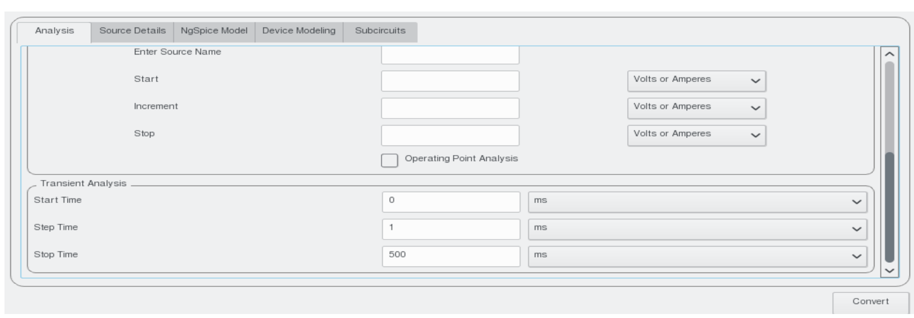

Figure 7.5: Choose analysis type as 'transient' and enter the values

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

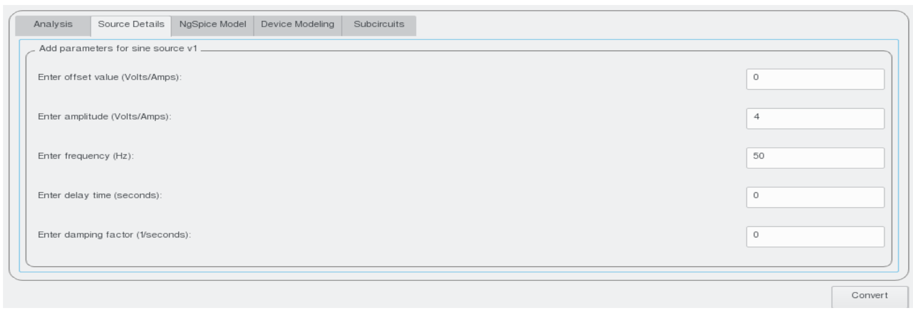

Figure 7.6: Enter the parameters of ‘Sine’ source

Ngspice Model: No Ngspice model to be given.

Device Model: Add thediode model availablein 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.

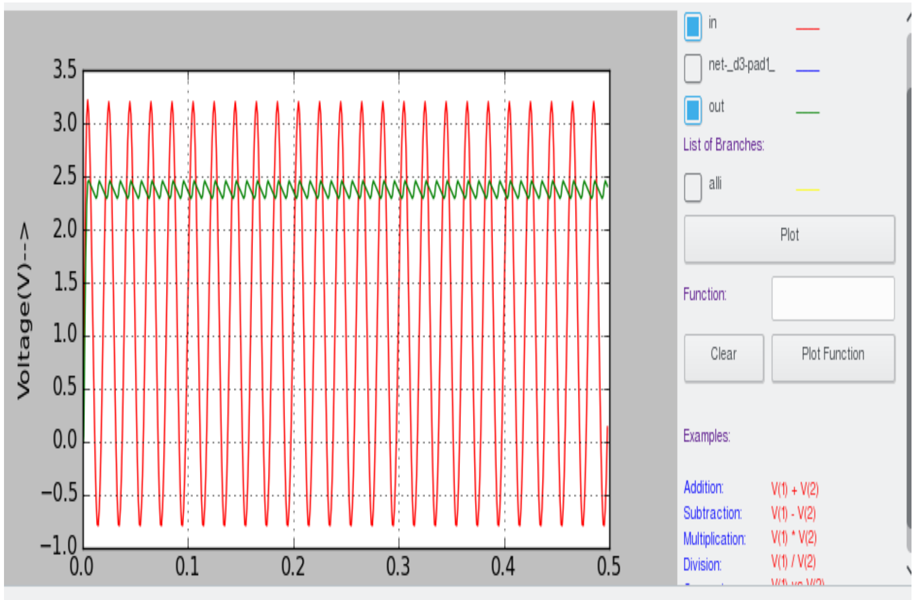

Figure 7.7: The transient response of the bridge rectifier with capacitor filter on python plotting window

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.

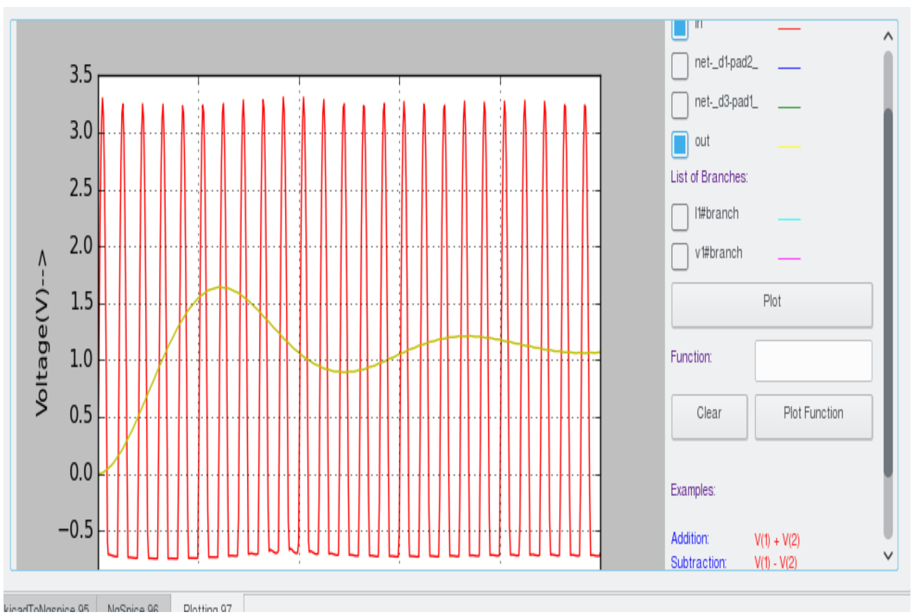

Figure 7.8: The transient response of the bridge rectifier with inductor and capacitor filter on python plotting window

.

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.

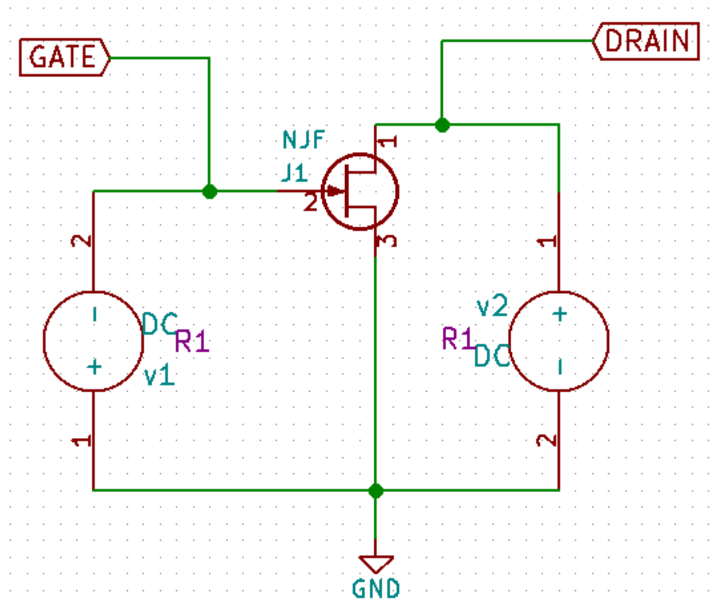

Figure 8.1: Schematic diagram for JFET characteristics

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 availbale 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.

