

Ex. No. 10	CHARACTERISTICS OF BIPOLAR JUNCTION TRANSISTOR (BJT)
Date:	

AIM:

To obtain the input and output ($V - I$) characteristics of a BJT in Common Emitter Configuration and to plot the characteristics.

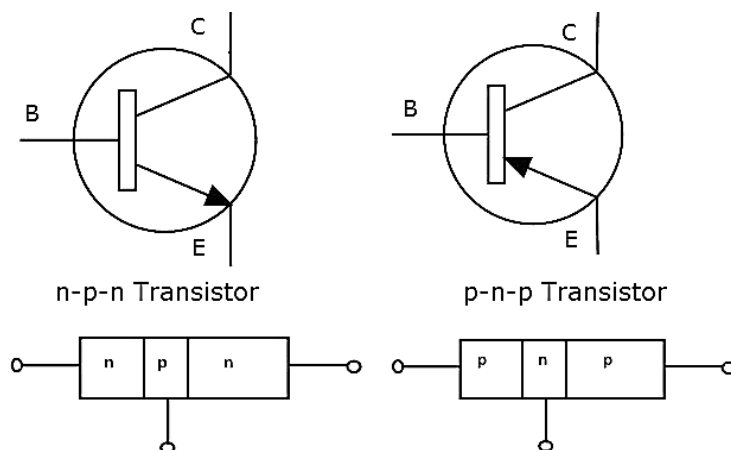
SOFTWARE REQUIRED

LT spice

THEORY:

Bipolar junction transistor is a two-junction, three-layer, three-terminal semiconductor device. The two junctions give rise to three regions: emitter, base, and collector. The emitter region is more heavily doped than any of the other regions because its main function is to supply the majority of charge carriers to the base. The base forms the middle section, which is very thin and is very lightly doped. The collector's main function is to collect the majority of charge carriers coming from the emitter and passing through the base.

Symbol and Structure of a Transistor

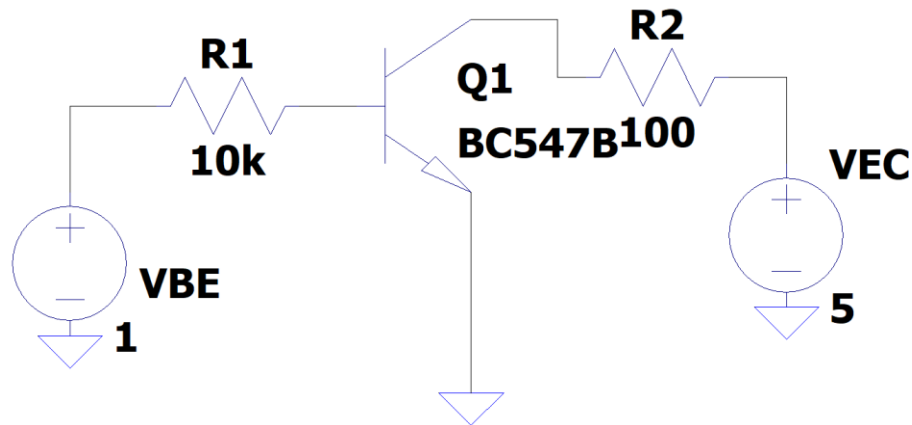


A transistor can be used in three different configurations. Out of these three configurations, the common emitter (CE) configuration is the most preferred. Emitter base junctions should be forward biased, and collector base junctions should be reverse biased. In CE configuration, an input signal is applied between the base and emitter, and an output signal is taken from the collector and emitter circuit.

