XMUT204 Electronic Design

Demo 5.2 – BJT Testing Simulation in LTspice

Introduction

Understanding the characteristics and operations of the bipolar junction transistor (BJT) is difficult and taking some time to master. But, simulation and experimentation with the device can make it easier for you to learn.

As a result, this exercise will be about setting up a circuit for testing the BJT transistor and to obtain the properties and characteristics for NPN transistors from simulation in LTspice (this works with PNP too, but circuit setup is different).

Three group of properties and characteristics are covered in this exercise e.g. DC and AC gains and current-to-voltage characteristics of the BJT amplifier. The transistor to be used for the exercise is 2N3904 BJT transistor which is also available in the library of LTspice software.

1. Testing of BJT Characteristics (Vce vs Ic, Beta and Alpha parameters)

Here is a general look at how this characteristic of the BJT (Ic vs Vce/Vcc) is visualized:

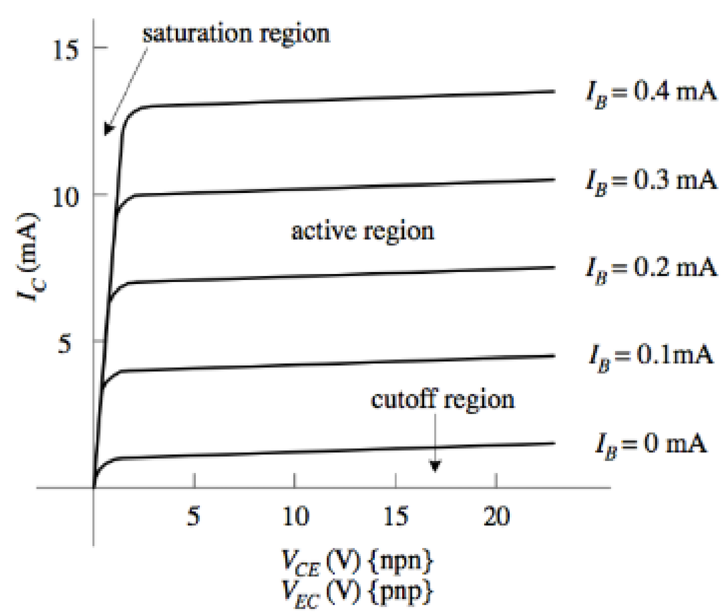


Figure 1: Vce (Vcc) vs. Ic characteristics of BJT transistor

Generally, 10 uA steps are used to visualize how the output current (Ic) is amplified. The output is typically measured in a range of milliamps.

These characteristics typically change (although follow the same pattern) based on several values based on the transistor's leg names: collector, emitter, and base measurements (voltage and/or current).

Also, a contributing factor is the current gain called Beta (β). There is another factor called Alpha (α), but can be calculated from Beta by knowing Ie = Ic + Ib.

$$\beta = Ic/Ib$$

Conversely, Alpha can be calculated from:

$$\alpha = Ie/Ic = \beta / (1+ \beta)$$

Beta values are typically listed in the datasheet of bipolar transistors. The values above are important to circuit designs with transistors involved.

For obtaining the characteristics graph of the BJT transistor, a biasing circuit needs to be constructed as shown in the following diagram. The circuit shows a common emitter biasing configuration of the BJT transistor.

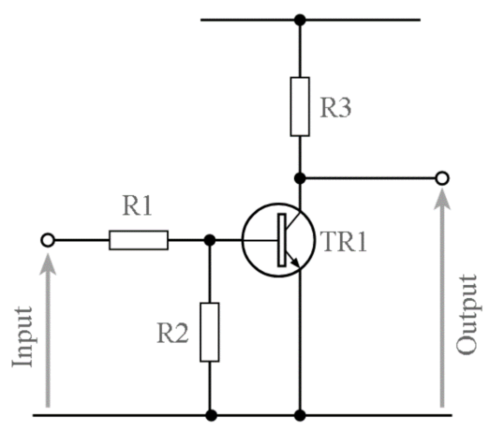


Figure 2: Common emitter biasing circuit of BJT transistor

2. Creating Schematic

In LTspice software, in order to get the same characteristics graph as shown in Fig. 1 (very essential info to have for design of specific transistors, models change based on what parameters they have in LTspice), it is suggested using a testing circuit like shown in the following diagram.

