

Lab 1: The Bipolar Junction Transistor (BJT): DC Characterization

Electronics II

Contents

Introduction	2
Day 1: BJT DC Characterization	2
Background	2
BJT Operation Regions	2
(i) Saturation Region	4
(ii) Active Region	4
(iii) Cutoff Region	5
Part 1: Diode-Like Behavior of BJT Junctions, and BJT Type	6
Experiment	6
Creating Your Own File	6
Report	8
Part 2: BJT I_C vs. V_{CE} Characteristic Curves - Point by Point Plotting	8
Prelab	8
Experiment	8
Creating Your Own File	8
Parameter Sweep	9
Report	10
Part 3: The Current Mirror	11
Experiment	12
Simulation	12
Hardware	12
Checkout	14
Report	14

Introduction

When designing a circuit, it is important to know the properties of the devices that you will be using. This lab will look at obtaining important device parameters from a BJT. Although many of these can be obtained from the data sheet, data sheets may not always include the information we want. Even if they do, it is also useful to perform our own tests and compare the results. This process is called device characterization. In addition, the tests you will be performing will help you get some experience working with your tools so you don't waste time fumbling around with them in future labs.

In Day 1, you will be looking at the DC characteristics of your transistor. This will give you an idea of what the I-V curves look like, and how you would measure them. You will also have to build and test a current mirror, which should give you an idea of how they work and where their limitations are. In Day 2, you will look at the AC characteristics. You will learn how to measure medium and higher frequency measurements, and how to calculate useful transistor characteristics from them.

Day 1: BJT DC Characterization

Background

The BJT is a three-terminal semiconductor device containing two PN junctions. If checked with a digital multi-meter it appears to be two diodes of opposite polarity connected in series. However, unlike two series diodes, the BJT can be used to amplify. This is because the base is small enough to allow the two sets of PN junctions to interact with each other. There are two basic types of BJT, as illustrated in Figure 1.

BJT Operation Regions

Figure 2 shows an example of an I-V curve of an NPN BJT transistor. As its name would imply, an I-V curve plots the device current of the device against a sweep of terminal voltage. Figure 2 shows an example of collector current as the collector-emitter voltage is changed, with a separate curve for different base currents. This family of curves is organized by base current since the diode-like characteristics of the current vs. base-emitter voltage would make the voltages of each curve very similar (and highly susceptible to process variations). Figure 3 shows the opposite type of curve, with a plot of collector current as the base-emitter voltage is swept. The collector-emitter voltage is not changed as this does not affect the family of curves significantly. This curve does not give as much insight as the first curve, but is shown as a comparison (more information could be obtained if I_{CE} was plotted on a log scale instead of a linear scale, but is still not as useful).

BJTs, like all transistors, are non-linear. The non-linear properties are what make it useful as an amplifier or switch, but as you will find in higher level courses, non-linear devices are a pain to analyze. Fortunately, we can treat a BJT as a linear device as long as its current/voltage characteristics remain in one of a few confined regions. A fourth region exists (reverse active) but is not used in practice due to its poor properties. The very same semiconductor conditions that give high gain in the normal active region are the same ones that cause low gain in the reverse active region.

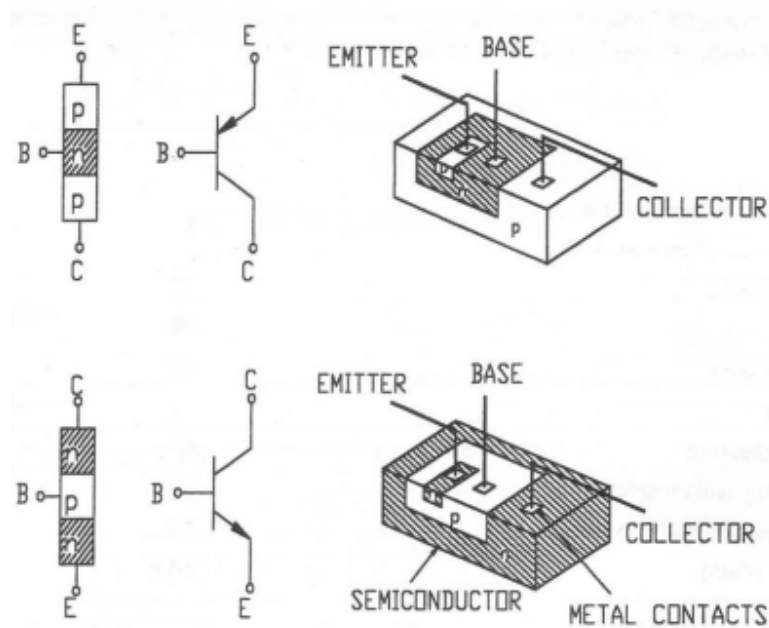


Figure 1: BJT (PNP top, NPN bottom; basic structure LEFT, symbol MID, practical structure RIGHT)

