A. Rasmussen/A. Stolp
9/2014

Lab #3 { pts} BJT Transistors

OBJECTIVES:

- Become familiar with the operation of the BJT transistor.
- Testing the BJT transistor in a circuit.
- Investigating the use of the BJT for impedance matching.
- Building and testing an amplifier.

BACKGROUND INFORMATION: (5 PTS)

1. Determine a Parts List for this lab.

BACKGROUND INFORMATION:

Bipolar junction (BJT) transistor internals are explained, testing a BJT transistor, measuring input resistance, measuring output resistance, and circuit noise issues are discussed.

Bipolar Junction (BJT) Transistor Internals:

At its heart a bipolar junction (BJT) transistor consists of two pn junctions (see **Fig. 1**) which can each individually act as diodes. These diodes can be tested just like any other diode. In particular, they can be tested with most multimeters. If both diodes test OK, and you measure no conductivity between the collector and emitter, then the transistor is almost always OK as well. This is a quick and dirty way to test a transistor and a good way to determine some important information about an unknown bipolar junction transistor.

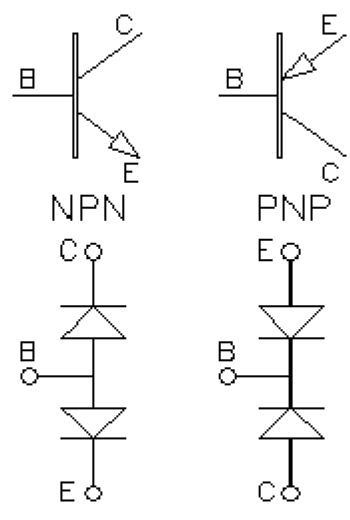


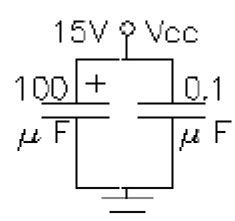
Fig. 1

Meter Diode Setting:

Recall from an earlier lab that most multimeters do not use enough voltage in the regular ohmmeter setting to forward bias a diode, so they give you a special setting to test diodes. If you don't use the special setting then the meter may show little or no conduction for either diode direction. Look for a diode symbol on your meter and set the meter to that position (it's a blue shift setting on the HP meter).

Circuit Noise Problems:

Circuit noise can manifest itself in a variety of irritating ways. Sometimes it's just fuzz on the scope that does little more than mess up the scope's peak-to-peak voltage measurements, forcing you to take them manually. Sometimes noise problems can be so severe that even DC voltmeter readings become weird. Just connecting the meter's leads may cause the noise (check with the scope). Usually capacitors are the solution to noise problems.

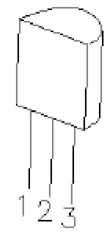


In **Lab #2** (the Op Amp Lab), you placed some filter capacitors across the power supply right on the breadboard. That's a good idea for all the circuits that you build, especially those with high gain factors. A 100 µF electrolytic or tantalum in parallel with a 0.1 or 1 µ F low-inductance ceramic disk is a good start. Shorten or eliminate all the leads that you can. This may mean the removal of measuring leads or substitution boxes. If you still have problems, you may have to place some small ceramic disk caps right in your circuit, from a noisy spot to ground. Be careful, though, this can seriously affect your circuit's frequency response.

IMPORTANT: *Remember to compensate your 10x probes before using them to make frequency dependent measurements.*

EXPERIMENT 1 TRANSISTOR DIODE TEST: (20 pts)

Procedure:



1. (20 pts) Multimeter transistor test

(a) Set your multimeter or ohmmeter to its diode test setting. Make a sketch of the transistor showing the leads as 1, 2, & 3, and a small table like **Fig. 2**.

(b) Measure the conductivity all six ways and record the meter readings in your table. The meter should only indicate significant conductivity in two of the six cases. The common lead to those two cases is the base.

- Determine which lead is the base. Determine from your data if the transistor is an NPN (base is + lead in both cases) or a PNP (base is - lead in both cases). Also, your lowest meter reading will often indicate the base/collector junction, and thus which lead is the collector.

