Chapter 11

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 11.10.

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 11.1.

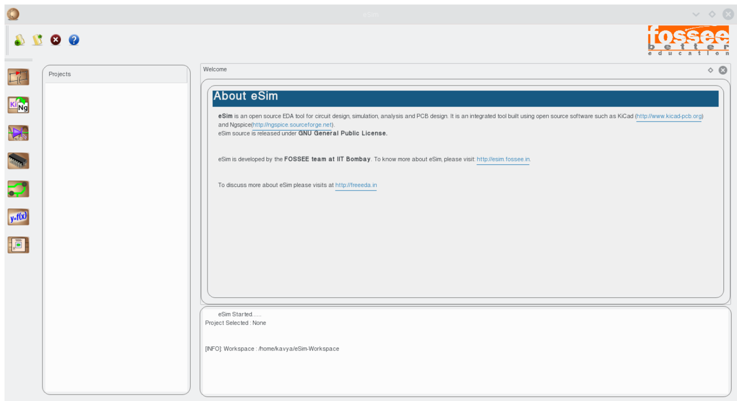

Figure 11.1: Launching eSim will take you to this window

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.11.2.

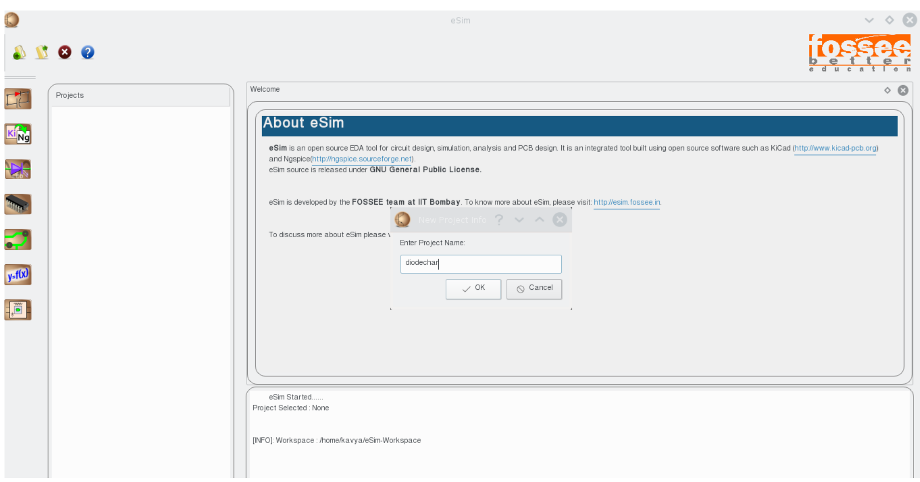

Figure 11.2: Creating new project

Create the Schematic

To create the schematic, click the very first icon of the left toolbar as shown in the Figure 11.3 .This will open KiCad Eeschema.

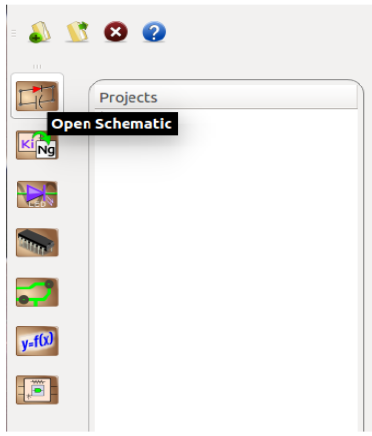

Figure 11.3: Creating new schematic diagram

To create a schematic in KiCad, we need to place the required components. See Figure 11.4. Figure 11.5 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 11.6. Scroll up and down for zooming in and out.