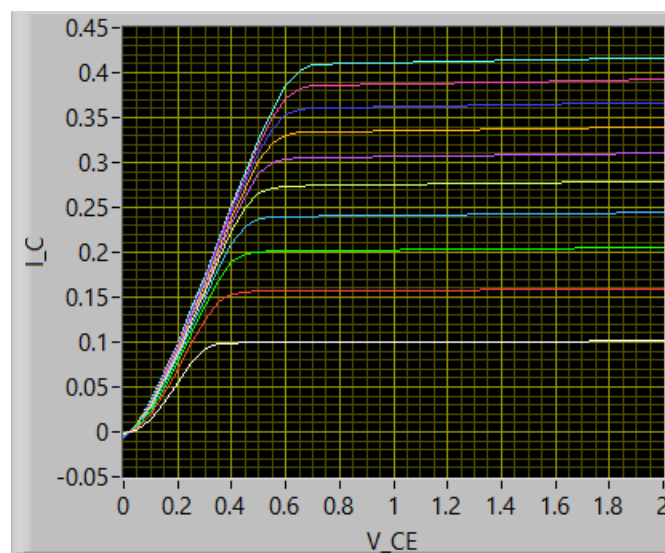


Figure 2: Plot of the I-V curves of a 2N3904 transistor sweeping V_{CE} (in Volts) for different I_B (in Amps)

To run the simulation of Figure 2 yourself, simply go to the following URL, and click **BjtAnalyzer_NPN_figure2.ms14**. Select **Open With: Multisim 14.2**, then hit **OK**. Once open, select **Simulate** → **Run**, then double click the **XLV1** component in the workspace.

<http://www.doe.carleton.ca/~nagui/Elec3509/Lab1/>

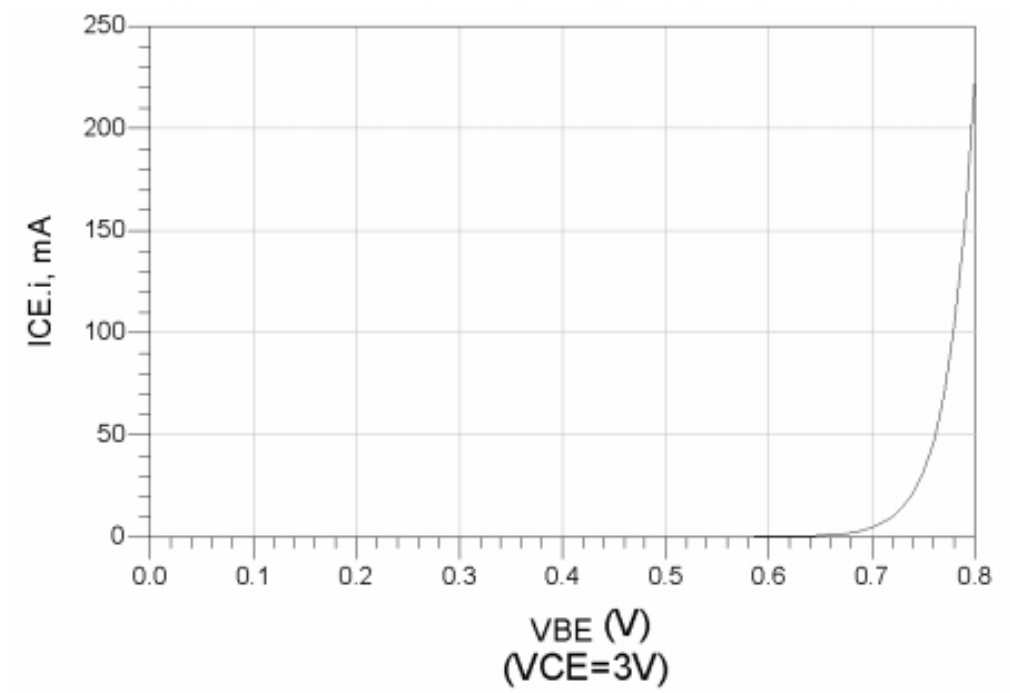


Figure 3: Plot of an I-V curve of an example transistor sweeping V_{BE} with a fixed V_{CE}

(i) Saturation Region

In this region, both BJT junctions are forward biased. V_{CE} is small, e.g. 50-100 mV, but quite large collector and base currents (I_C & I_B) can flow. This region is not used for amplification. There is a low resistance between the C and E terminals; the BJT acts like a closed switch. Figure 4 shows an actual circuit of a BJT in saturation and the small-signal equivalent (that is, the linear model) of the circuit.