RED +	BLK −	Meter Shows Conduction?
1	2	
1	3	
2	3	
2	1	
3	1	
3	2	

Fig. 2

(c) Look at the data sheet for this transistor to see if you were correct.

(d) Comment in your notebook about the usefulness of this procedure. (Note that h_{FE} is similar to β)

EXPERIMENT 2 DC OPERATION OF BJT: (25 pts)

Procedure:

1. (10 pts) Analyze the circuit to the right in **Fig. 3** to determine the DC values for $I_B, I_E, I_C, V_C, V_B, V_E$. Assume $\beta = 150$.

2. (10 pts) Use PSpice to simulate the circuit on the right. Run a transient analysis to determine the DC values from your hand analysis. Compare these values to hand analysis.

3. (5 pts) Build the circuit and measure all currents and voltages and compare these values to your hand analysis and PSpice results. Explain any differences.

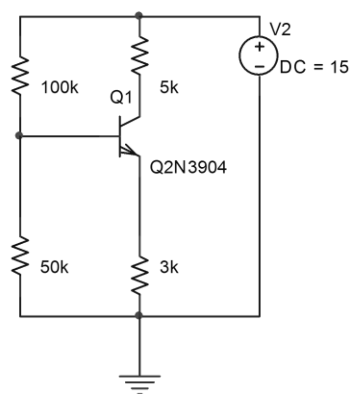
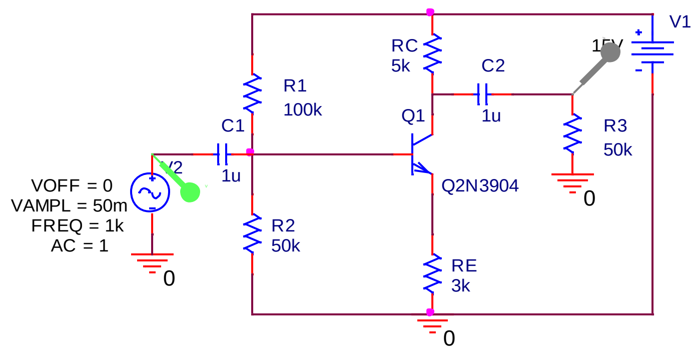


Fig 3.

EXPERIMENT 3 SIMULATIONS: (35 PTS)

Procedure:



1. (5 pts) Modify the circuit within PSpice to look like **Fig. 4**. Run a transient simulation for 2ms. Turn on the voltages and currents to verify that all voltages and currents are the same as DC analysis. Print the circuit with all currents and voltages showing. Measure the gain by measuring the input peak to peak value and the output peak to peak value. Note that the input and output is marked by the location of the probes. Print this graph and put into your lab notebook. Record the values for the input and output peak to peak values.

Fig. 4

2. (5 pts) Change the probe locations to be on the other terminal of the capacitors for both the input location and the output location. Rerun the transient simulation and observe the difference in the signal. Comment what the capacitors C1 and C2 are doing in this circuit.

3. (5 pts) Change the value of R2 to a parameter and perform a transient parametric sweep for 2ms over the following values for R2: 1k, 40k, 50k, 60k, 100k, 120k. Print these graphs at Vout which is the voltage above the R3 resistor. Explain what is happening in these graphs. If you need to understand it better, you can perform more simulations by changing R2 to a value and looking at the voltages and currents for one specific value of R2. Put any simulation results in your lab notebook along with your explanations.

4. (5 pts) Set R2=50k. Run an AC sweep(from 0.1 to 10Meg) and measure Vout/Vin. Print the magnitude bode plot result and place in your lab notebook. Create another table for gain, Rin, and Rout. Make a column for simulation and measurement. Record your value for the gain under the simulation results.

5. (5 pts) Change R3=50k to R3=100. Rerun the AC sweep and measure Vout/Vin. Print this magnitude bode plot and explain why it is attenuating the output signal.