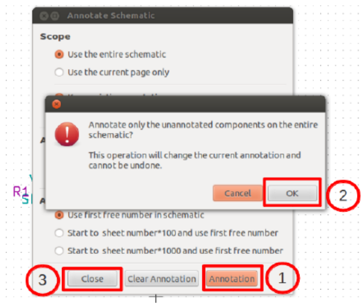

Figure 8.2: Annotation

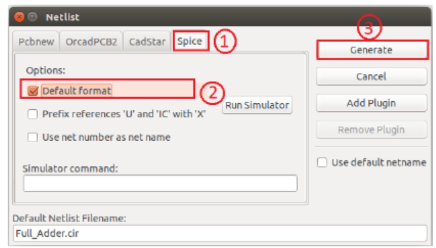

Figure 8.3: Netlist Generation

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.

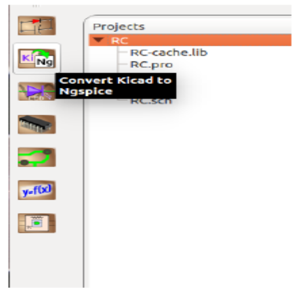

Figure 8.4: Choose Kicad to Ngspice tool

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.

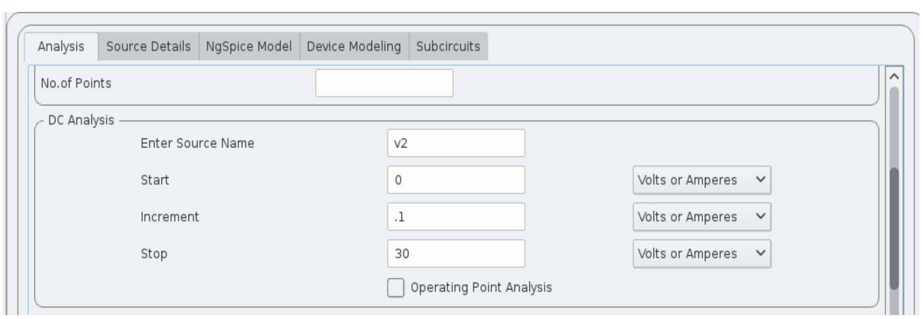

Figure 8.5: Choose DC analysis type and enter the values of V2

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

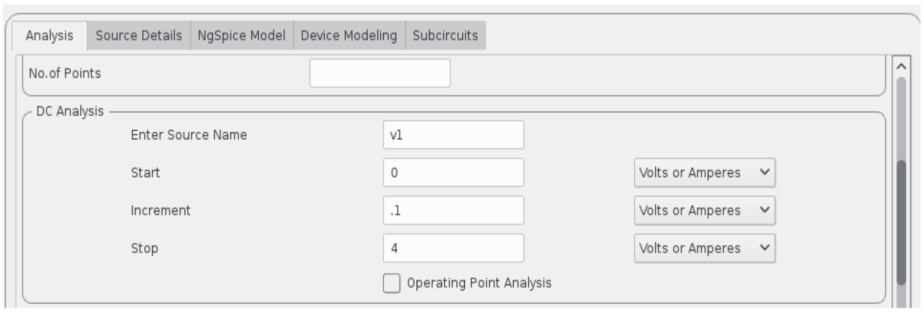

Figure 8.6: Choose DC analysis type and enter the values of V1

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

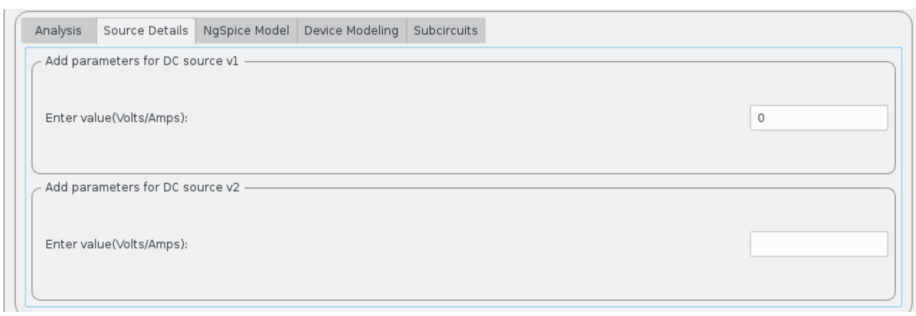

Figure 8.7: Enter the details of fixed source V1

- **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.

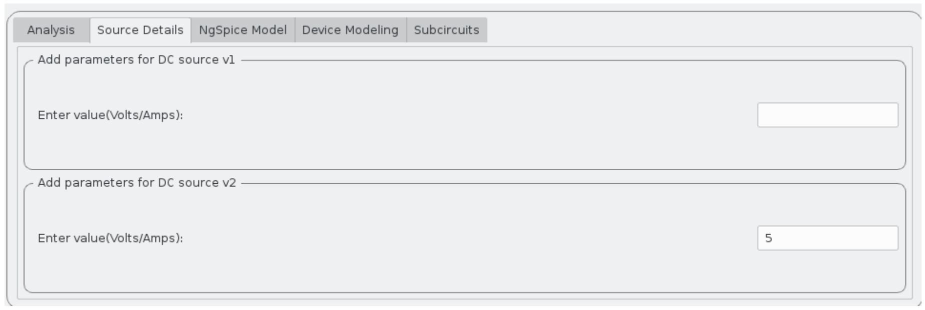

Figure 8.8: Enter the details of fixed source V2

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 differnt device model the characteristics would be slightly different.