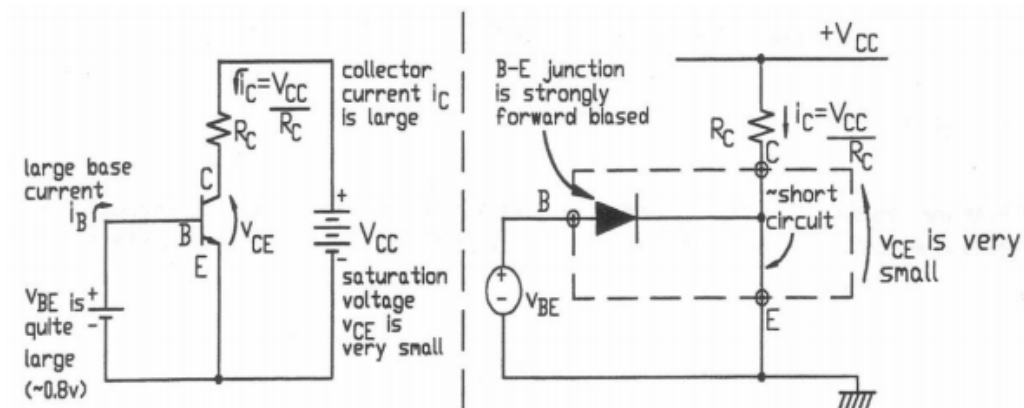


Figure 4: Saturation Region, Both B-E and B-C Diodes are Forward Biased

(ii) Active Region

Here the B-E junction is forward biased but the B-C junction is reverse biased. Because the two junctions are very close together, the emitter "emits" carriers which shoot across the central base region and are "collected" by the collector region. This flow of carriers manifests itself externally as a relatively large collector current I_C . This process is strongly influenced by the external injection of a much smaller current I_B into the base region.

This lets the collector current be controlled almost completely by the base-emitter junction voltage, and is nearly independent on the collector node voltage (a very useful result). As the collector/emitter current ratio is dependent on fixed semi-conductor parameters, we can define the BJT current gain using Equation 0.0.1.

$$\beta = \frac{I_C}{I_B} \quad (0.0.1)$$

Figure 5 shows an actual BJT operating in the active region and the small signal equivalent model. Do not confuse this with a MOSFET in saturation, which behaves similarly to the BJT in the active region.

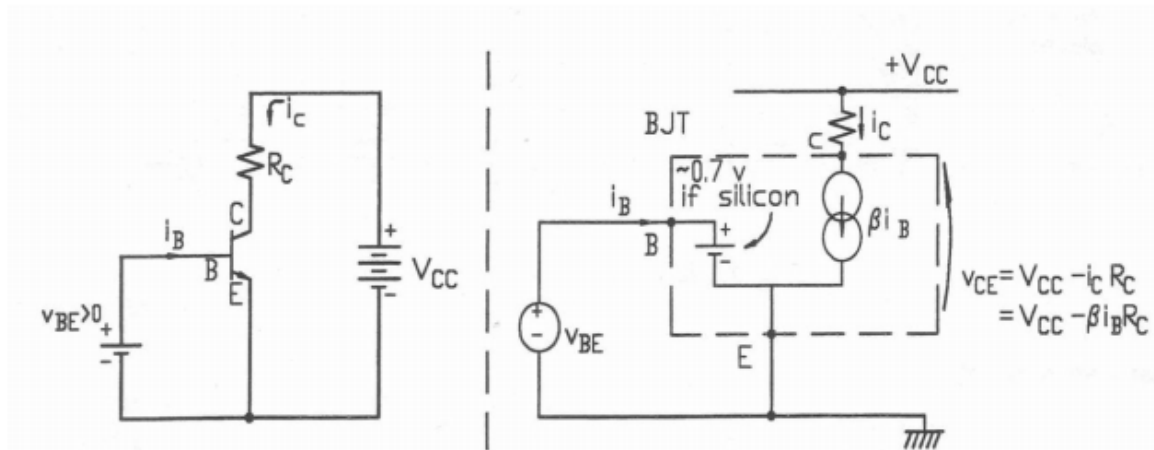


Figure 5: Active Region, B-E Diode is Forward Biased and B-C Diode is Reverse Biased

(iii) Cutoff Region

If both the junctions are reverse-biased, only very small reverse leakage currents can flow across them. No gain is available in this mode, and there is a high resistance between the C and E terminals.

The small-signal circuit model is not shown; it is just an open circuit between all 3 nodes.

Part 1: Diode-Like Behavior of BJT Junctions, and BJT Type

Using a Digital MultiMeter (DMM) on the “diode” setting, one can measure the forward and reverse voltages of the B-E, B-C and C-E junctions of a 2N3904 transistor, shown in Figure 6. The “diode” setting actually forces an output current of 1 mA from its positive terminal, and then measures the voltage developed between the positive and common terminals. Thus, the DMM can directly measure the forward voltage drop of PN junction under 1 mA of bias. Note the orientation of the device as seen in Figure 6. It is a very common mistake to switch the emitter and collector terminals. Another common mistake is to confuse a PNP transistor for an NPN transistor or vice-versa.



Figure 6: Lead Configuration of a 2N3904 Transistor. Lead configuration for the 2N3906 is the same.

There should be diode-like behavior between the B-E and B-C terminals, but not between the C-E terminals in either direction. Any other behavior usually indicates a damaged BJT. Of course, it is possible for a BJT to be damaged but still pass this test.