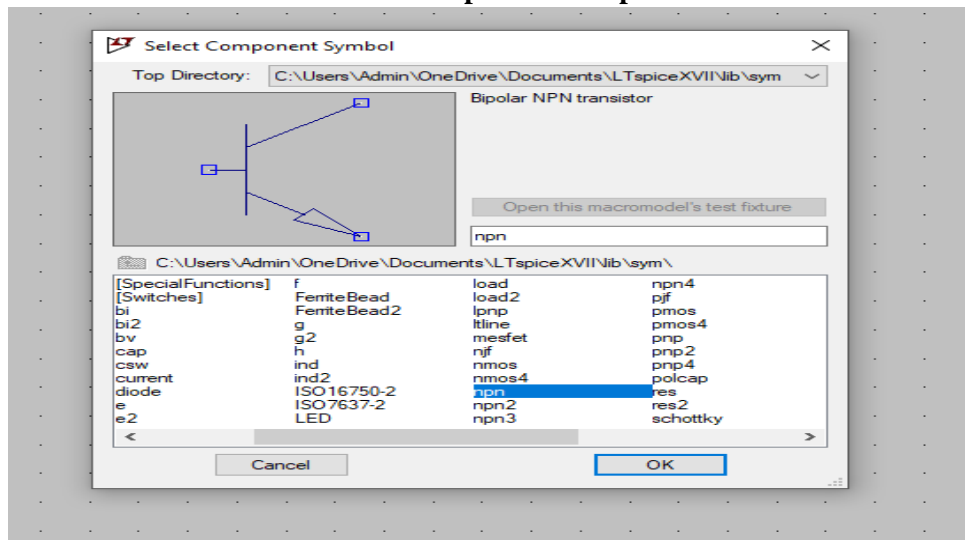
PROCEDURE

1. Construct the given circuit in LT spice

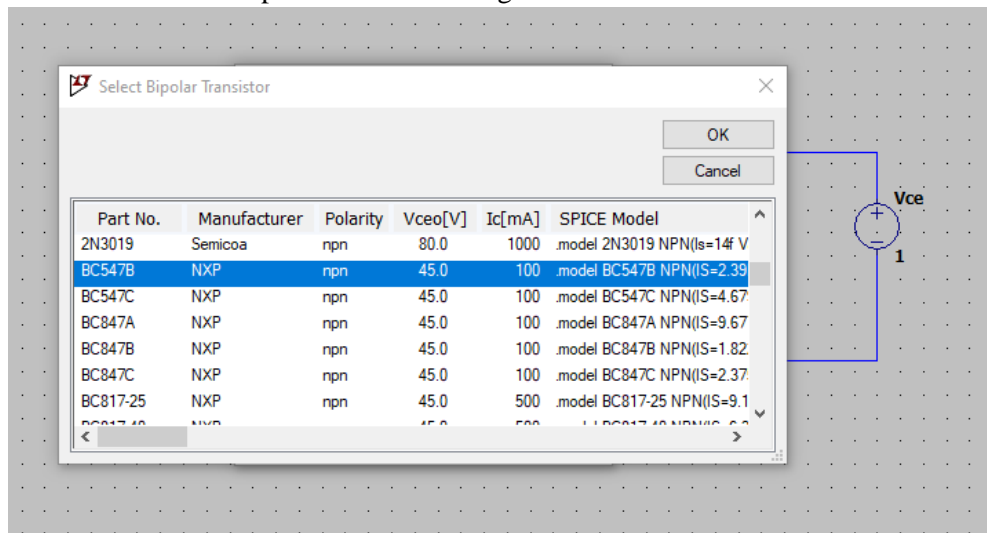
.dc VEC 0 20 2 VBE 500m 4000m 500m



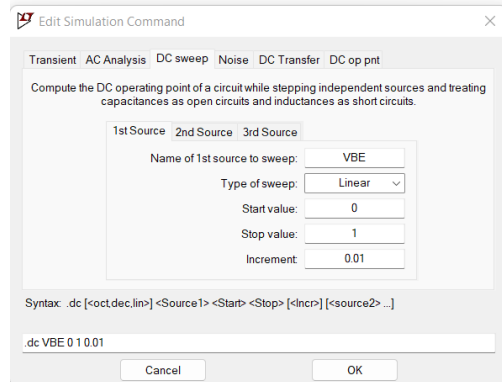
- To select the transistor -> **components** -> **npn**



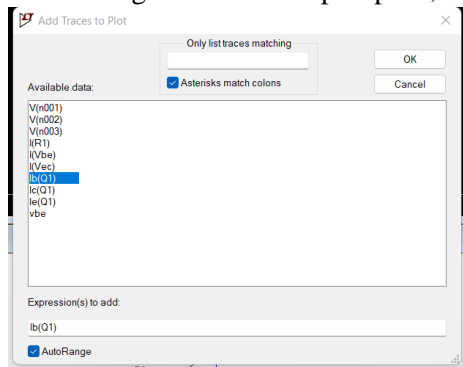
- To select the particular transistor go to **Pick new Transistor** and select **BC547B**