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

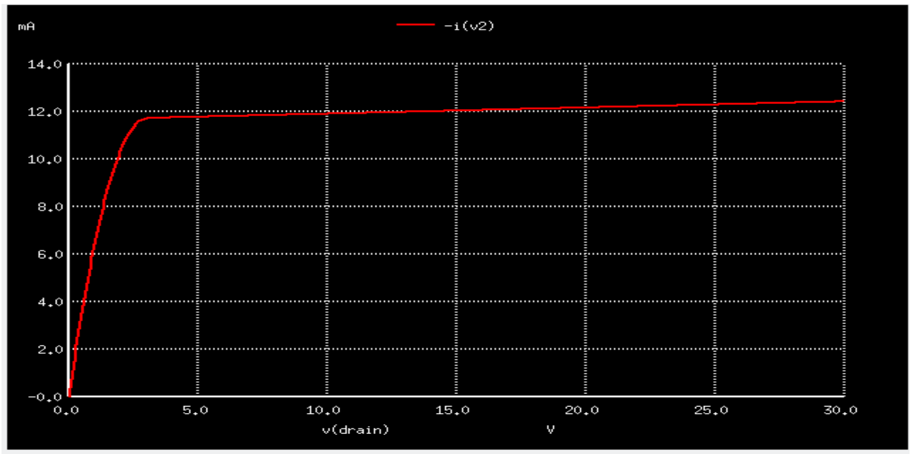

Figure 8.10: The drain characteristics of JFET with gate voltage =0V

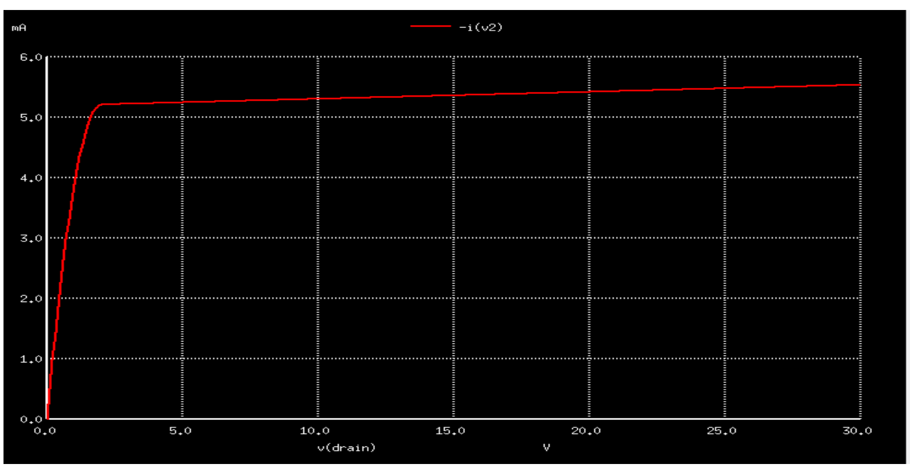

Figure 8.11: The drain characteristics of JFET with gate voltage =1V

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 differnt device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 8.12.

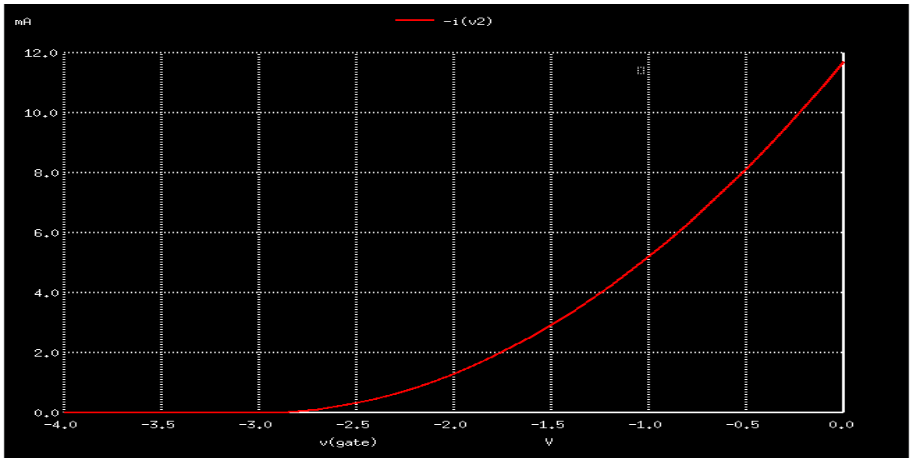

Figure 8.12: The transfer characteristics of JFET with drain voltage =3V

RESULT

The circuit for plotting the charateristics 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.

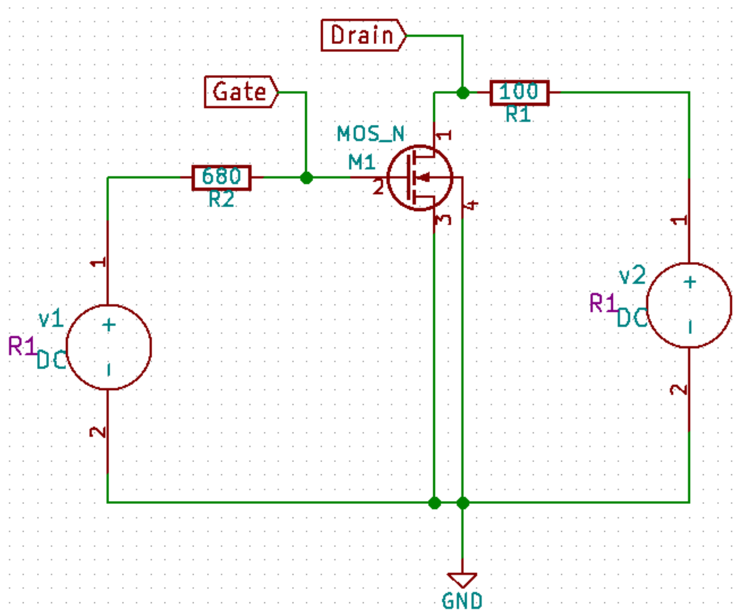

Figure 9.1: Schematic diagram for MOSFET characteristics

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 availbale 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.

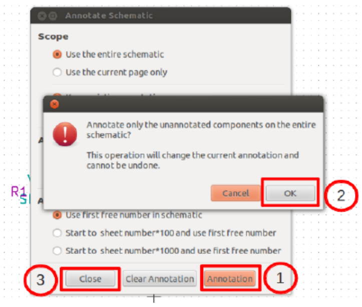

Figure 9.2: Annotation

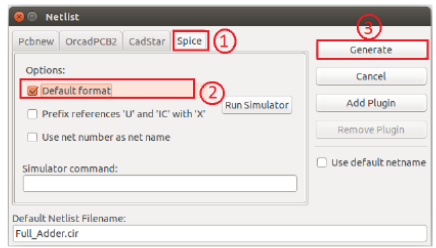

Figure 9.3: Netlist Generation

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.

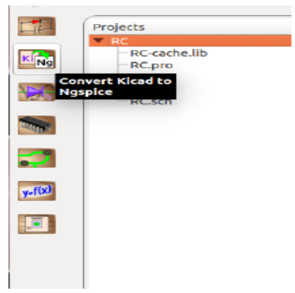

Figure 9.4: Choose Kicad to Ngspice tool

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.

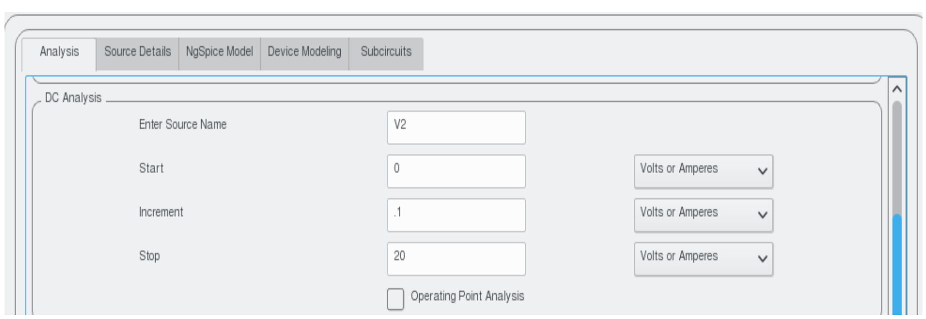

Figure 9.5: Choose DC analysis type and enter the values of V2

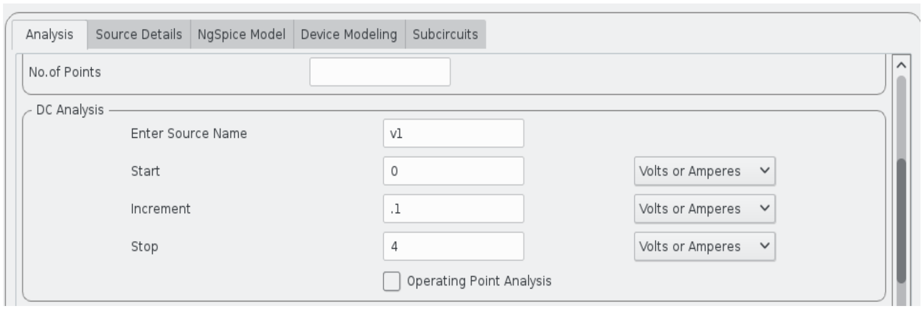

Figure 9.6: Choose DC analysis type and enter the values of V1

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.)