6. (5 pts) Change R3 back to 50k. Use the following information to find Rin through simulation. This is a different way of measuring these values that is easier to do in the lab environment. You will simulate this method and then repeat the same measurement in lab.

To Measure Input Resistance:

Add a resistor (R_{test}) between the source and the circuit input (as shown in **Fig. 5**). Measure the ac signal voltage on both sides of R_{test} using the scope (peak-to-peak is alright).

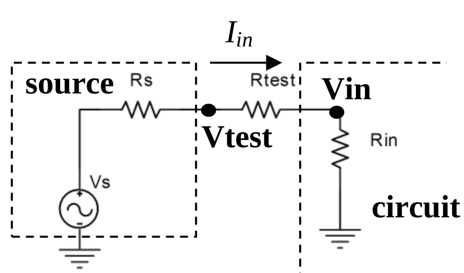


Fig. 5 **Formula 1.(**

R_{in} is the value you are trying to find.
To find R_{in}, use **Formula 1.0.**

$$I_{in} = \frac{V_{test} - V_{in}}{R_{test}} \qquad R_{in} = \frac{V_{in}}{I_{in}}$$

In **Fig. 5**, Rtest is unknown and needs to be added to the circuit. Modify the circuit to look like **Fig. 6**.

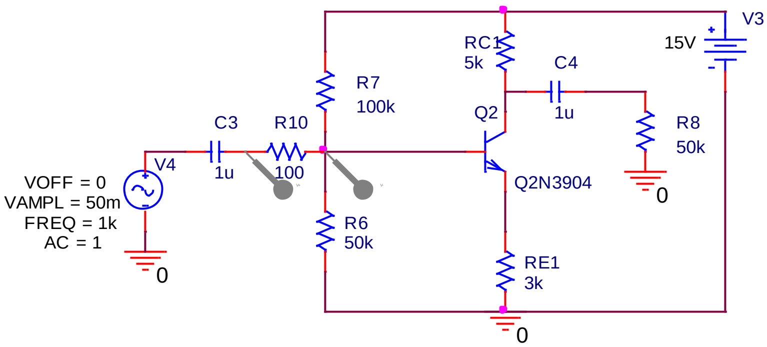


Fig. 6.

Note that R10 is the same as Rtest seen in **Fig. 5.** Measure the peak to peak value across R10 when a transient analysis is run. This gives the value for $V_{test} - V_{in}$. Measure the peak to peak value at the right side probe of R10. This is V_{in}. Calculate Rin from these measurements and record these calculations in your notebook.

7. (5 pts) Change the circuit back to look like **Fig. 4.** Now measure Rout using the method described below.

<u>To Measure Output Resistance:</u>
Measure the signal voltage output without R_L. R_L in **Fig. 4** is C2 and R3. This is the *open-circuit* output (V_{TH}). Remember that the open circuit voltage relates directly to the Thévenin voltage of a Thévenin equivalent circuit.

Reconnect R_L and measure the loaded output (V_L). Use these two measurements to calculate the output resistance (R_{out}) of this amplifier, using **Formula 1.1**.

Formula 1.1

$$R_o = \frac{V_{TH} - V_L}{V_L} \cdot R_L$$

Measure the peak to peak values. Replace both C2 and R3 of **Fig.** 4 and measure V_L (peak to peak value) at the node right above R3.

<u>IMPORTANT:</u> *Be sure to measure AC signal voltages (peak-to-peak is alright).*

<u>EXPERIMENT 3 MEASUREMENTS OF BJT CIRCUITS:</u> (40 pts)

Procedure:

1. (5 pts) Build the circuit of **Fig. 4**. Observe the output versus the input on the oscilloscope and print these graphs for your notebook. Record the peak to peak values for the input and output. Divide the output peak to peak value by the input peak to peak value. This is the gain of the circuit.
2. (5 pts) Multiply the gain by 0.708 (note that this is -3dB in V/V). Determine at what magnitude your output peak to peak will be for this new gain value.
3. (5 pts) Reduce the frequency from 1k until the output measures this new peak to peak value. This frequency is the starting frequency for the bandwidth of the circuit. Note its value.
4. (5 pts) Measure Rin and Rout the same way as you performed the simulation. Record their values and comment on the differences between the simulation and measured values.

