

Chapter 2

Diode Characteristics

AIM

To design and implement a circuit for simulating 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 which may be varied during simulation. The diode in the circuit should be associated with a coresponding ‘Diode model’ during simulations. As in a hardware circuits lab, a curenrt 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 below:

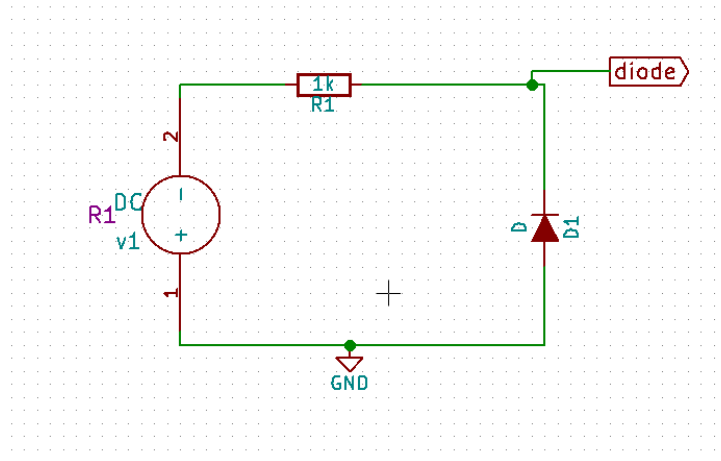


Figure 2.1: Schematic diagram for diode characteristics

PROCEDURE

Launch eSim

Launching eSim will take you to the dialog box which asks for the default workspace. Browse the folders and set the workspace location. It will finally end up in the eSim window shown in Figure 2.2.

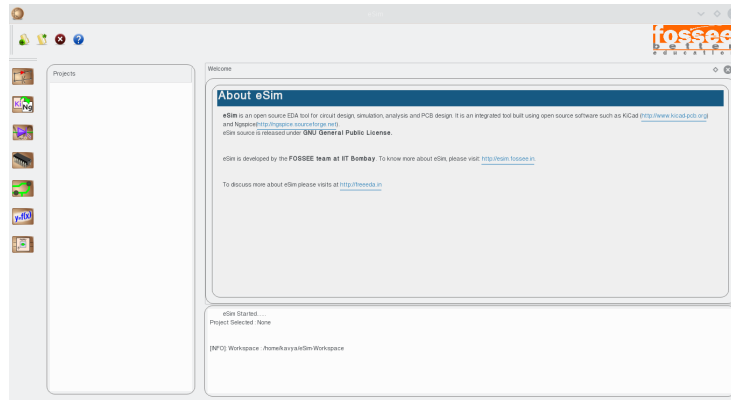


Figure 2.2: Launching eSim will take you to this window

Create a New Project

The new project is created by clicking the New icon on the menubar. The name of the project is given in the pop up window as shown in Figure 2.3.

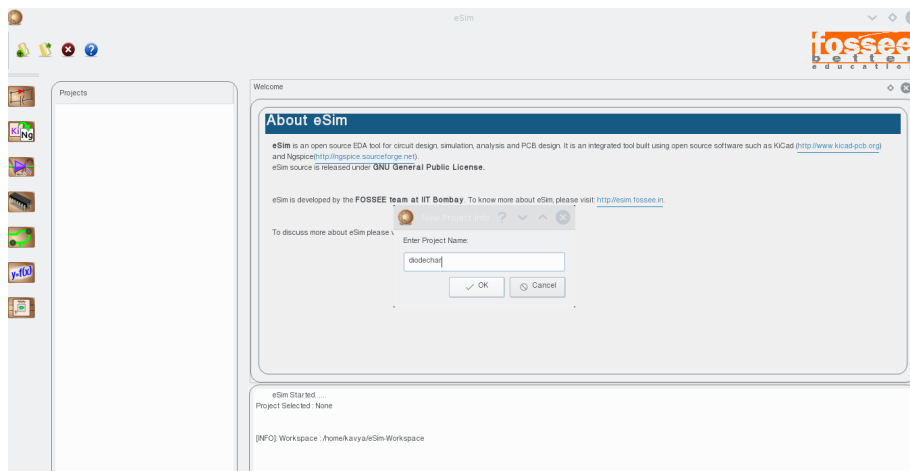


Figure 2.3: Creating new project

Create the Schematic

To create the schematic, click the very first icon of the left toolbar as shown in the Figure 2.4 .This will open KiCad Eeschema.