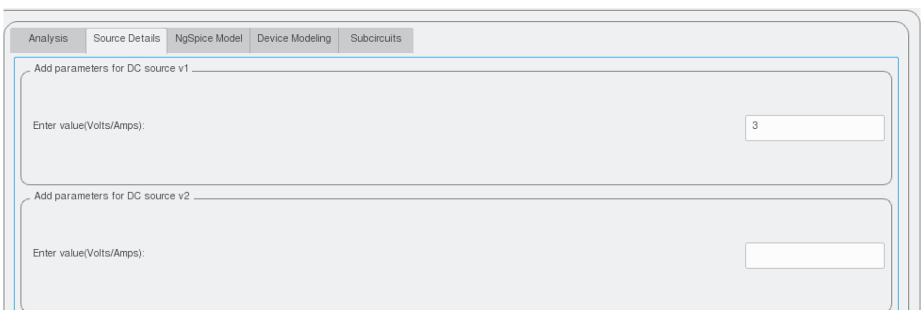

Figure 9.7: Enter the details of fixed source V1

- **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.

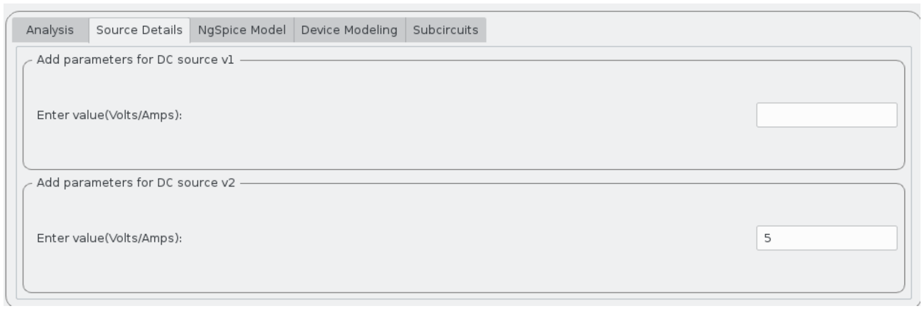

Figure 9.8: Enter the details of fixed source V2

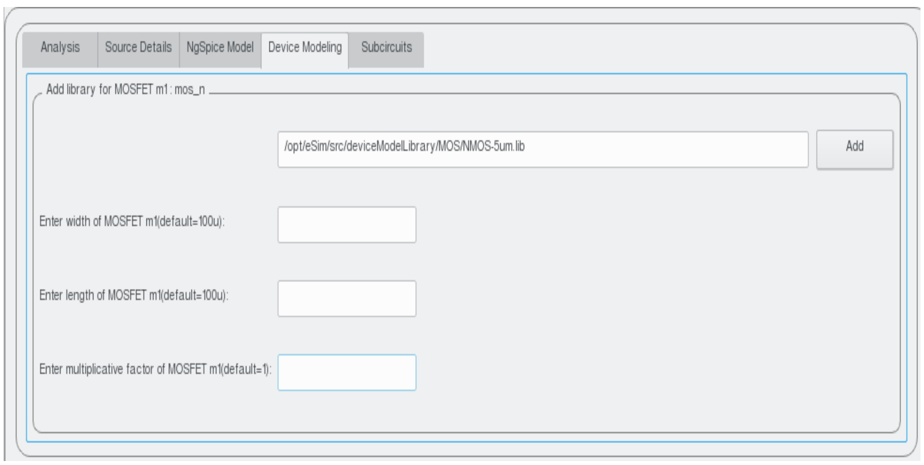

Figure 9.9: Choose the required MOSFET model

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 differnt device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.10.

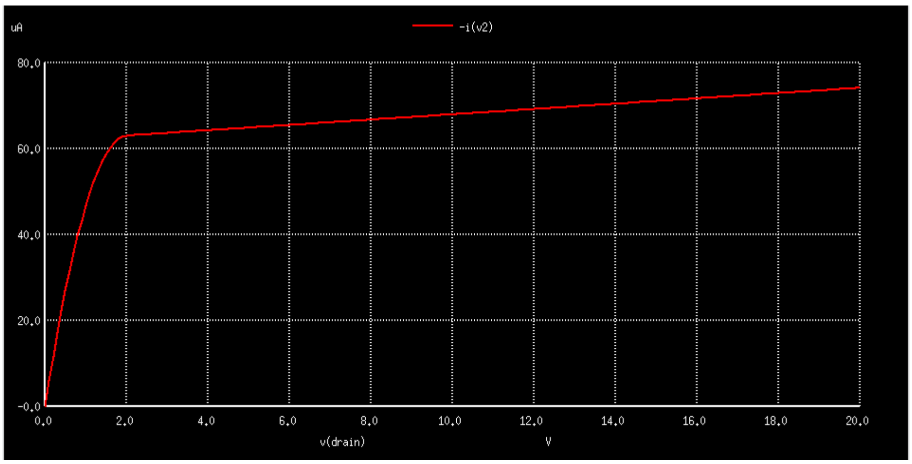

Figure 9.10: The drain characteristics of MOSFET with gate voltage =3V

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 differnt device model the characteristics would be slightly different.

The resultant characteristics is shown in the Figure 9.11.

RESULT

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

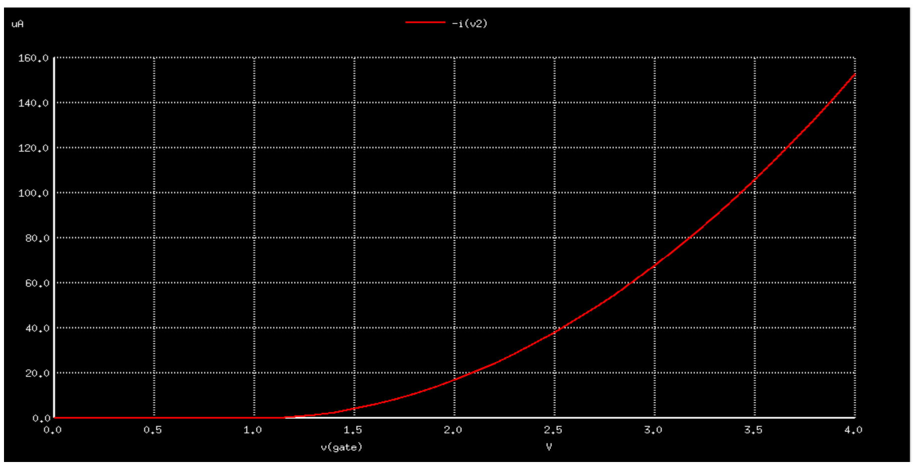

Figure 9.11: The transfer characteristics of MOSFET with drain voltage =10V

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 colector- 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 constsnt 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 collector-emitter 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 availbale 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