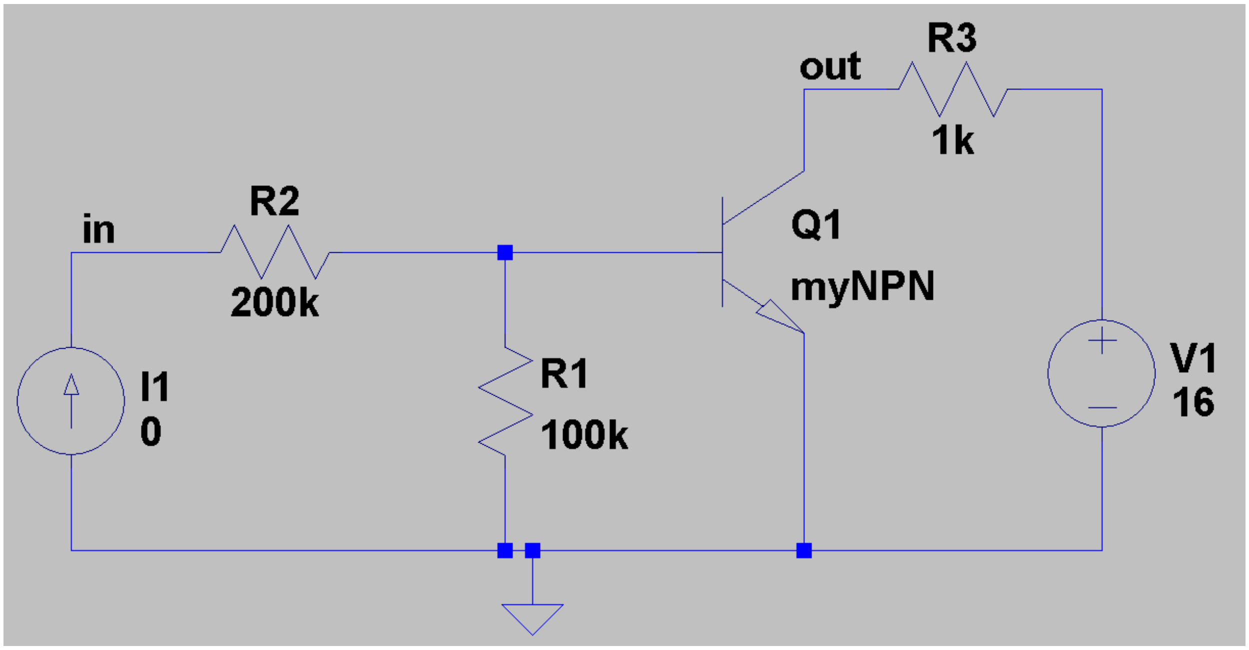


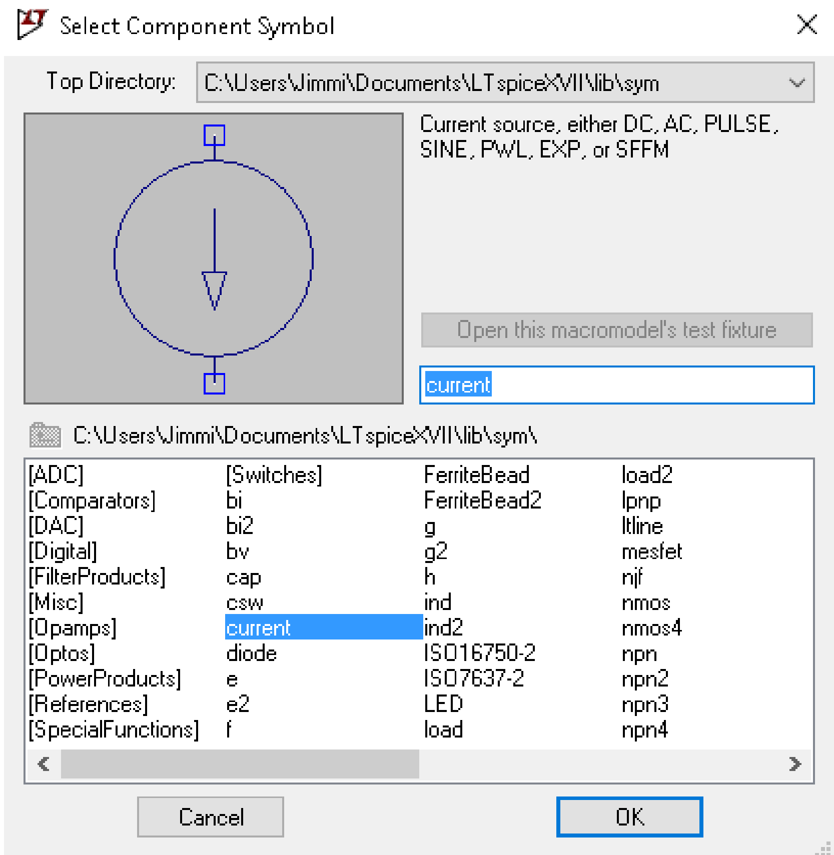
Figure 3: Testing circuit of BJT transistor

First, add the resistors (i.e. R1, R2 and R3), voltage source (V1) and ground as shown in the circuit above and place them to the schematics in LTspice.