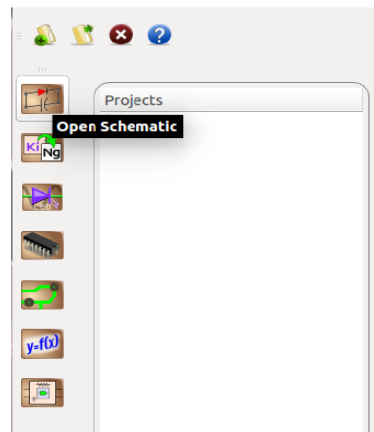


Figure 2.4: Creating new schematic diagram

To create a schematic in KiCad, we need to place the required components. See Figure 2.5

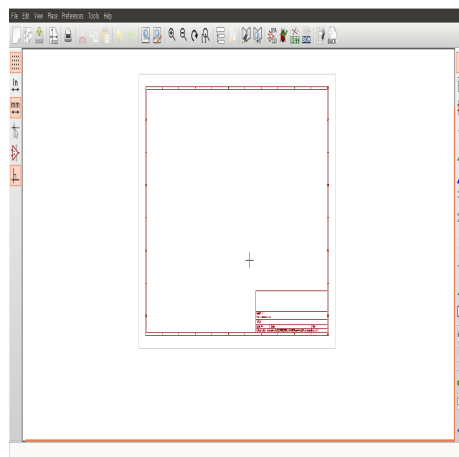


Figure 2.5: The Kicad Eeschema page

Figure 2.6 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 2.7. Scroll up and down for zooming in and out.

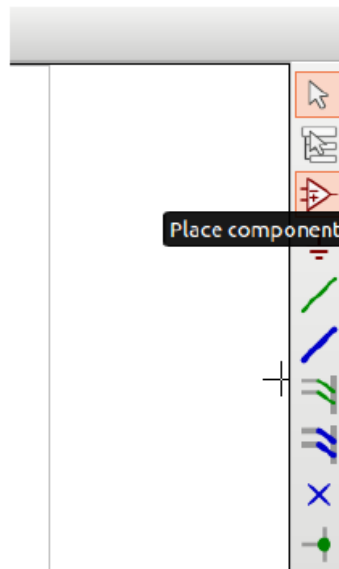


Figure 2.6: Place component icon

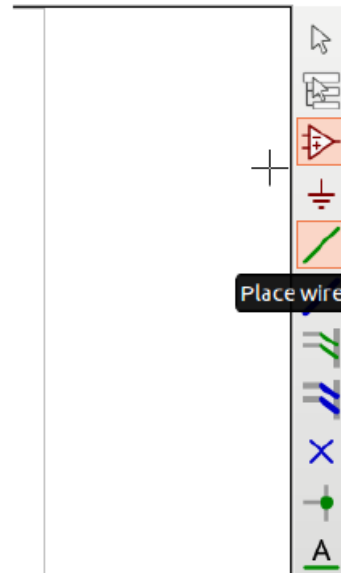


Figure 2.7: Place wire icon

Placing the Components: Normally all the components available in eSim can be chosen by left mouse click in the grid. The components are listed in different libraries. See Figure 2.8.

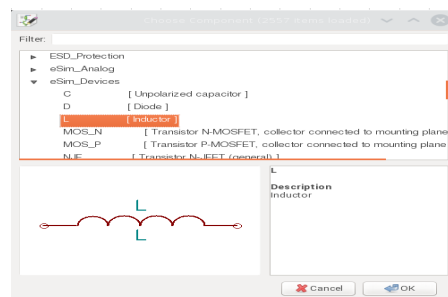


Figure 2.8: The Kicad Libraries of components

- 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 2.9.

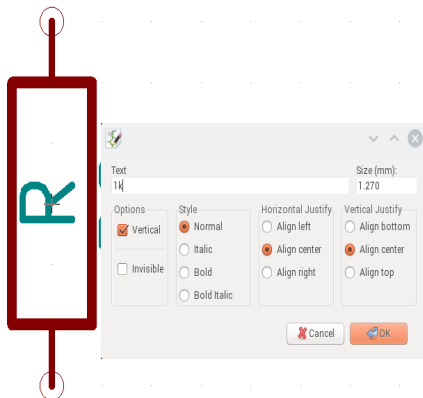


Figure 2.9: Editing the value field of component R

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 2.10 and in the subsequent dialogue boxes appearing click ok and finally close. See Figure 2.11).



Figure 2.10: Choose annotate from the top toolbar

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

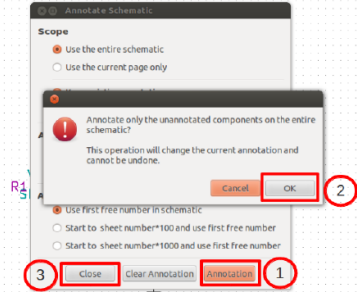


Figure 2.11: Annotation

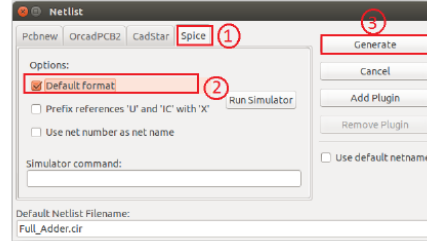


Figure 2.12: Netlist Generation

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 2.12. Now the netlist is ready to be simulated.