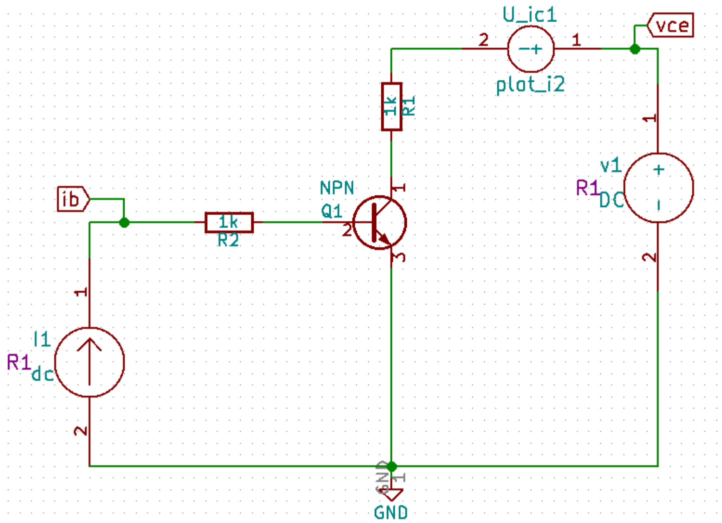

Figure 10.1: Schematic diagram for CE output characteristics

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.

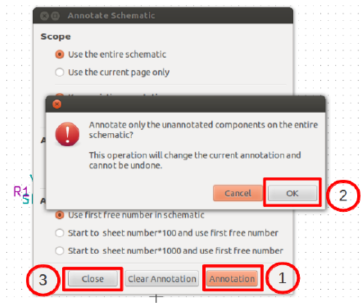

Figure 10.2: Annotation

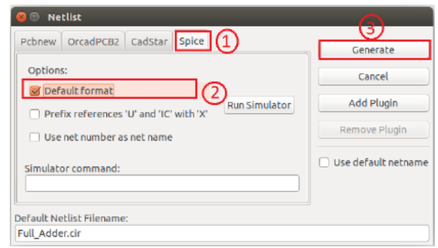

Figure 10.3: Netlist Generation

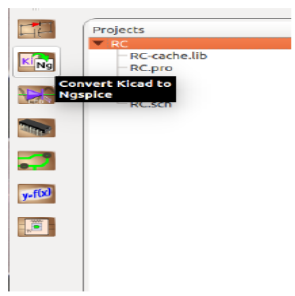

Figure 10.4: Choose Kicad to Ngspice tool

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

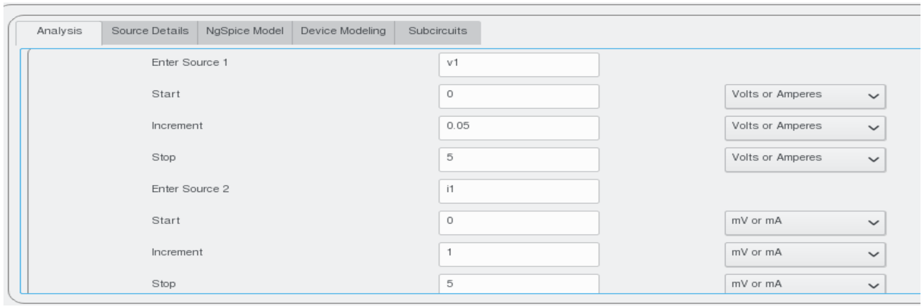

Figure 10.5: Choose DC analysis type and enter the values of V1 and I1

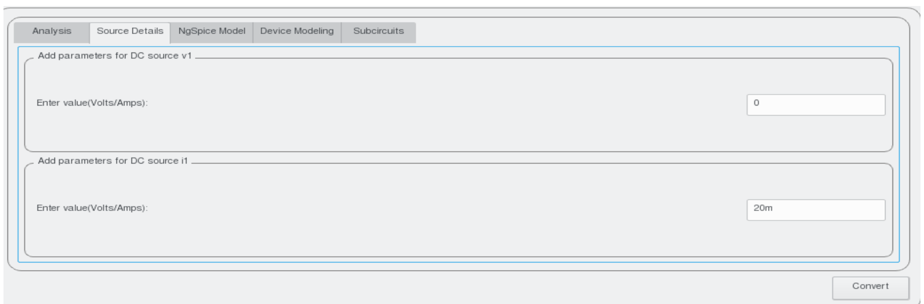

Figure 10.6: GiveSource Details of V1 and I1

RESULT

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

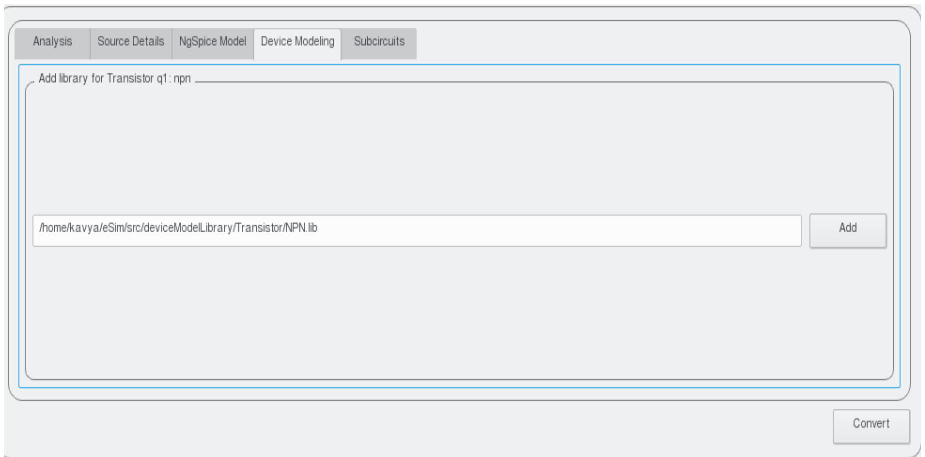

Figure 10.7: Choose the required NPN model

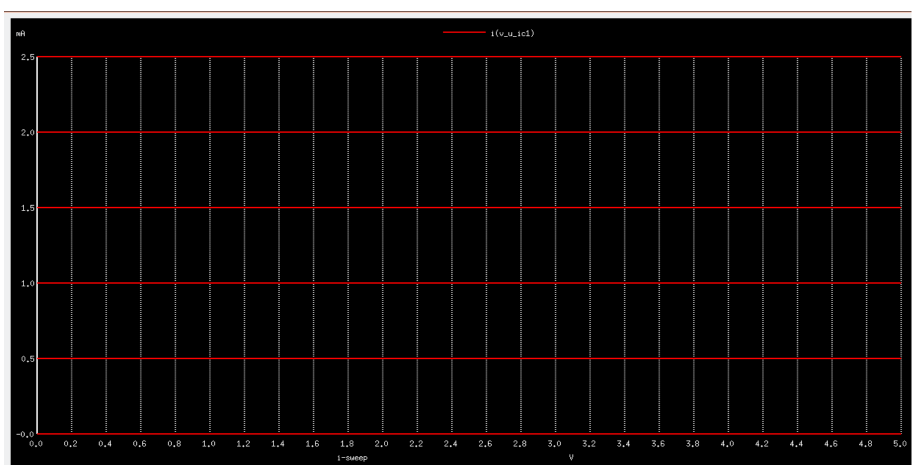

Figure 10.8: The output characteristics of NPN transistor

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 colector- baser 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 collector-base 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 availbale 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