The circuit of **Fig. 4** is a Common Emitter (CE) with Emitter Degeneration. **Fig. 7 and 8** are modifications which turn it into a Common Emitter (CE) amplifier.

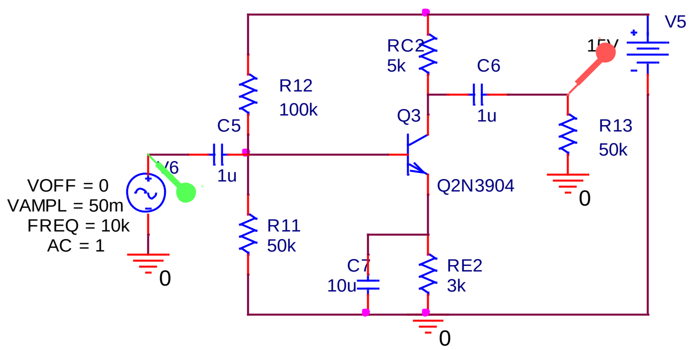
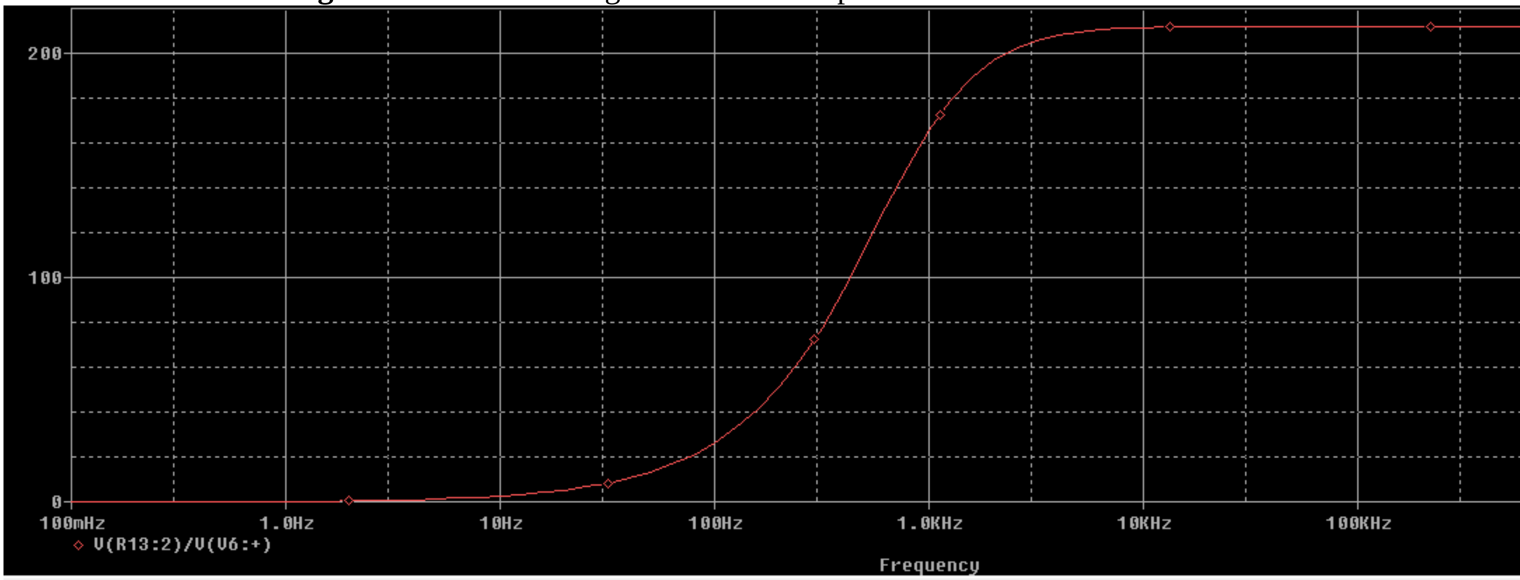


Fig. 7.

