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

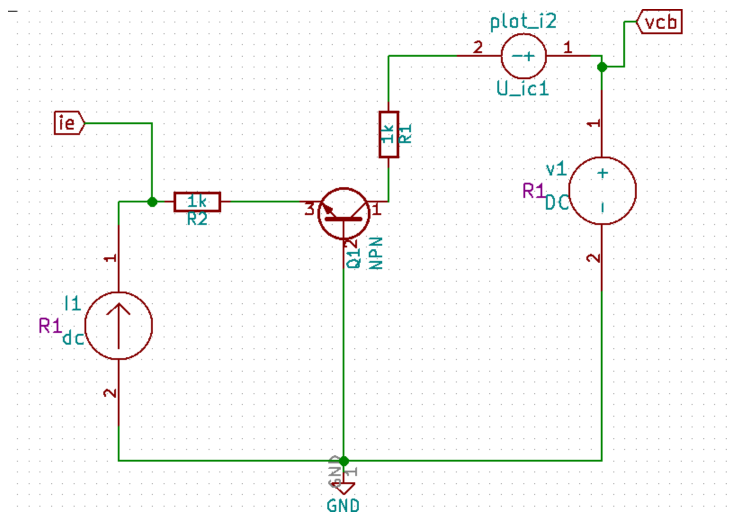

Figure 11.1: Schematic diagram for CE output characteristics

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.

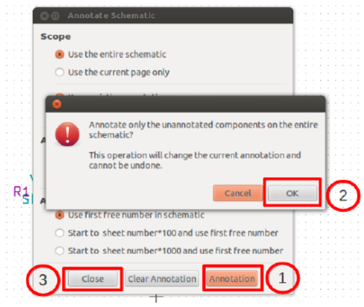

Figure 11.2: Annotation

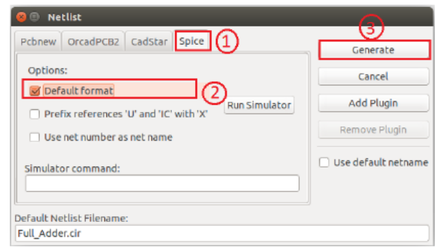

Figure 11.3: Netlist Generation

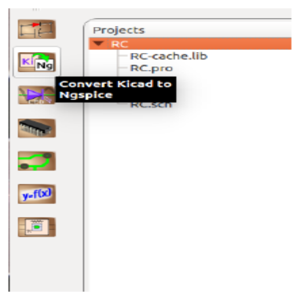

Figure 11.4: Choose Kicad to Ngspice tool

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

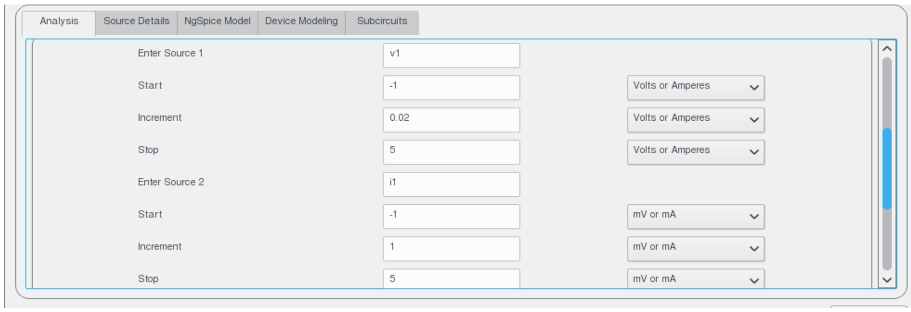

Figure 11.5: Choose DC analysis type and enter the values of V1 and I1

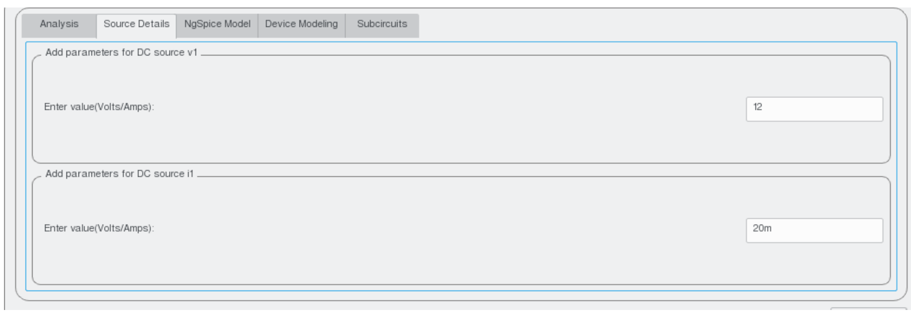

Figure 11.6: Give Source Details of V1 and I1

RESULT

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

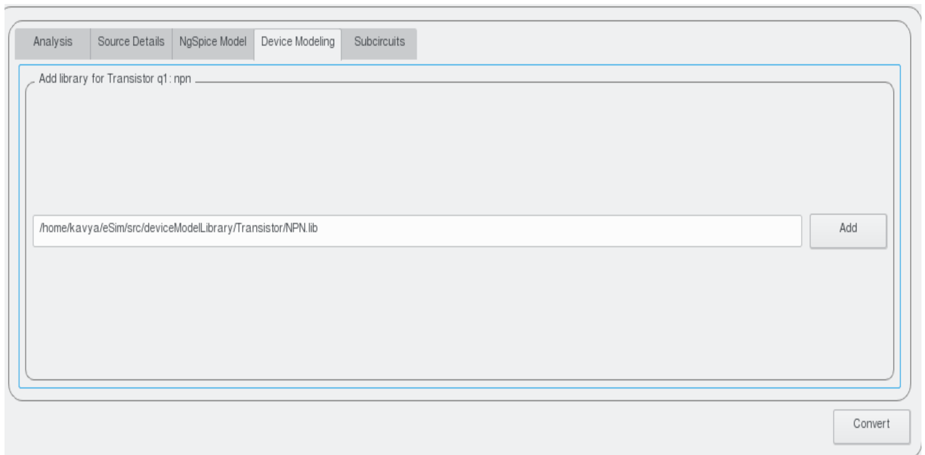

Figure 11.7: Choose the required NPN model

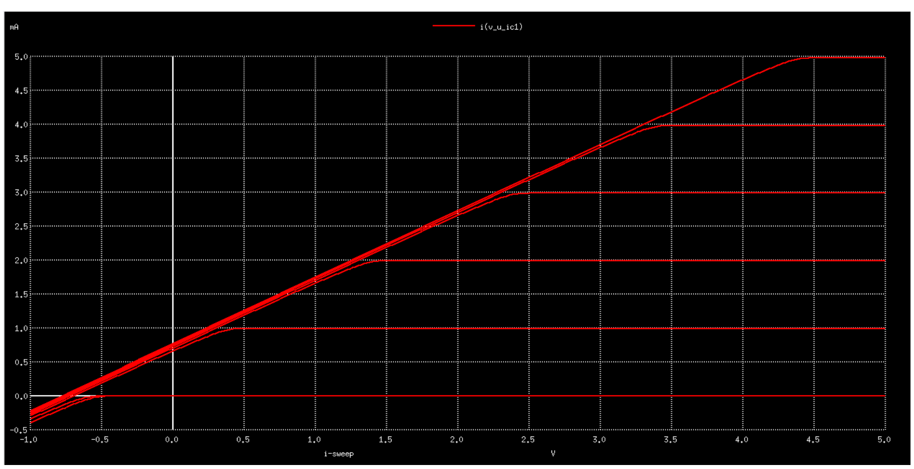

Figure 11.8: The output characteristics of NPN transistor