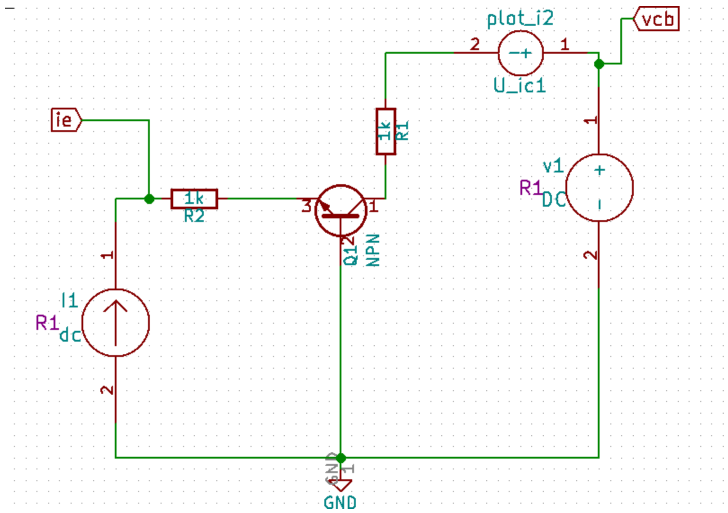

Figure 11.1: Schematic diagram for CE output characteristics

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.

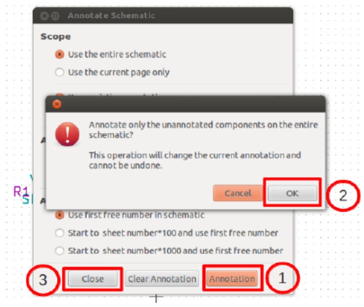

Figure 11.2: Annotation

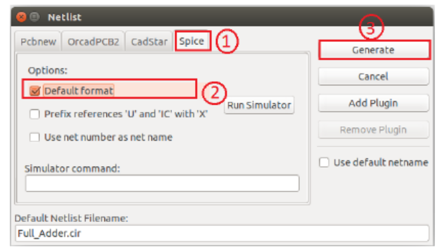

Figure 11.3: Netlist Generation

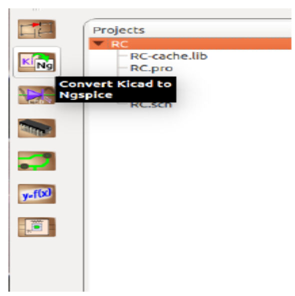

Figure 11.4: Choose Kicad to Ngspice tool

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

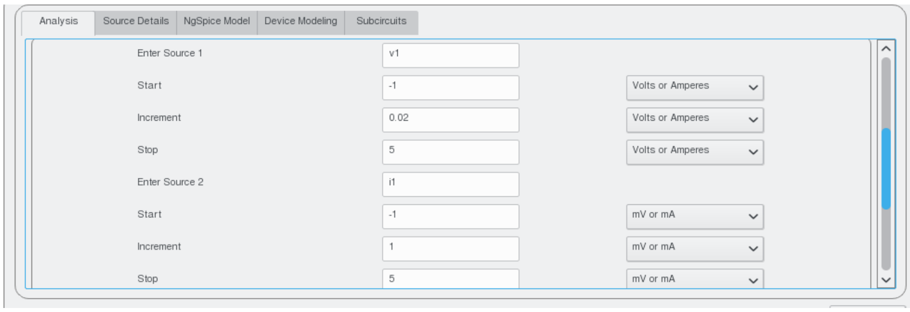

Figure 11.5: Choose DC analysis type and enter the values of V1 and I1

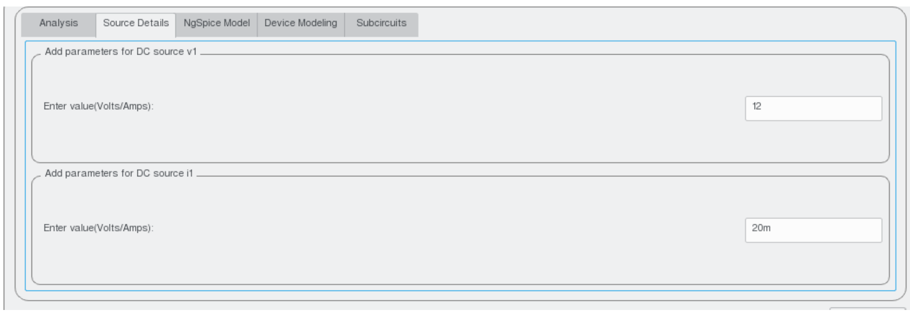

Figure 11.6: Give Source Details of V1 and I1

RESULT

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

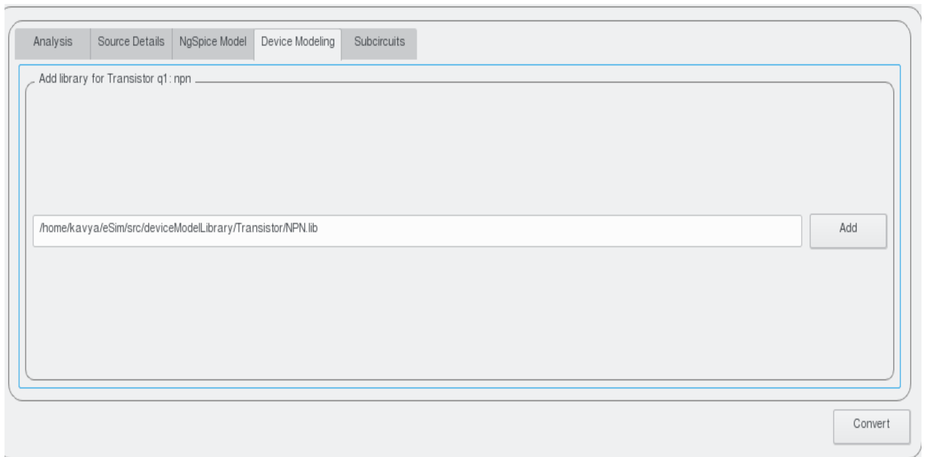

Figure 11.7: Choose the required NPN model

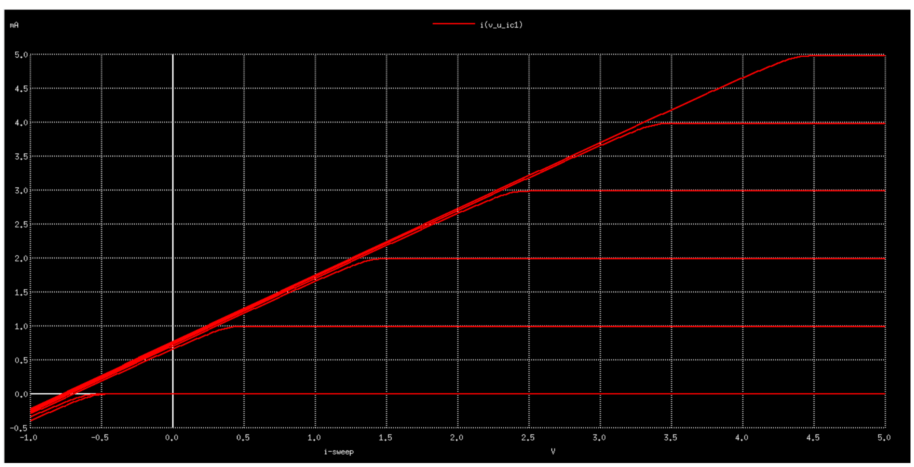

Figure 11.8: The output characteristics of NPN transistor

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 acroos 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.