Simulation results for **Fig. 7** show the following for the AC sweep:



If the input amplitude of 50m is used, the output AC would be Vout=Vin*210 which results in Vout=10.5V. This would definitely pull the transistor out of saturation. Also, it is observed from this sweep that the bandwidth starts at approximately 900Hz. To test this circuit, the frequency needs to be higher. Test it at 10kHz. The input signal can be modified by a voltage divider as shown in **Fig. 8.**

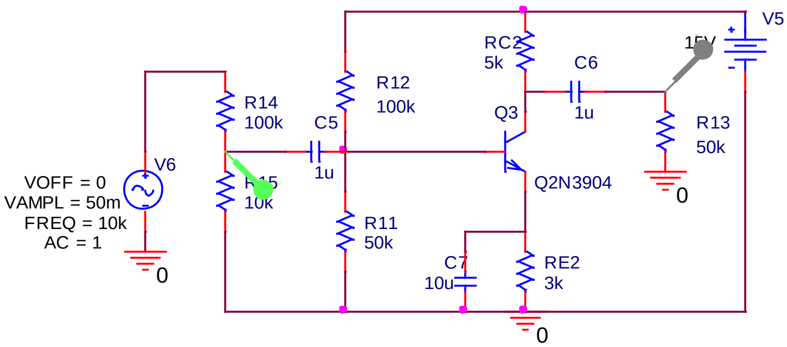
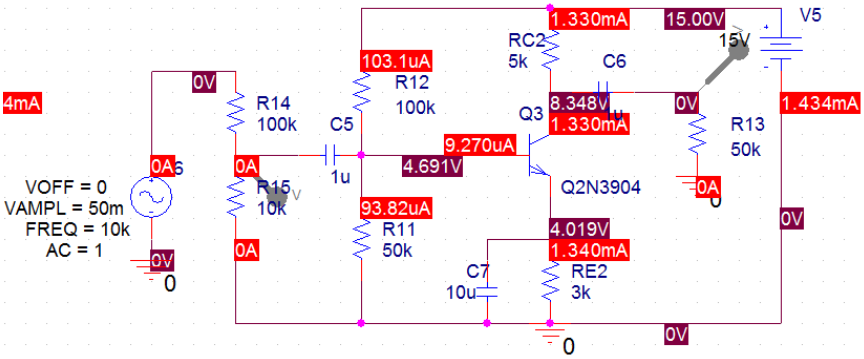
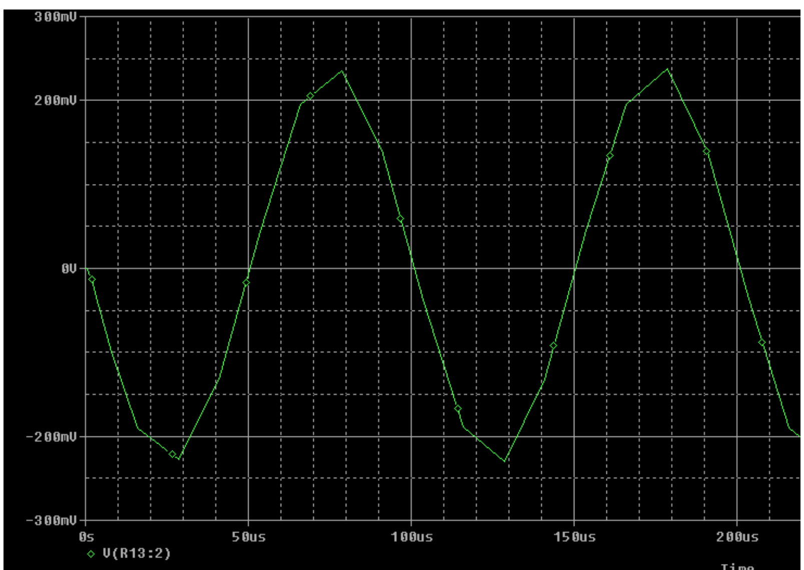
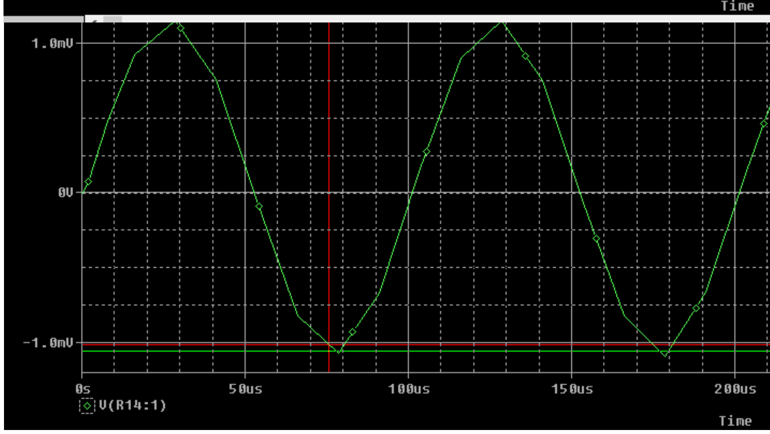