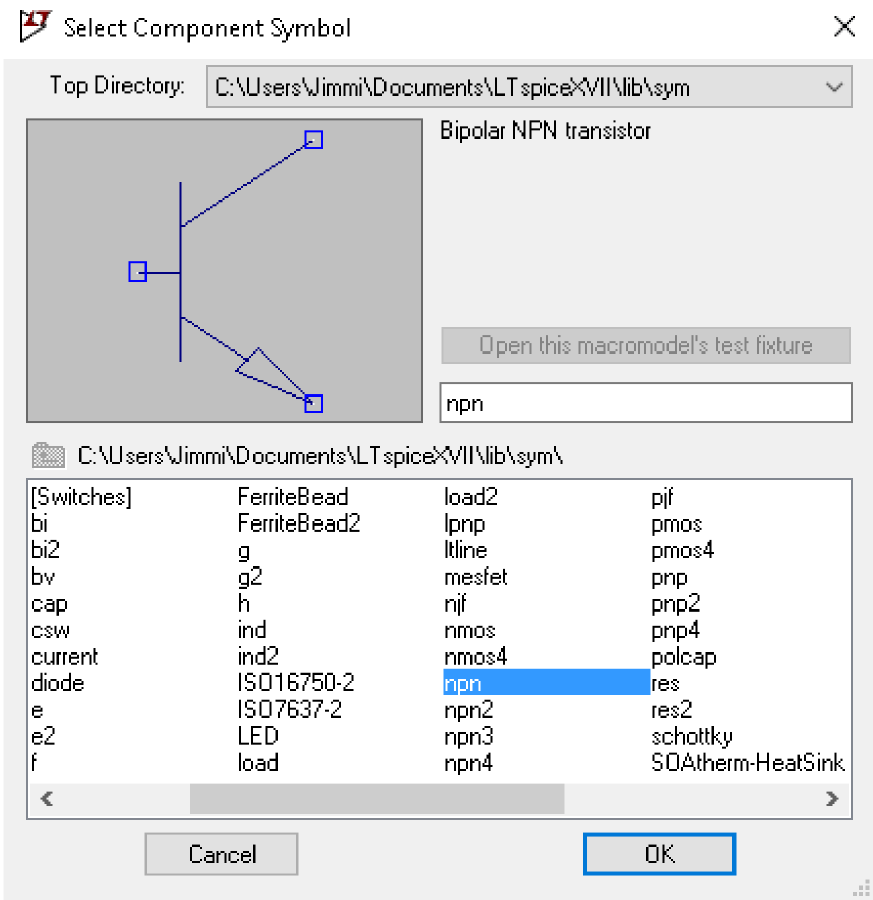
Configure the devices, so R1 = 100k Ω, R2 = 200k Ω and R3 = 1k Ω. Set the voltage source V1 to be 16 V.

Wire these devices in the schematic so they are connected with one to another.

Beside those components described above, there are two parts that should be also added into the schematic: the current source I1 and transistor Q1 (i.e. this transistor is based on a modified 2N3904 BJT transistor model in LTspice).



Search for “current” in the component library and place the device where it is located in the circuit. Make sure to rotate the component so that the arrow is facing up, as that is the current direction that we want.

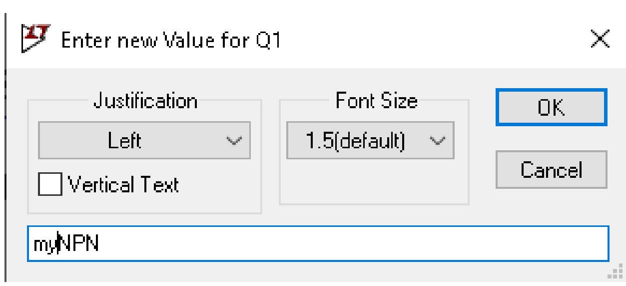


Search for “npn” in the component library and place/rotate it as seen in the diagram above.

The values of the resistor on the left of the circuit (200kΩ and 100kΩ) are for reducing input current to microampere levels.

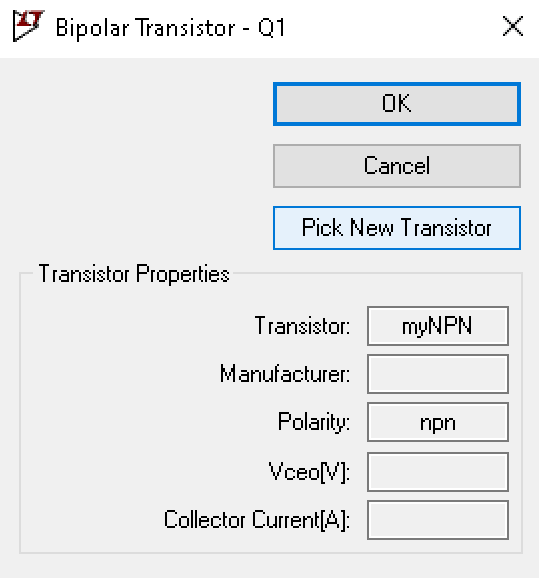
The input current is connected to the base leg of the NPN transistor.

Next, change the name of the transistor (not Q1, but the text below that) by right-clicking it.