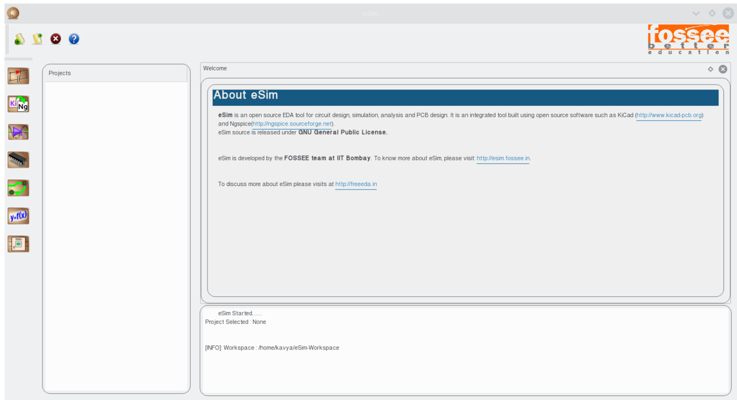

Figure 12.1: Launching eSim will take you to this window

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.

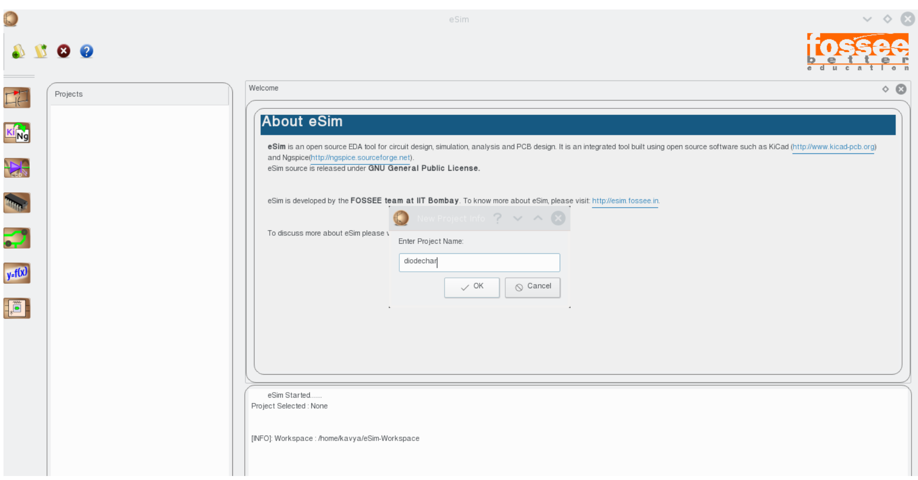

Figure 12.2: Creating new project

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.

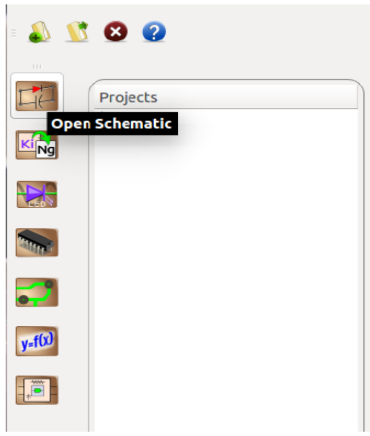

Figure 12.3: Creating new schematic diagram

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.

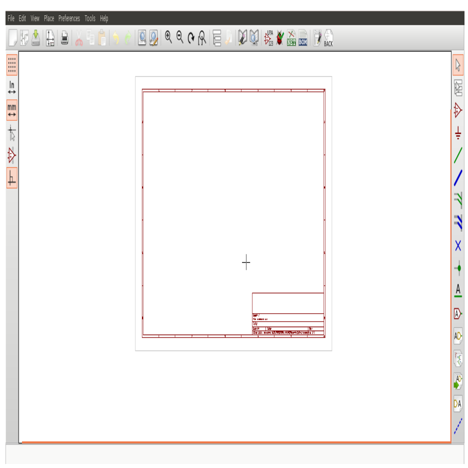

Figure 12.4: The Kicad Eeschema page

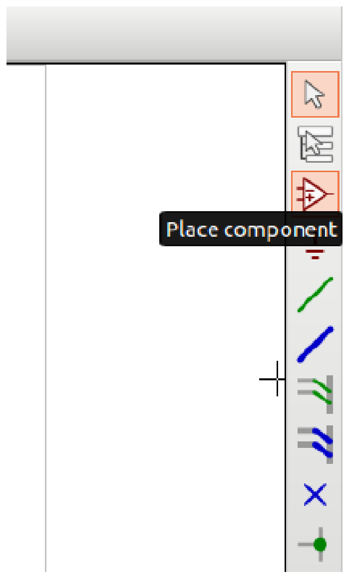

Figure 12.5: Place component icon

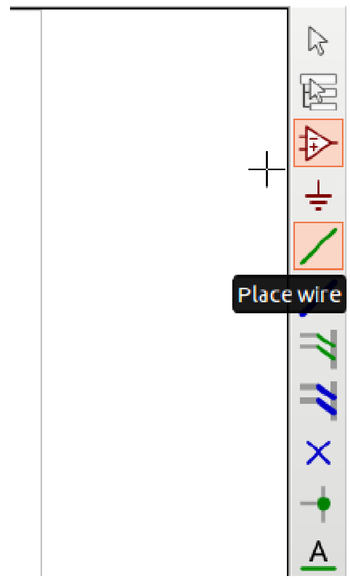

Figure 12.6: Place wire icon

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

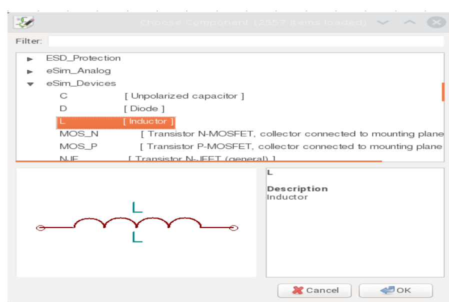

Figure 12.7: The Kicad Libraries of components

- 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.

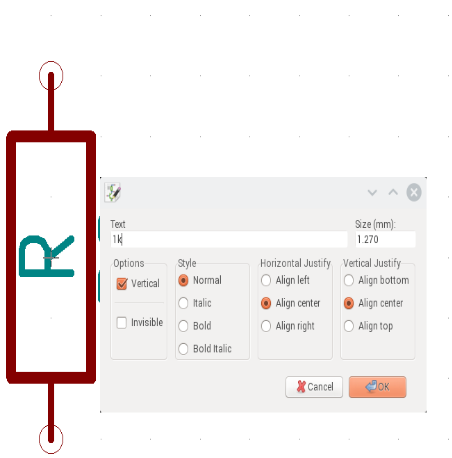

Figure 12.8: Editing the value field of component R

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.