Fig. 8.

The transient analysis also showed the following results:







5. (10 pts) Modify the circuit to look like **Fig. 8.** Measure the gain of this circuit and compare to the simulation provided above. Comment on why this gain is so much higher than for the CE with emitter degeneration.

The circuit can be changed into a Common Collector (CC) which is seen in **Fig. 9.**

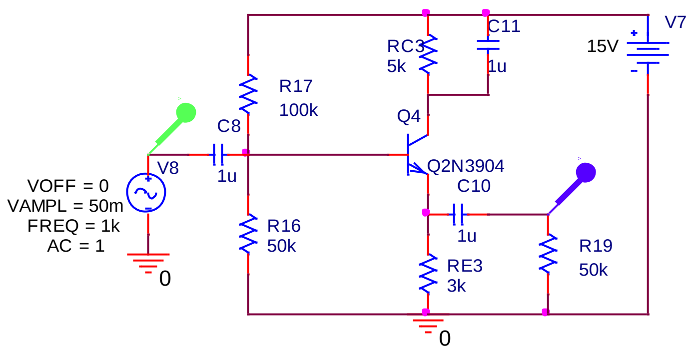
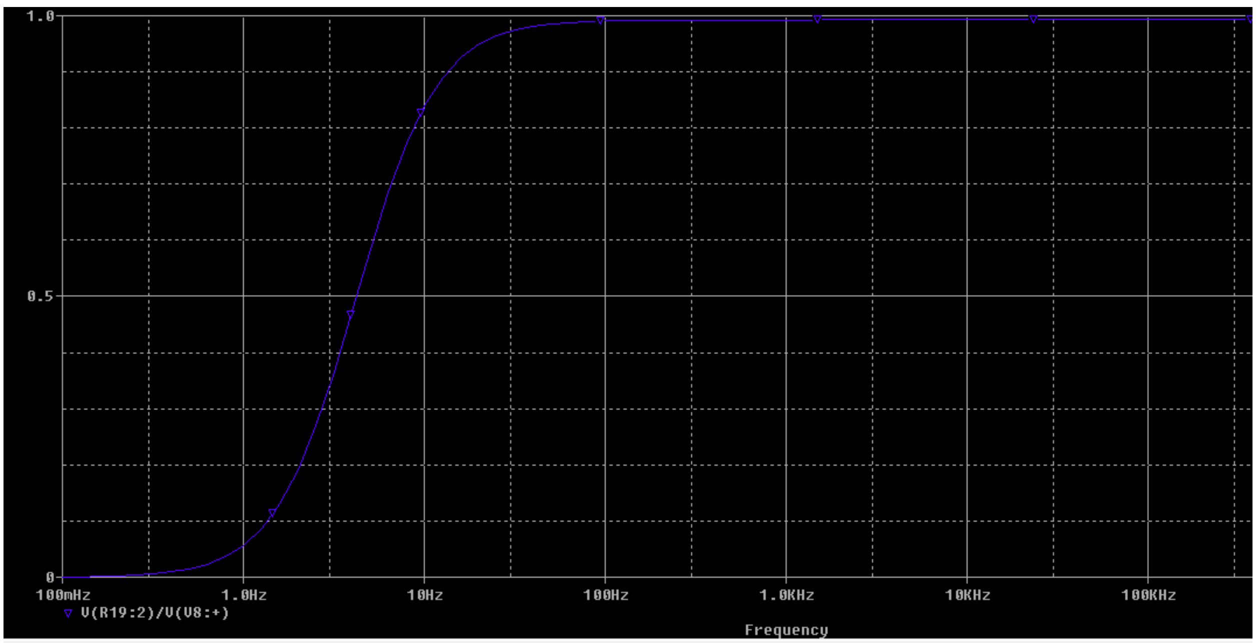