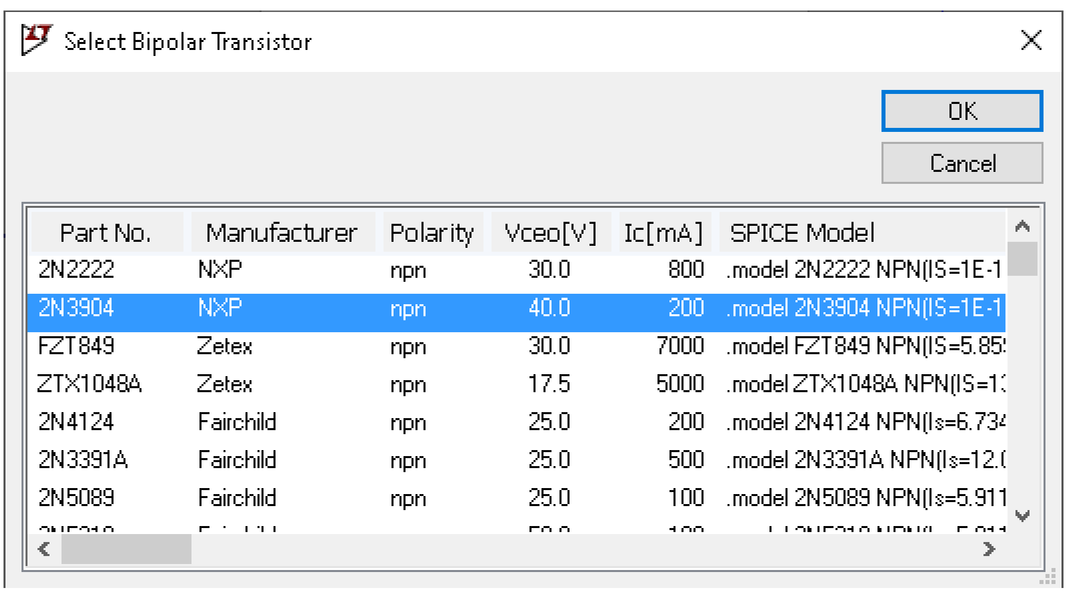


Name the transistor something unique and easy, just use “myNPN” by default. This custom name will be used in a particular SPICE command that lets the transistor behave as any model in the program’s library, but can be edited quickly.

Before adding the command, right-click the transistor and press the “Pick New Transistor” button.

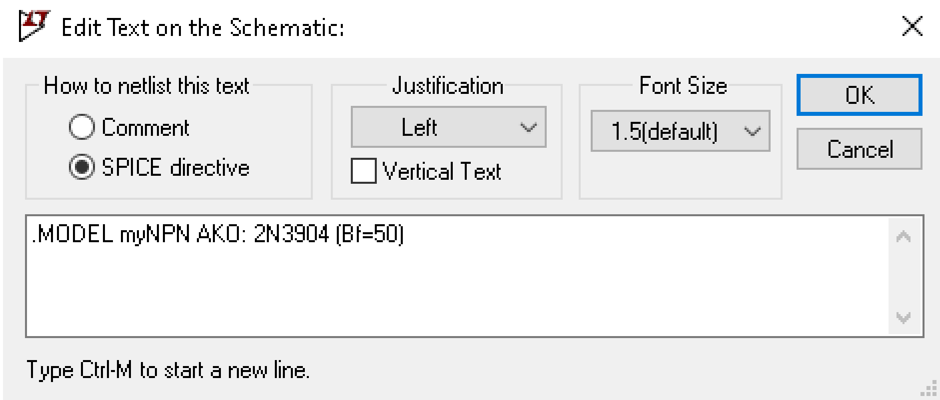


This will bring up the list of all the models the program installs with by default (quite a bit to use just for transistors). Any of the names on the left can be used in the command. The following diagram is the list that is available in LTspice.



Make sure that in the SPICE model column that the transistor has a Beta factor to edit (will be called bf=# or BF=# in that line somewhere).

Then use the following command to “copy” attributes to the transistor in the diagram, (press the “s” key on your keyboard to bring the SPICE command window up).



This allows us to quickly edit any parameter of the 2N3904 (e.g. replace 2N3904 with any other name to test different versions) transistor in the library.

.MODEL myNPN AKO: 2N3904 (Bf=50)

Note:

The syntax AKO (= A Kind Off) which is an alias assignment in SPICE. This shows that the following SPICE directive: .MODEL myNPN AKO: 2N3904 (Bf=50). This means that the myNPN transistor is the same as 2N3904 transistor, except it has a lower Beta value than original 2N3904 transistor.

In this case, we are only interested in the Beta factor as that is a main key factor in a bipolar transistor. The Bf=# can be any number, expect less significant values with a lower number though.