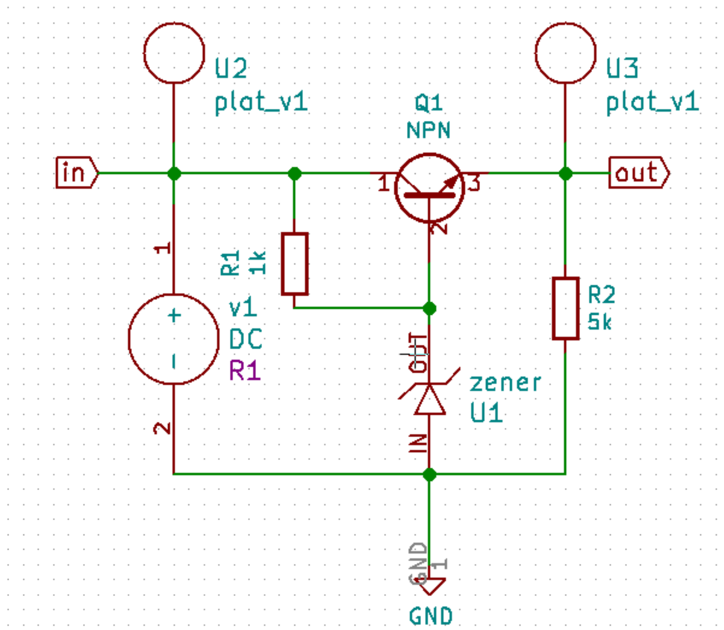

Figure 12.10: Schematic diagram for Zener Diode Regulator

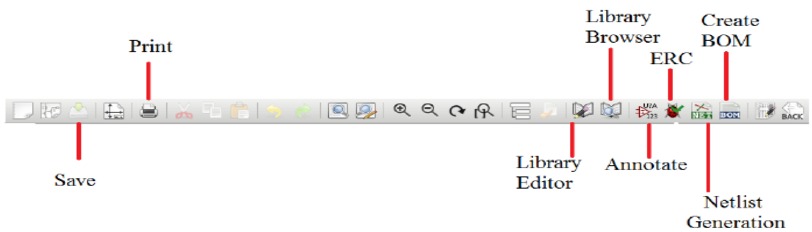

Figure 12.9: Choose annotate from the toop tool bar

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.

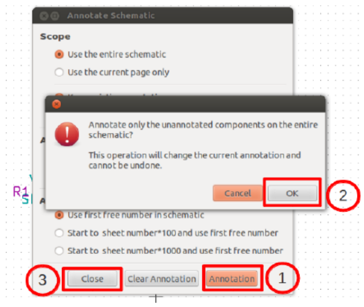

Figure 12.11: Annotation

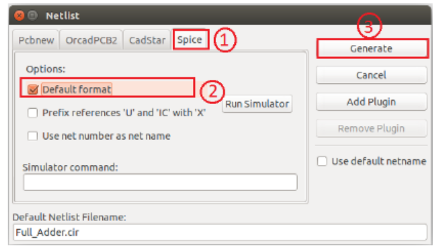

Figure 12.12: Netlist Generation

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.

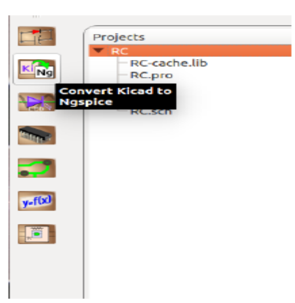

Figure 12.13: Choose Kicad to Ngspice tool

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.

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

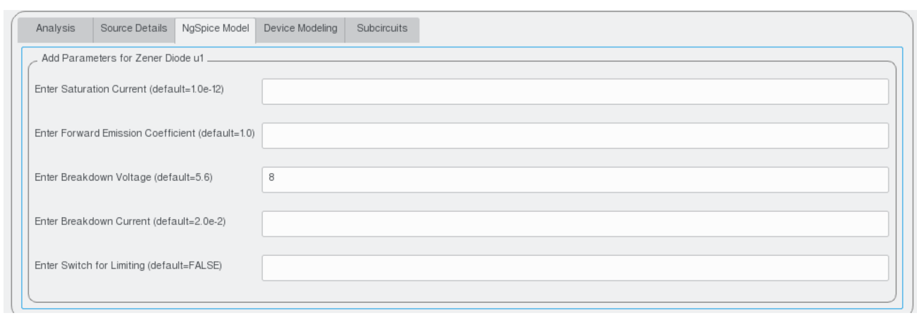

Figure 12.15: Choose ngspice model values

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

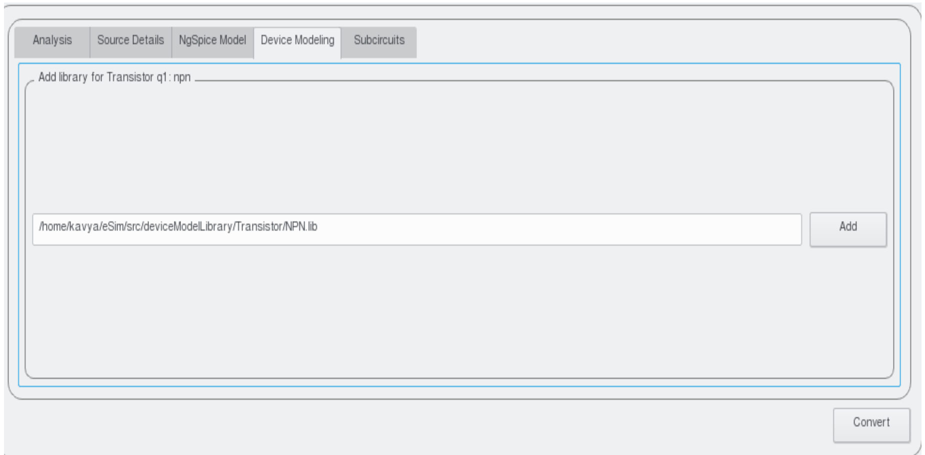

Figure 12.16: Choose the required Transistor model

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.

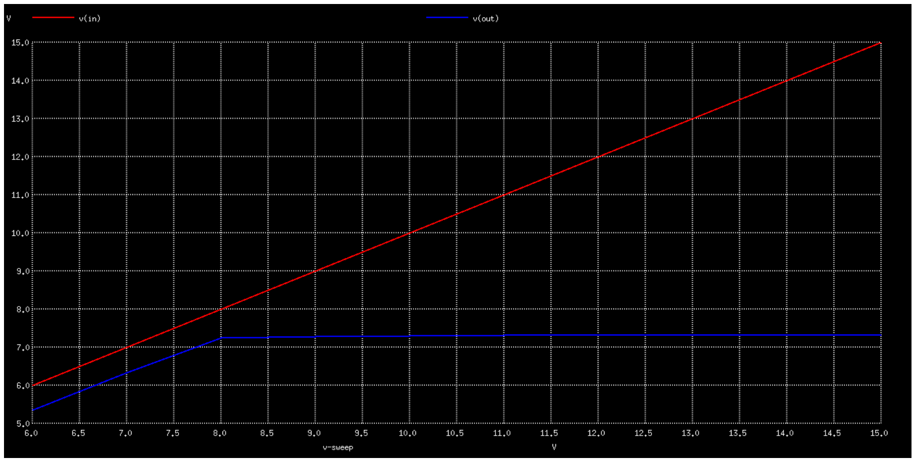

Figure 12.17: The line regulation characteristics of zener diode

RESULT

The circuit for plotting the charateristics 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