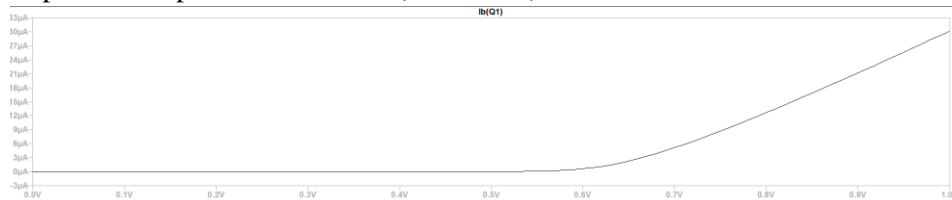
2. To plot the input characteristics
 - i. Set $V_{EC} = 5V$ constant
 - ii. Set $V_{BE} = 1V$
 - iii. Go to Simulation -> **Edit simulation cmd** -> **DC sweep**
 - iv. Select first source and enter the following values



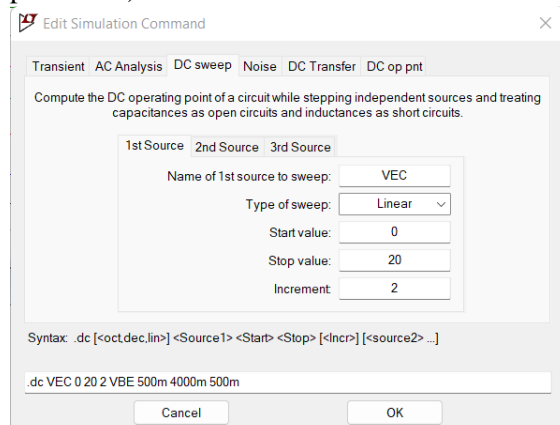
- v. Click ok and Run the simulation
- vi. Right click on the plot pane, select Add trace, select **Ib** and click ok



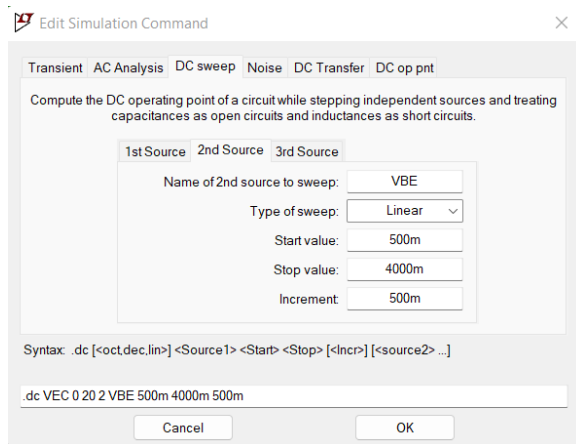
- vii. Capture the input characteristics (V_{BE} vs I_b)



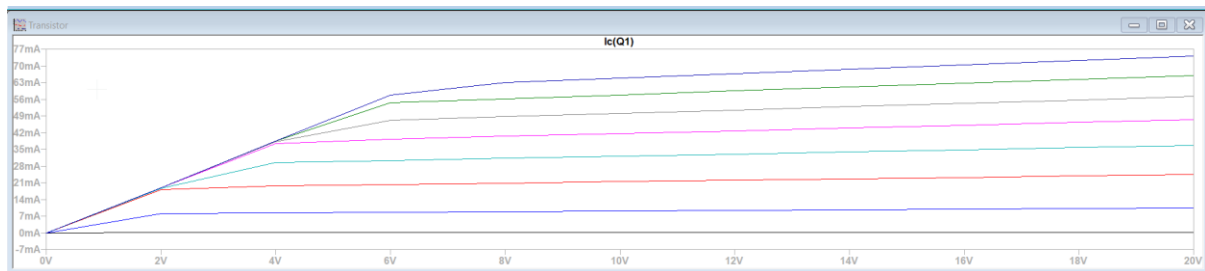
3. To plot the output characteristics
 - i. Go to Simulation -> Edit simulation cmd -> DC sweep
 - ii. Select first source and enter the following values V_{EC} (First source will act as x-axis parameter)



- iii. Select the second source and enter the values as shown in the image



- iv. Click ok and run the simulation
- v. Right click on the plot pane, select Add trace, select Ic and click ok



RESULT:

Thus the transistor characteristics are simulated using LTspice.