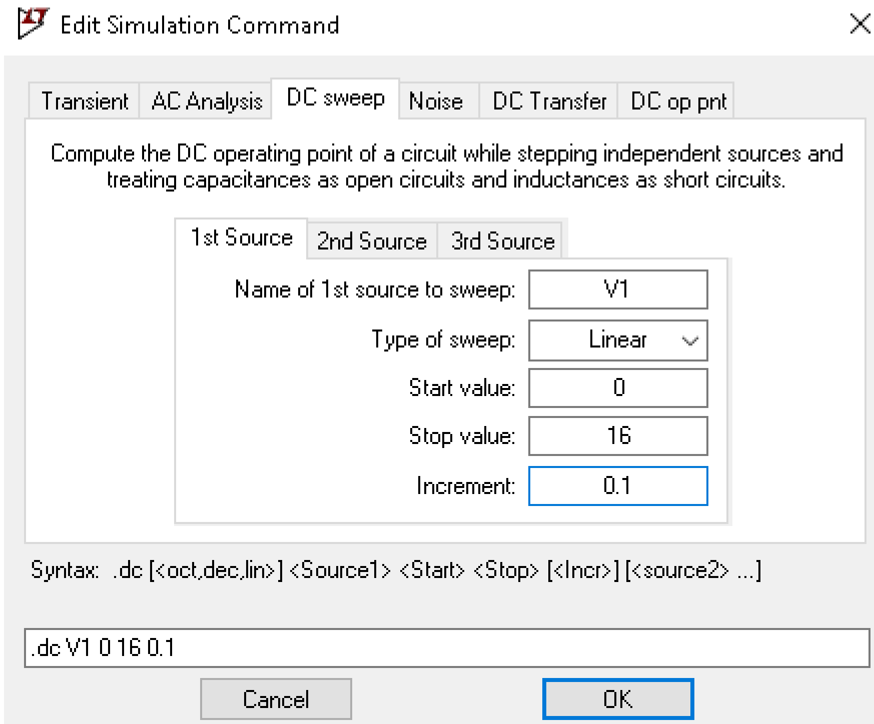
3. Simulation of the Circuit

To obtain the same output characteristic (Ic vs. Vin) on the virtual oscilloscope of LTspice, a stepped/ incremental values of devices simulation must be used i.e. a DC sweep simulation in LTspice.

The DC Sweep will be used to sweep both of our sources: the input voltage V1 and the input current source I1.

The DC voltage is typically swept in a smooth linear fashion while the current source is stepped with significant microampere stages.

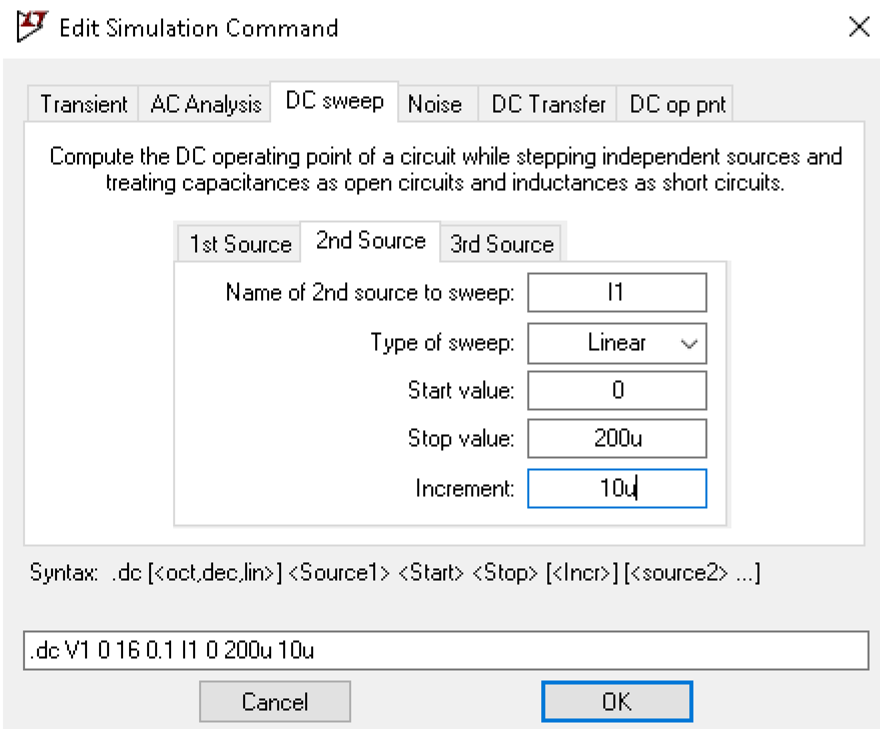
To achieve this, click run and swap to the DC Sweep tab and edit the voltage source V1 in the circuit in the 1st Source tab as seen in the figure below.



The V1 is the voltage source that we sweep.

We are using linear interpolation starting at 0 V and ending at 16 V (can be any number of preference), and the increment is each 0.1 V. This will create a smooth linear input voltage rise.

Then, click the 2nd Source tab to edit the values for the current source I1.



The second source is stepped in significant stages with linear interpolation.