Experiment

By measuring the polarity of the voltage appearing across the multimeter leads, determine whether the 2N3904 is a PNP or NPN type. Do the same for the 2N3906 transistor. Remember to record which meter lead is connected to which transistor terminal.

To simulate your circuit and measure the voltage drop you will need to log in to the JPKnight server, found at the IP address 134.117.38.243. Instructions on how to do this can be found in the Lab Remote Connection Reference Document.

The circuit has already been built for your convenience. Should you want to construct the circuit yourself to gain a better understanding of the software (suggested), instructions on how to do so are provided at the end of Part 1. The files needed for the lab can be found at the URL below.

<http://www.doe.carleton.ca/~nagui/Elec3509/Lab1/>

Click **Diode_like_behavior_figure6.ms14** and select **Open With: Ni Multisim 14.2**. Select **OK**. In this file you will see the Agilent multimeter along with an NPN and PNP BJT. Instructions in the file describe how to simulate the circuit to gain the DC biases for the two transistors.

Creating Your Own File

You can open Multisim from the desktop using the application as seen from Figure 7. Once the application has opened, you will be greeted with a blank workspace. In this workspace you can add components to build your desired circuit and add measurement probes for testing and simulation.

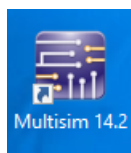


Figure 7: Multisim Icon

To place the 2N3904 and 2N3906 BJTs on the workspace, select **Place** → **Component**. A window will appear as shown in Figure 8. Components are sorted into Groups and subsequent Families. The groups and families can be changed along the left side of the window. Change the Group from **<All groups>** → **Transistors**. Leave the family as **All Families**. Search the list to find each transistor and place them on the workspace by selecting **OK** and clicking on the desired location on the workspace.

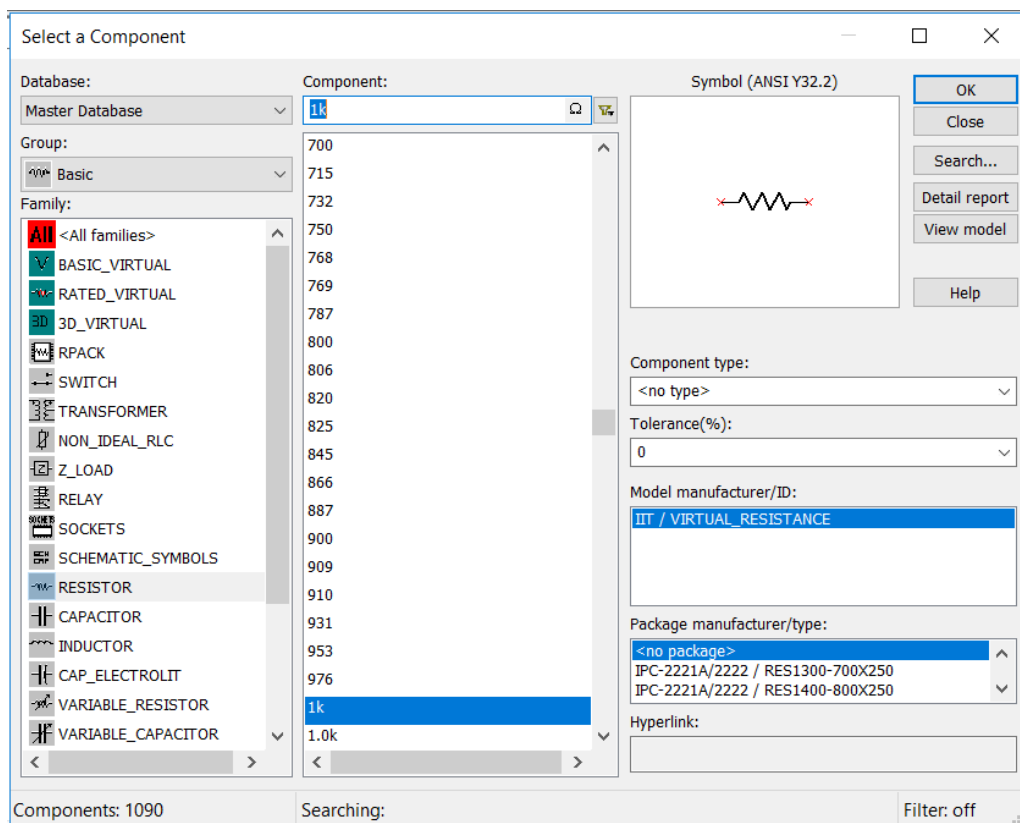


Figure 8: Tool Menu

Once you have your two transistors in the design window, you must add the Agilent multimeter, which can be found in the menu **Simulate** → **Instruments** → **Agilent multimeter**. Double click on the Agilent multimeter to see the pop up window like that in Figure 9. Begin by turning on the multimeter by pushing the **Power** button on the left. To put the multimeter into Diode mode, first hit the **Shift** button (dark grey on right side), then the **Cont** button (up one, left 2 from shift button, under the Function wing). A diode symbol will appear in the bottom right corner of the Agilent display window when this is done correctly. Finally, you must add wires to connect to the pins of the BJT. The top right red port is the positive terminal, and the right middle black port is the common terminal.