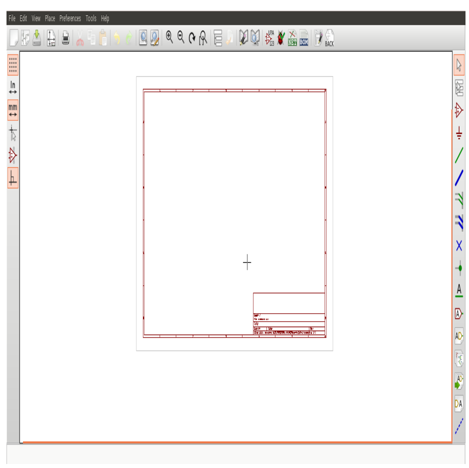

Figure 11.4: The Kicad Eeschema page

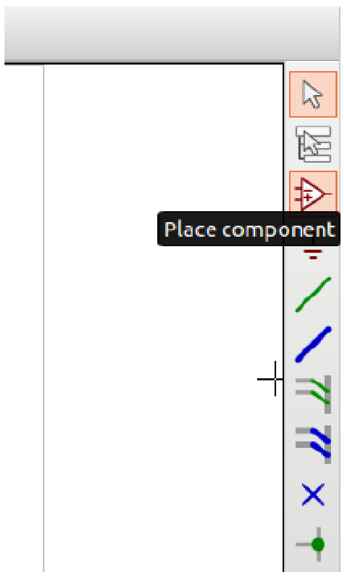

Figure 11.5: Place component icon

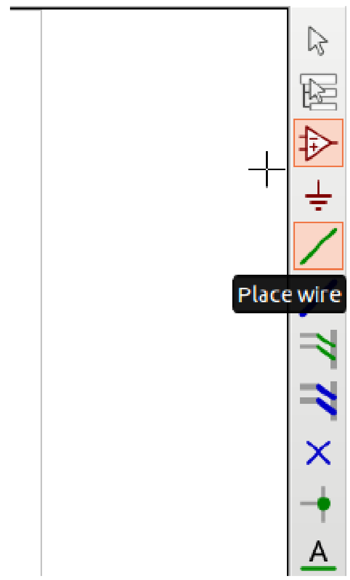

Figure 11.6: Place wire icon

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 11.7.

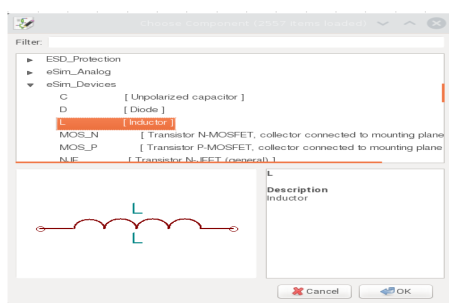

Figure 11.7: The Kicad Libraries of components

- 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 11.8.

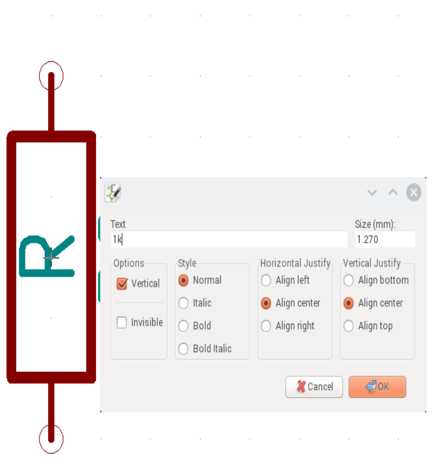

Figure 11.8: Editing the value field of component R

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 11.9 and in the subsequenct dialogue boxes appearing click ok and finally close. See Figure 11.11.

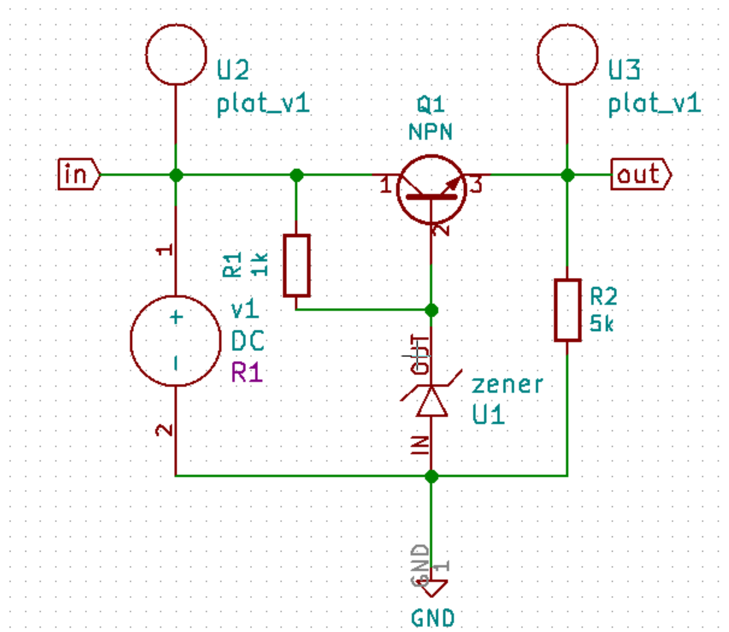

Figure 11.10: Schematic diagram for Zener Diode Regulator

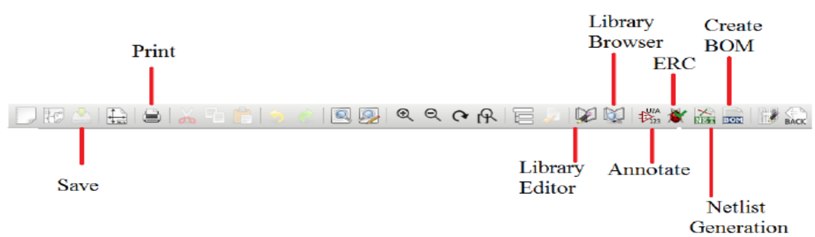

Figure 11.9: Choose annotate from the toop tool bar

Now we have the circuit diagram as shown in Figure 11.10.

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.