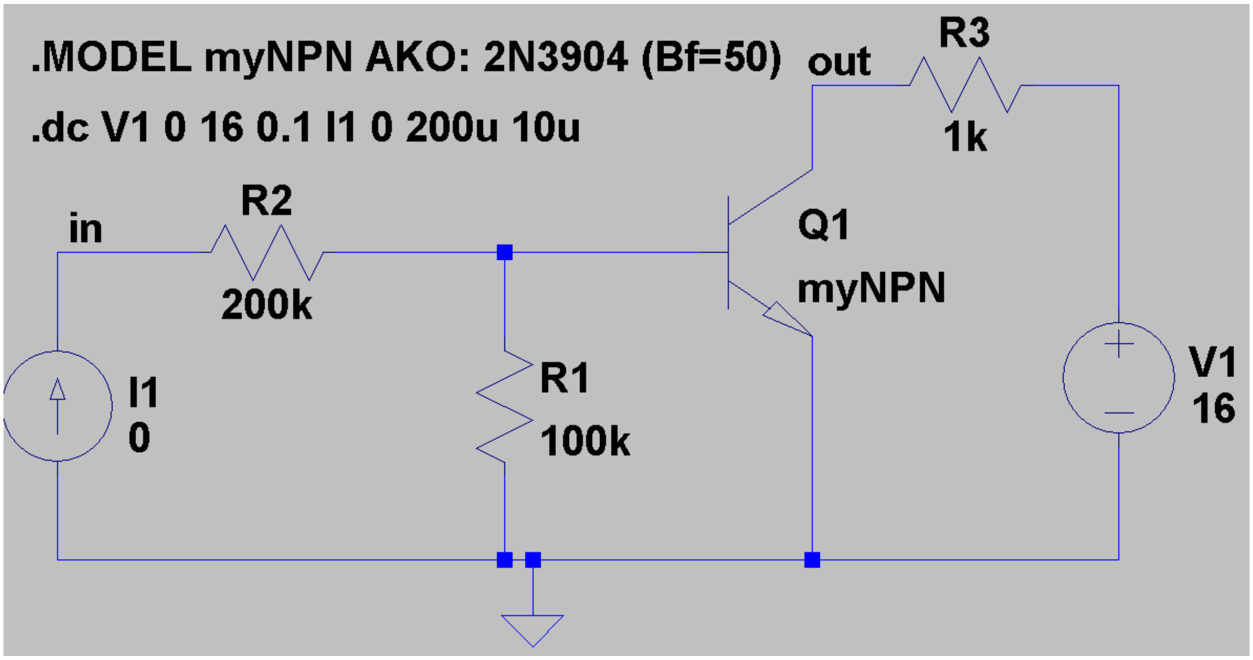
Begin with 0 A and end with somewhere around 200 uA (can be more or less, depends on transistor choice and beta value, that is why experimentation works best with this method).

Use an increment value of 10 uA steps or anything beneath 200 uA.

.dc V1 0 16 0.1 I1 0 200u 10u

Smaller steps will take longer for it to simulate, but will be more accurate. Larger steps will take less time, but will not be a good representation of data.

Click OK and there are a few places to consider measuring for experimentation.



4. Plotting the Graph

Specifically for getting the same graph as mentioned before, probe the current on the leg where the 1kΩ line goes into the transistor (the collector). A special symbol (i.e. a magnifying glass symbol) will show up if one hovers over the leg.

Here is the voltage-to-current characteristics of the BJT transistor that we get with a Beta factor of 50 on a 2N3904 transistor with the default setup.