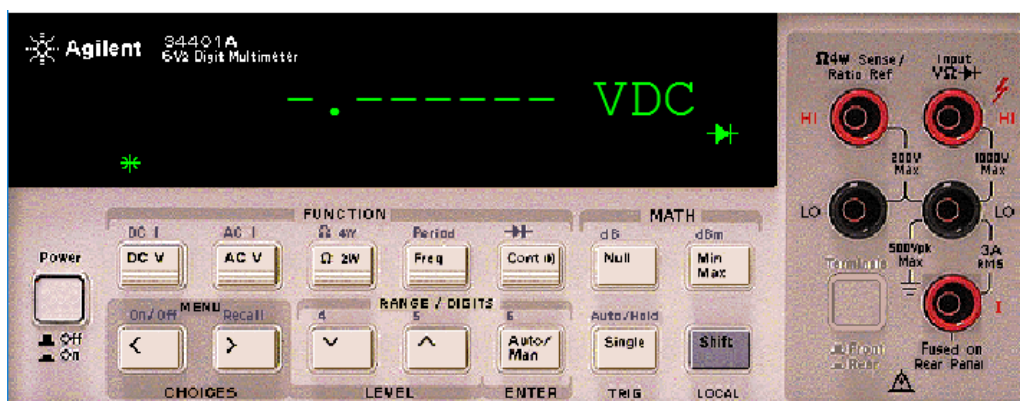


Figure 9: Agilent Multimeter Face

To connect wires, hover over the end of the desired component pin. Your mouse will change to a black dot with crosshairs. Click on the pin, then click on the desired location pin.

Finally, select **Simulate** → **Run**, and double click on the Agilent multimeter to see the measured result. You must measure the forward and reverse voltage biases for the B-E, C-E, and B-C junctions of both BJTs.

Note that interactive simulations do not treat floating voltages well. Make sure to include pull down resistors in your circuit if you would like to incorporate switches in the simulation.

Report

Show your resulting bias voltages for the NPN and PNP BJT in a table and explain your conclusions. Which region is the transistor in for each bias?

Part 2: BJT I_C vs. V_{CE} Characteristic Curves - Point by Point Plotting

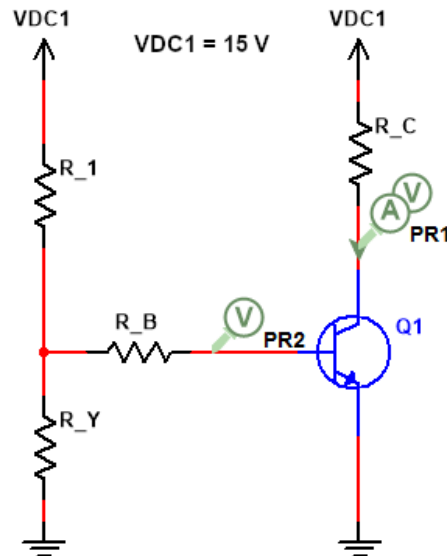


Figure 10: Test Circuit for Part 2

For this section, you will generate a plot similar to Figure 2 by plotting I_C vs. V_{CE} . The figure shows several curves for different I_B values. You will produce one curve at a single constant I_B value that gives I_C in the active region around 2 mA.

Prelab

1. Determine appropriate values of R_1 , R_B and R_C for Figure 10 (use decade multiples of 1.0, 1.2, 1.5, 1.8, 2.2, 2.7, 3.3, 3.9, 4.7, 5.6, 6.8, and 8.2 k Ω . As an example, if your R_1 is 124.6 k Ω , you will round this to 120 k Ω or 150 k Ω). You will need to justify the choices for your resistor values in your report with good reasons.

Obviously there will be many possible values of R_C , with each representing a different V_{CE} . Choose at least 5 values in the active region and another 5 or more values in the saturation region. Show the expected V_{CE} for each resistance.

As a hint for these calculations, keep in mind that V_{BE} is around 0.7 V in the active region, V_{CE} sat is 0.5 V, beta is around 150 and that the voltage drop across any resistor should be at least 0.25 V to be able to measure it accurately. In addition, the current flowing in the left branch should be much greater than the base current. R_Y is 500 Ohms.

Experiment

The circuit is already assembled for you on [Nagui's website](#), you should follow the link, open up part2.circuit.figure10.ms14, and run a Parameter Sweep for R.C. Instructions on how to set up and perform the Parameter Sweep can be found in the Parameter Sweep section below.

Creating Your Own File

Multisim has a large library of components available for use, but for the purposes of lab 1 only a few are needed. The components you will need can be found in {Group: Family: Component}: Sources: POWER_SOURCES: DC_POWER, Sources: POWER_SOURCES: Ground, and Basic: RESISTOR: {Value}.