KiCad to Ngspice conversion

To convert KiCad netlist of RC circuit to NgSpice compatible netlist click on KiCad to Ngspice icon as shown in Figure 2.13. Now you can choose the type of analysis, source details, device models ngspice models and subcircuit models.

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.

Source Details: Leave this empty.

Ngspice Model: No Ngspice model to be given.

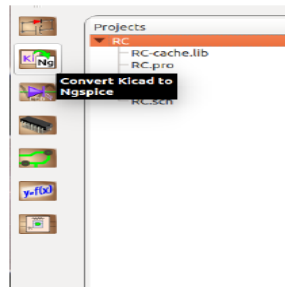


Figure 2.13: Choose Kicad to Ngspice tool

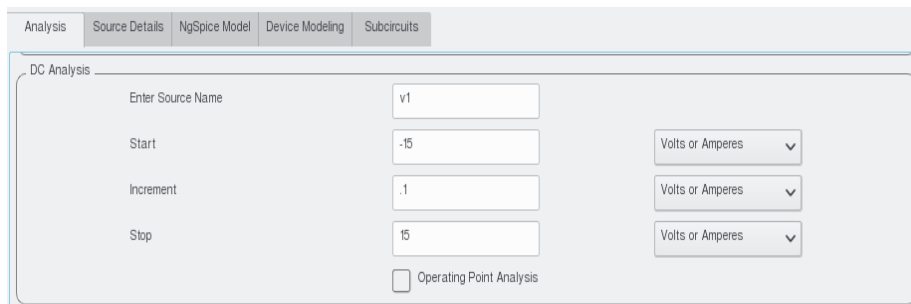


Figure 2.14: Choose DC analysis type and enter the values

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

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. In order 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)
```

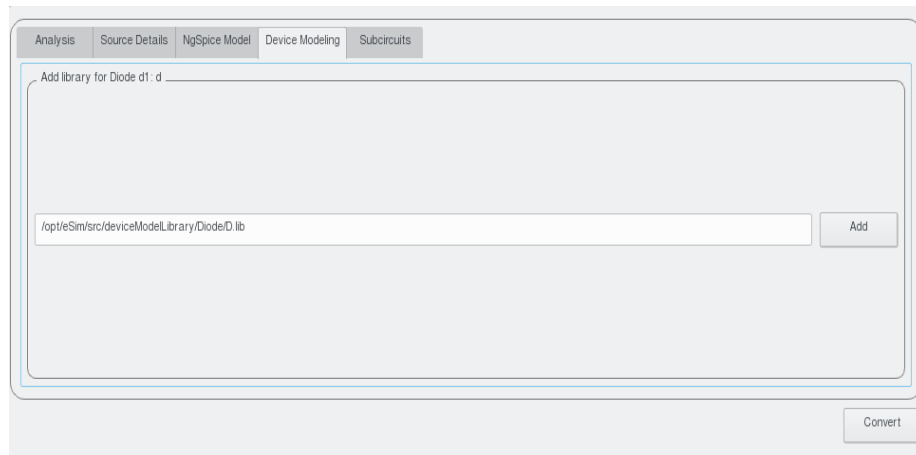


Figure 2.15: Choose the required diode model

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

The resultant characteristics is shown in the Figure 2.16.

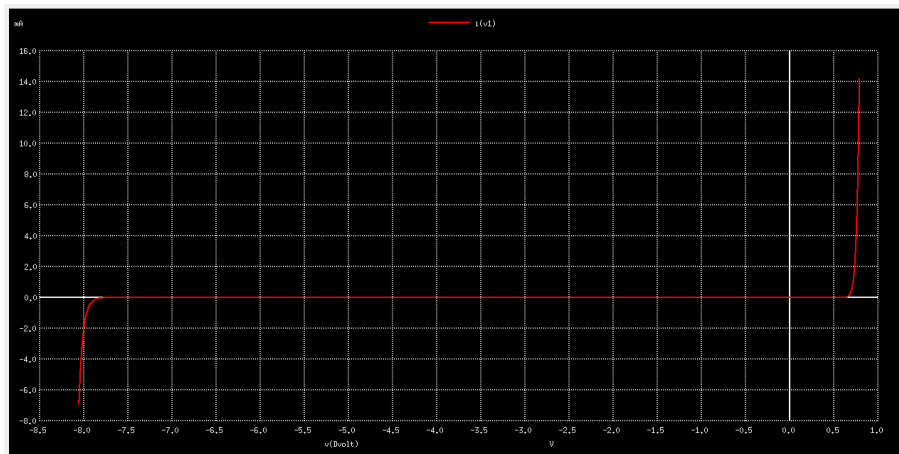


Figure 2.16: The characteristics of Diode

RESULT

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