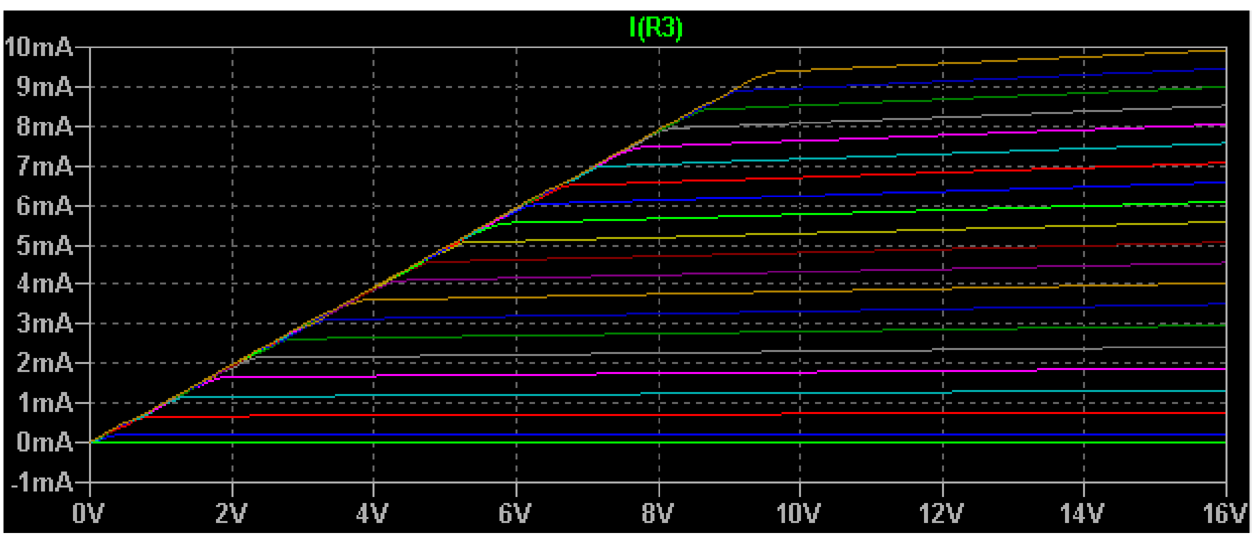


Figure 4: Graph of Vcc vs. Ic of BJT transistor in LTspice

What can be interpreted from this graph? For example, it takes around a 10 V input to obtain a 9 mA boosted signal from an input of 200 uA because the transistor has not saturated yet for lower levels of voltage input (the beta value is affecting this).

Amplification needs to be consistent for most applications, so it would be best to have an output that does not change that much per input voltage change. After 10 V (for the top orange line), the output does gradually increase current, but not significantly.

The measurement from when it levels off around 10 V to the full 16 V is only a 544.51 uA difference while the difference between 0 mA and 9 mA is a steeper change. This is very useful information with certain designs.

One final detail to mention is that Beta and/or Alpha can actually be represented visually because there are three more equations to mention that are key for transistors like as shown below:

$$I_e = I_b + I_c \qquad I_c = \beta * I_b \qquad I_c = \alpha * I_e$$

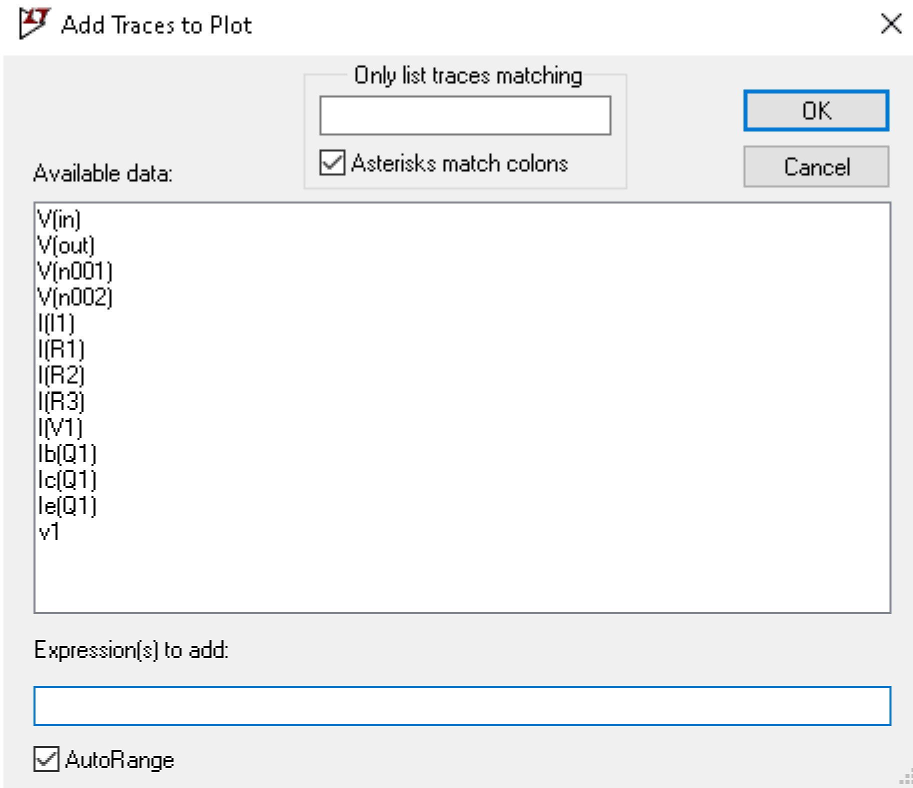
Beta can be solved for:

$$\beta = I_c/I_b$$

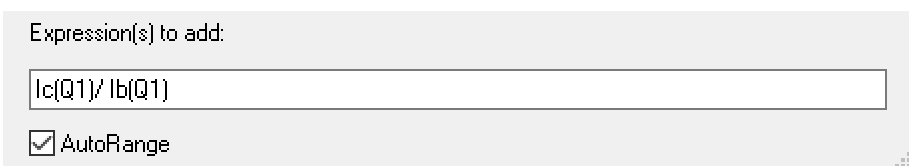
Alpha can be solved for:

$$\alpha = I_c/I_e$$

To accomplish this in LTspice right click on the oscilloscope area and click on Add Traces.



In the Expression(s) to add: textbox, type (or click the data points and add a slash symbol (/) between them) Ic(Q1)/Ib(Q1) to plot Beta:



The plot of the Beta of our modified BJT transistor that will be seen is as shown in the figure below.