Once on the workspace, components can be dragged to new locations, and double-clicked to edit their values and names. Adjust the DC power supply to 15 V.

With the circuit is built to match Figure 10 using your calculated resistor values add a voltage and current probe to the collector of the BJT to measure I_C and V_{CE} . Add an additional voltage probe to the base to measure V_{BE} . Probes can be placed by selecting **Place** → **Probe** and selecting the appropriate one.

Parameter Sweep

In order to take measurements the circuit must first be simulated. To make things easier, a simulation of many resistances for a range of R_C values can be done by performing a Parameter Sweep. To set up the simulation, go to **Simulate** → **Analysis and Simulation**. A new window will appear like Figure 11.

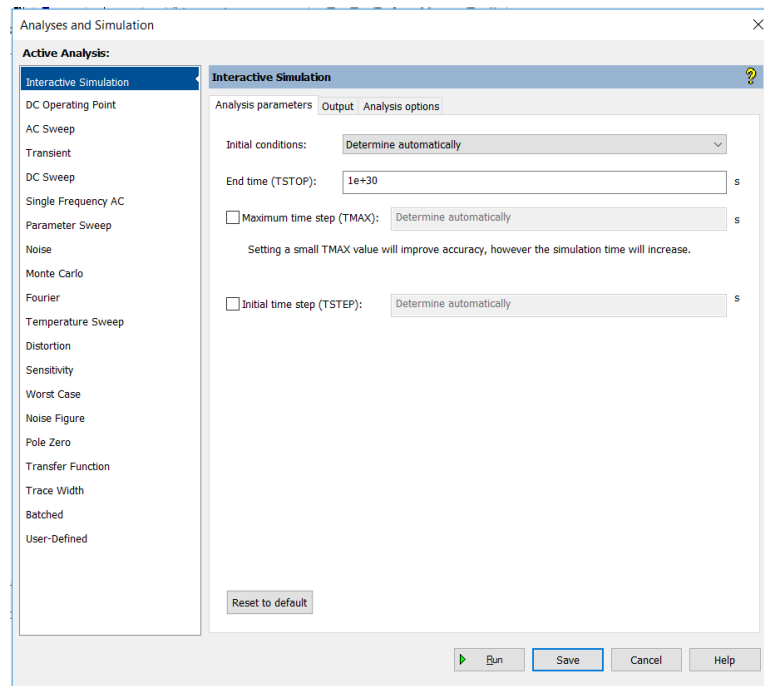


Figure 11: Simulation Settings

Change the Active Analysis from Interactive Simulation to Parameter Sweep on the left-hand side of the window in Figure 12. In Sweep Parameters change Device Type to Resistor, the Name to your R_C label, and the Parameter to Resistance.

Next, in Points to Sweep change the Sweep Variation Type to Decade, and Number of Points Per Decade to 20. Lastly, change the Start and Stop values to your maximum and minimum calculated R_C values from your prelab respectively.

Lastly, in More Options change Analysis to Sweep from Transient to DC Operating Point. Hit Save at the bottom of the window, then select Simulate → Run.

A window will appear showing a graph of the measured voltages and current. This graph should look something like Figure 12.

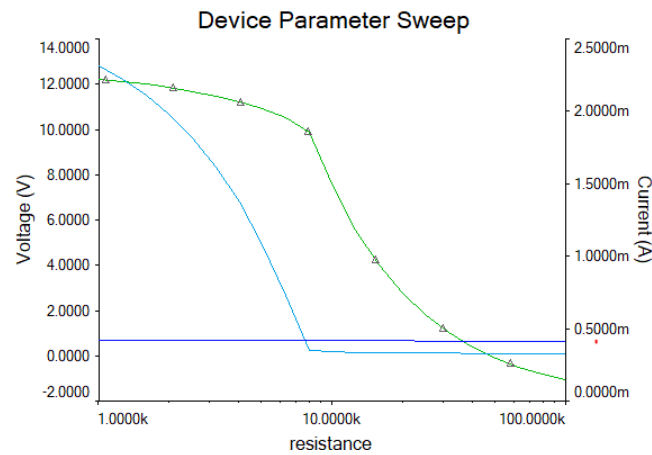


Figure 12: Example Sweep

To obtain the data for report graphing and calculations, on the Grapher View window go to **Tools** → **Export to Excel**. Select all traces to be exported, and select **OK**. A new Excel spreadsheet will automatically be generated and open. Save this file to your H: Drive for later use.

Report

1. Plot the I_C - V_{CE} line for the base current that was used. Determine the I_C/I_B current ratio for the transistor at each point.
2. Plot V_{BE} vs. V_{CE} . Does V_{BE} change very much as V_{CE} is changed? Why? What happens to this current ratio when the transistor goes into saturation?
3. From your plot, determine the value of beta and a value for the Early voltage V_A .
4. Define each of the regions the from the I_C - V_{CE} for your graph and for figure 2 of the introduction.

