

# ECE 205 – BJT LTSpice Simulation

## Challenge: BJT LTSpice Simulation

In ECE 205, we will use the BJT large signal model for all our transistor calculations. This model is generally sufficient for the purposes of current amplification but is inaccurate in more sophisticated circuits.

Challenge: Evaluate some BJT circuits using the circuit analysis techniques you learned in ECE 205 lecture. Simulate these circuits using LTSpice and compare the results to your manual calculations to gain an understanding of the accuracy and limitations of our ECE 205 BJT circuit model.

Prelab Deliverables: (on Compass)

- In class, I drew a BJT I-V characteristic on the right. **By hand, draw an equivalent I-V characteristic, but using the model that we learned in class.** Use the parameters  $\beta = 100$ ,  $V_{BEon} = 0.7\text{ V}$ , and  $V_{CESAT} = 0.2\text{ V}$  for the transistor. Graph  $V_{CE}$  on the abscissa from 0-10 V,  $I_c$  on the ordinate from 0-4 mA. Draw at least 5 curves corresponding to  $I_b = 0\mu\text{A}, 10\mu\text{A}, 20\mu\text{A}, 30\mu\text{A}, 40\mu\text{A}$ . Hint: how is our transistor model simplified from the I-V characteristic I drew in class? For example, the model we discussed in class has a **constant value** for the saturation voltage – where would that show up in the graph on the right?
- The circuit to the right shows a *common emitter amplifier*, which is the simplest form of voltage amplifier which can be made with a single transistor. By using hand calculations, draw the equivalent **voltage transfer graph** (which is a plot of  $V_{in}$  on the abscissa “x-axis” vs  $V_{out}$  on the ordinate “y-axis”). Use at least following values of  $V_{in}$ : (0V, .25V, .5V, .75V, 1V). For each these values, compute and plot  $V_{out}$ .

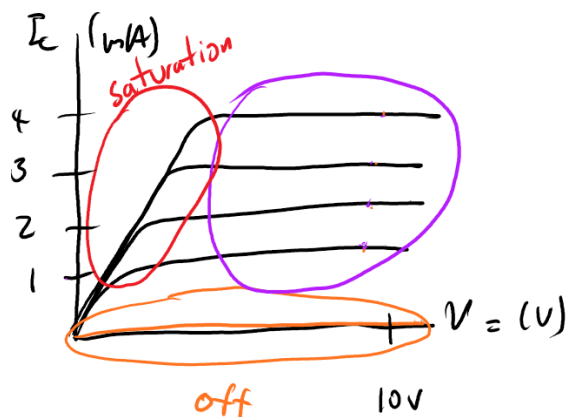


Figure 1: I-V Characteristic

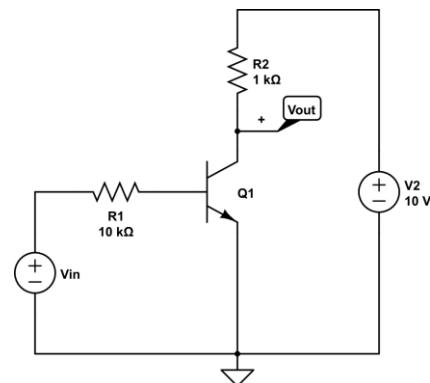


Figure 2: Common Emitter Amplifier

After doing the prelab, you should have two graphs (the overall transistor I-V characteristic graph and the transfer graph for the common emitter amplifier). Your challenge is to replicate both graphics in LTSpice and verify the accuracy of said graphs compared to the ones you computed by hand.

Challenge: Use LTSpice to generate the two plots you drew by hand for your pre-lab. For the transistor, you can use a 2N3904 (add the NPN transistor part, and then right click and click on “pick new transistor” and pick 2N3904). You may want to use AC (that is, time varying) sources to generate the

graph, for example, to create an AC voltage source which goes from 0 to 10V, place a voltage source and then right click to change it to SINE, DC offset = 5V, Amplitude = 10V, Freq = 1 Hz. Then you may do a time varying simulation to graph the results of that voltage changing from 0 to 10V. You will likely want to do this to generate both graphs. Hint: combine this with the .step directive you already learned to generate the transistor I-V characteristic graph.

Report Deliverables:

- Hand drawn transistor I-V characteristic. Make sure you point out the way(s) in which your graph for the transistor I-V characteristic differs from the actual one we drew in class.
  - Hand drawn common-emitter amplifier voltage transfer graph. Estimate the “voltage gain” which is the general relationship between  $V_{in}$  and  $V_{out}$  (note that this is the slope of the voltage transfer graph). Note that this number will be very inaccurate.
  - Simulated transistor I-V characteristic graph. Label the 3 major operating regions (off/forward active/saturation). Include a procedure which detailed how you obtained this graph as a screenshot of any LTSpice circuits you created.
  - Simulated common-emitter amplifier voltage transfer graph, annotate an estimated “voltage gain” and compare it to your by-hand version. Include a procedure which detailed how you obtained this graph as a screenshot of any LTSpice circuits you created.
- 
-