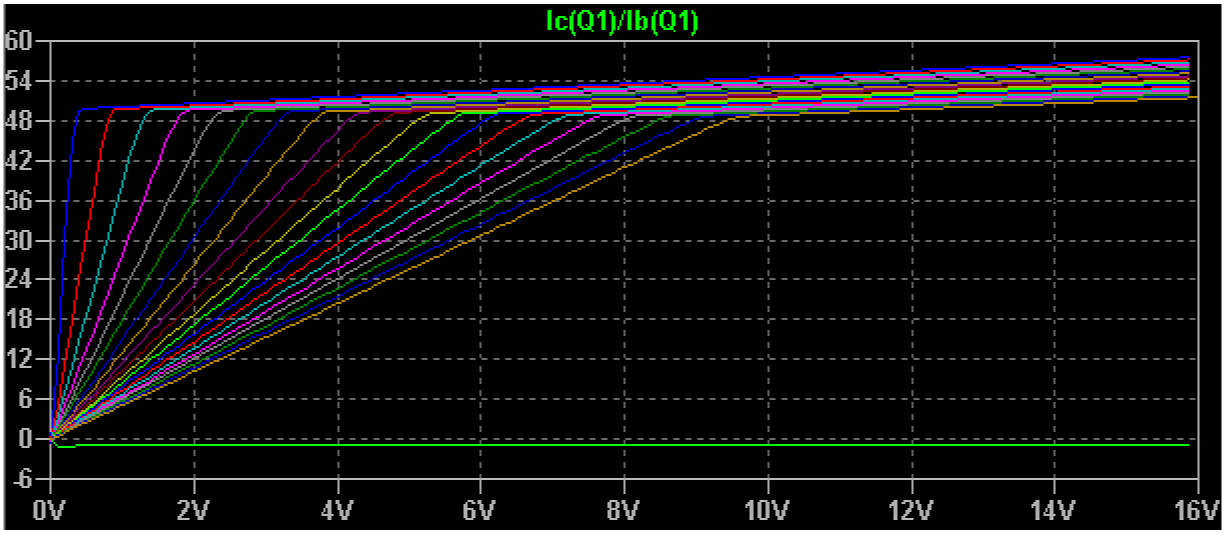


Figure 5: Graph of Beta of BJT transistor

Notice how they all float around 50 after the transistor stabilizes. This result is expected because Beta is typically held to be a constant value, so the graph is reasonable for the changing input current versus changing input voltage.

To plot Alpha of the BJT transistor, we will do the same process as before, but use Ic(Q1)/-Ie(Q1) [negative Ie as Ie is the opposite of Ic to get positive values].

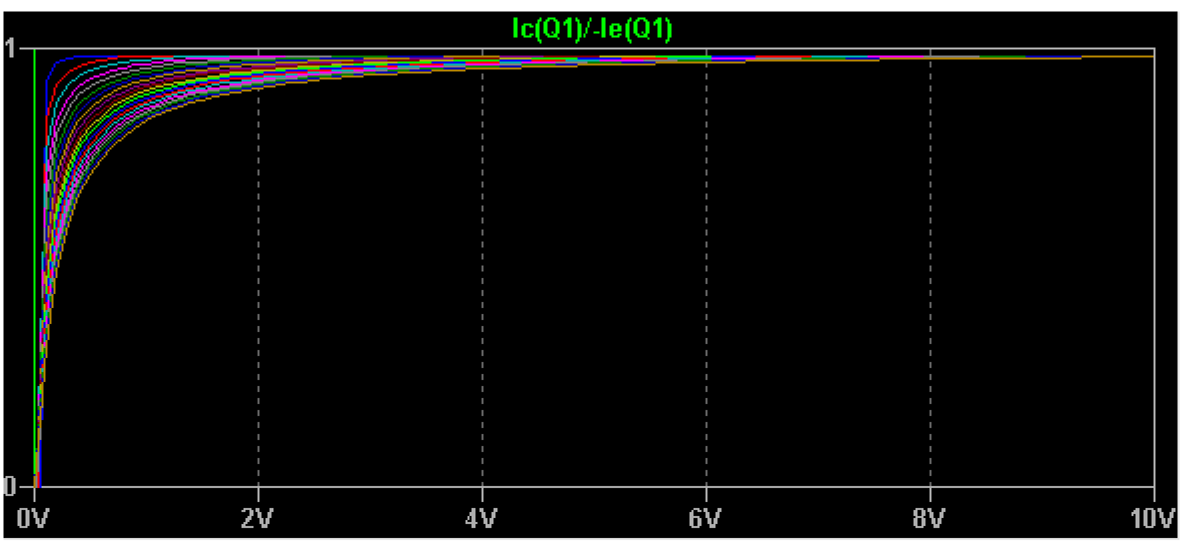


Figure 6: Graph of Alpha of BJT transistor

This result is also expected because when Alpha is calculated from Beta its value should be around 0.99.