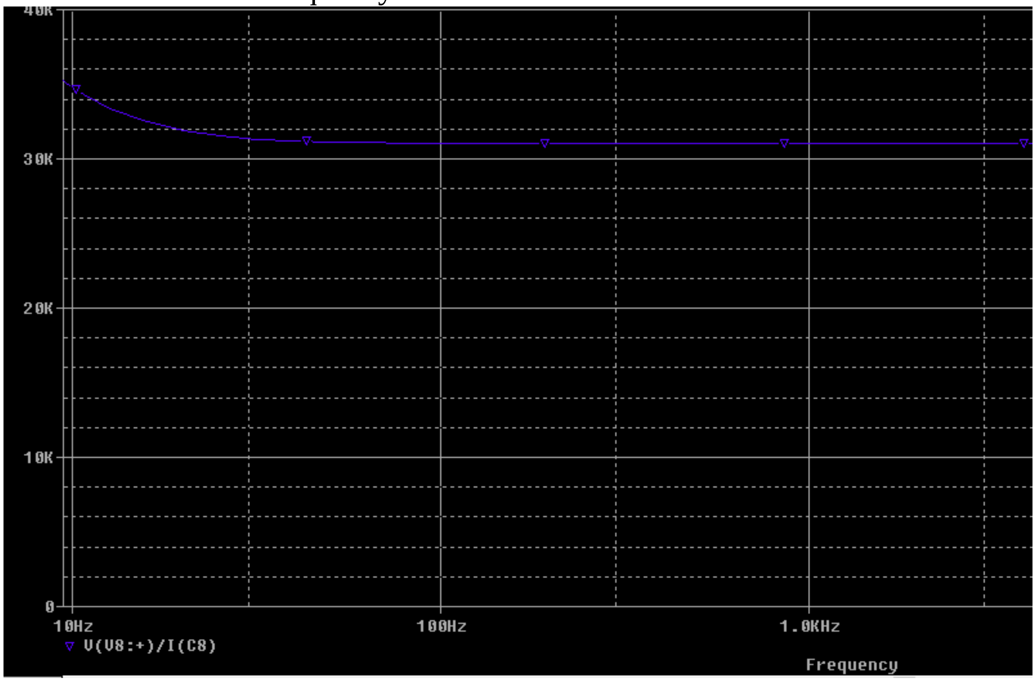


Fig. 9.

Simulation results for **Fig. 9** show the following for the AC sweep:



Rin simulation over frequency:



Rout calculation from simulation:
$V_{TH} = 99.2m$ (peak to peak)
$V_L = 99m$ (peak to peak)
$R_L = 50k$

$$R_{out} = \frac{V_{TH} - V_L}{V_L} R_L = \frac{(99.2m - 99m)}{99m} \cdot 50k = 101$$

6. (10 pts) Build the circuit in **Fig. 9.** Measure the gain, Rin and Rout. Compare the values to the simulated values.