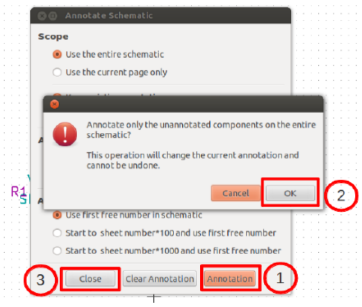

Figure 11.11: Annotation

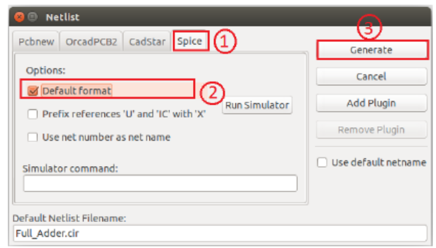

Figure 11.12: Netlist Generation

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 11.12. 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 11.13. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

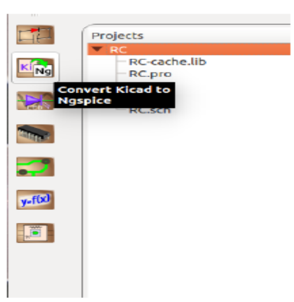

Figure 11.13: Choose Kicad to Ngspice tool

Analysis: Choose DC analysis type. Give the values of DC variables as shown in Figure 11.14. 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 11.14: 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.11.15

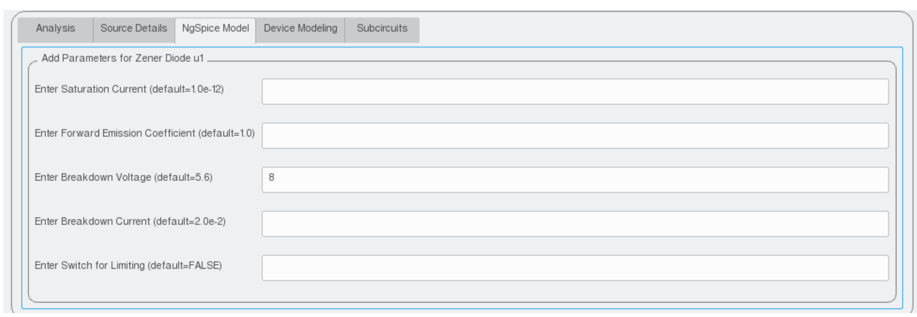

Figure 11.15: Choose ngspice model values

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 11.16.

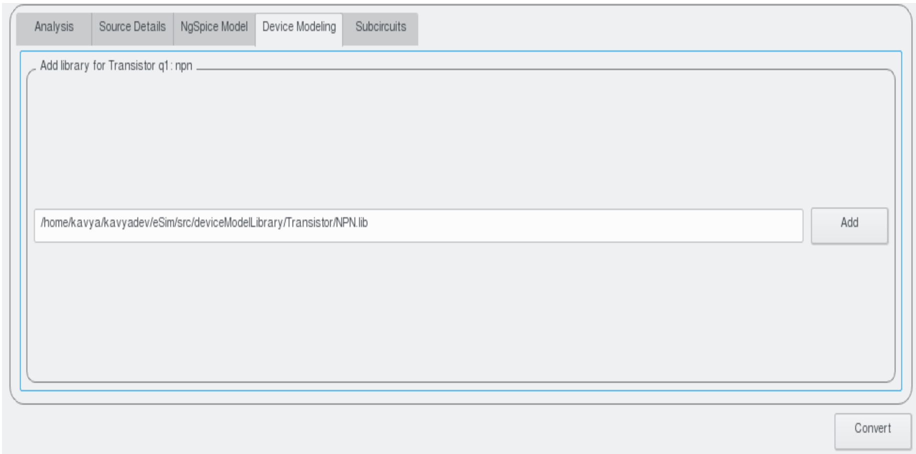

Figure 11.16: Choose the required Transistor model

Subcircuits: No subcircuits to be given.

Once these details are provided click on convert button. See Figure 11.16. 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 11.17.

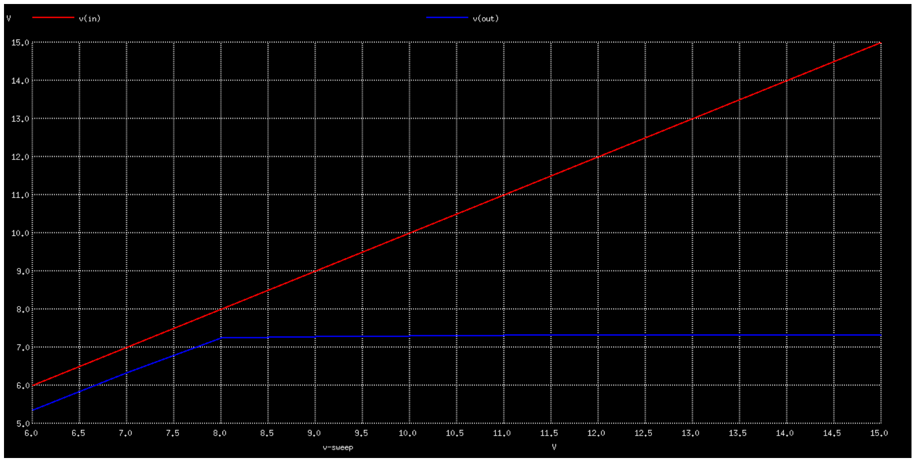

Figure 11.17: The line regulation characteristics of zener diode

RESULT

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