Part 3: The Current Mirror

In network analysis, you have encountered two types of sources: fixed voltage and fixed current sources. With a voltage source, the voltage across the terminals is always a fixed value, while with a current source, the current flowing through the source is always a fixed value. While it is fairly easy to implement a fixed voltage source (e.g. a voltaic cell), implementing a current source is more difficult.

Your notes show examples of a circuit called a current mirror which is the easiest implementation of a current source. A current mirror typically has two or more branches. In one branch (known as the reference branch) a fixed, known current flows, which can be controlled by adjusting a potentiometer, some controllable switches or by a self-correcting circuit such as a band-gap generator. The other branch has a current flowing through them which is a direct multiple of the reference branch: ratios of 1:1 to 1:4 are common, and higher ratios are possible if less accuracy is needed.

An NPN current mirror with emitter degeneration resistors is shown in Figure 13. The emitter resistors, R_{e1} and R_{e2} , reduce the effect that mismatches in V_{BE} between the two discrete transistors have on the matching of the output current to the reference current. These resistors can also be conveniently used to alter the ratio between the two currents. Different permutations of this circuit exist which allow the mirror to operate at a wider output voltage range, or increase output impedance.

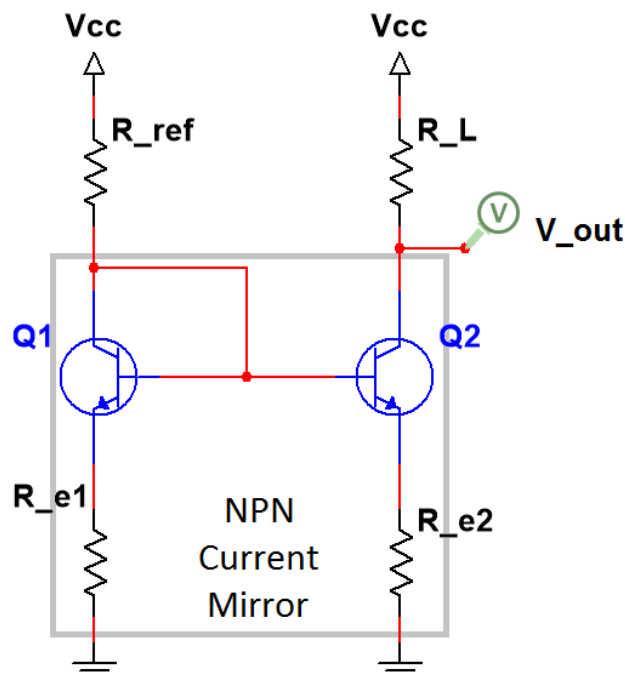


Figure 13: The Current Mirror and Test Circuit

Prelab

- Given the circuit in Figure 13, derive an equation for the output current (I_{out}) in terms of I_{REF} , V_{BE1} , V_{BE2} , β_1 , β_2 , R_{E1} , R_{E2} and V_{CC} .
 - You may assume $V_{BE1}=V_{BE2}$ and that β s are both very large. V_{CC} is known to be 15 V.
- With R_{E1} set to 2 k Ω , pick a value of R_{E2} so that the current is mirrored (1:1 ratio). Choose an appropriate value of R_{Ref} (remember that the base is shorted to the collector) so that the reference current is approximately 0.5 mA.
- Determine the maximum value of R_L for which the current mirror will still be able to deliver the necessary amount of current. Keep in mind that the behavior of the transistor changes if it enters a different region of operation. Choose 10 values for R_L , 5 that show the range of load resistances over which the current mirror operates and 5 that show where the current mirror fails. Show the expected current and output voltage for each load resistance.
- Design a **1:2 current multiplier** where the output current is two times the reference current.

5. Design a PNP current mirror such that it can be connected to the NPN current mirror as can be below. Note that the PNP current mirror may have other connections not seen in the figure (such as a connection to power). The PNP current mirror should mirror its reference current in a 1:1 ratio.

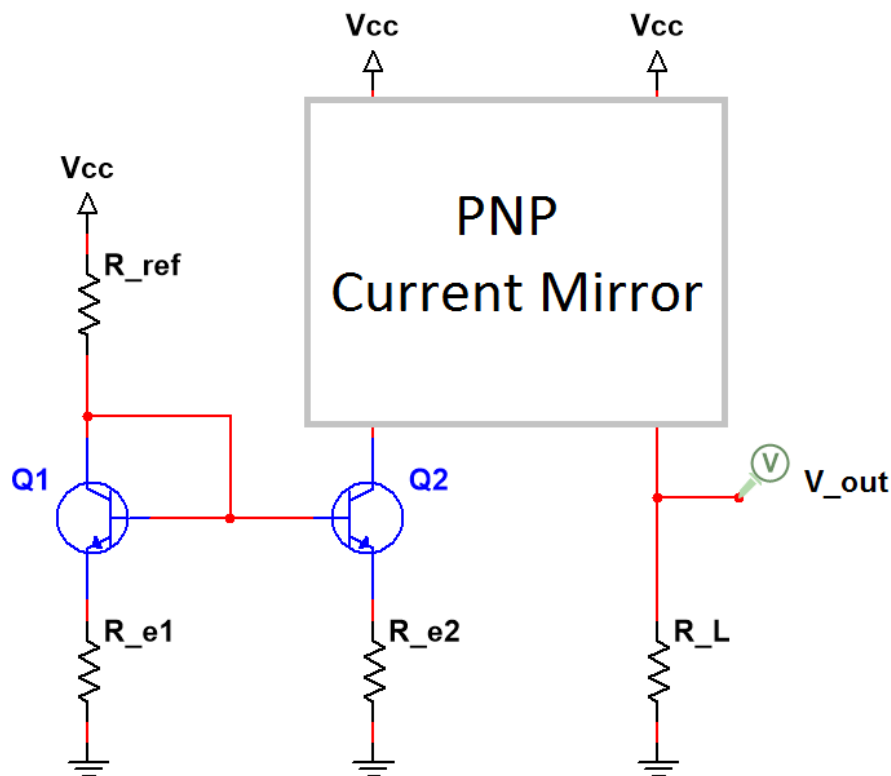


Figure 14: Schematic of the NPN Current Mirror Connected to the PNP Current Mirror with Test Circuit

Experiment

Simulation

1. The circuit is already assembled for you on Nagui's website, you should follow the link, open up npn_mirror.ms14, and run a Parameter Sweep for R.L.

Optionally, you may construct the NPN current mirror as shown in Figure 13. Test the circuit with a parameter sweep. The maximum and minimum loads must be the maximum and minimum loads picked out in the prelab. Measure the output voltage (V_{out}), and calculate the current flowing through the load resistor.

2. Modify the circuit to create a 1:2 current multiplier and measure the current flowing through different loads using a parameter sweep to show that the current mirror is in fact still acting as a current source with the correct output current.
3. Change your original NPN mirror back to a 1:1 ratio and connect the PNP current mirror that you designed as shown in Figure 14. Note that the load resistance of the NPN current mirror is replaced by the reference branch of the PNP current mirror. Measure the current flowing through different loads (parameter sweep) to show that the current mirror is in fact still acting as a current source with the correct output current.
4. Check your work to see that you have completed the simulation part of the lab before moving on.

Hardware

1. The current mirror of Figure 13 has already been constructed in the lab and attached to a 15 volt source and a Rohde & Schwarz Digital Oscilloscope. You must log into your assigned lab computer to access this lab equipment. Instructions on how to log into the lab computer and control lab equipment can be

found in the Lab Remote Connection Reference document.

Use the camera application found on the VLSI lab computer desktop to view the live circuits. Map out how the breadboard is designed, and check to make sure the leads are correctly connected.

The R_L used is a digital potentiometer. To control the value of this resistor, open the Carleton University Arduino GUI application found on the desktop, see Figure 15. This digital potentiometer has a variable range from about 0 k Ω to almost 100 k Ω . It can be assumed that the increments are linear for the potentiometer. You can use your best judgement to determine which resistor (A or B) is the load resistor.

You must use the oscilloscope accessed through your favorite browser (see Lab Remote Connection Document: Step 4), where you can open something like Chrome and search scopeXX.vlsi, where XX is your VLSI computer number. Channel 1 is the voltage on the load, V_{OUT} , and Channel 2 is the voltage on the bases V_B . Measure both voltages while the load resistor changes through the digiPOT application.

Compare the values found through your simulation with those of the physical circuit.

2. See what you have completed, make sure you've recorded the values of the load resistors and the voltages.

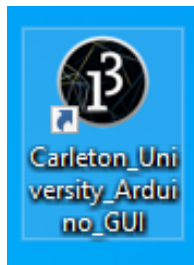


Figure 15: DigiPOT Desktop App Shortcut



Figure 16: Application for Load Resistor Control

Checkout

When you have completed both the hardware and software labs, make sure to contact the TAs and show them the work you've done so that you can receive checkout marks.

Report

- Show your derivations for your equation for the current mirror's output current and the 1:2 current multiplier.
- With your measurements from your 1:1 mirror, plot output current vs. load resistance and output voltage vs output current. From the second plot, determine the output impedance. Keep in mind that a current mirror can be modeled as an ideal current source in parallel with a large resistance (which is its output resistance). Do this for both the simulation and the hardware circuits, and compare.
- Make another plot showing the output current vs. load resistance for your multiplier.
- Comment on the first transistor (the one with the base and collector shorted), what two-terminal device does it resemble the closest? With the transistor configured in this way, can it enter the saturation region of operation? If so, under what conditions? If not, why not?
- Consider the range of loads over which the 1:2 mirror can deliver its rated current. How would you expect this range to change with the multiplier and why?