

Lab 1: Further Adventures with LTSpice

Abstract

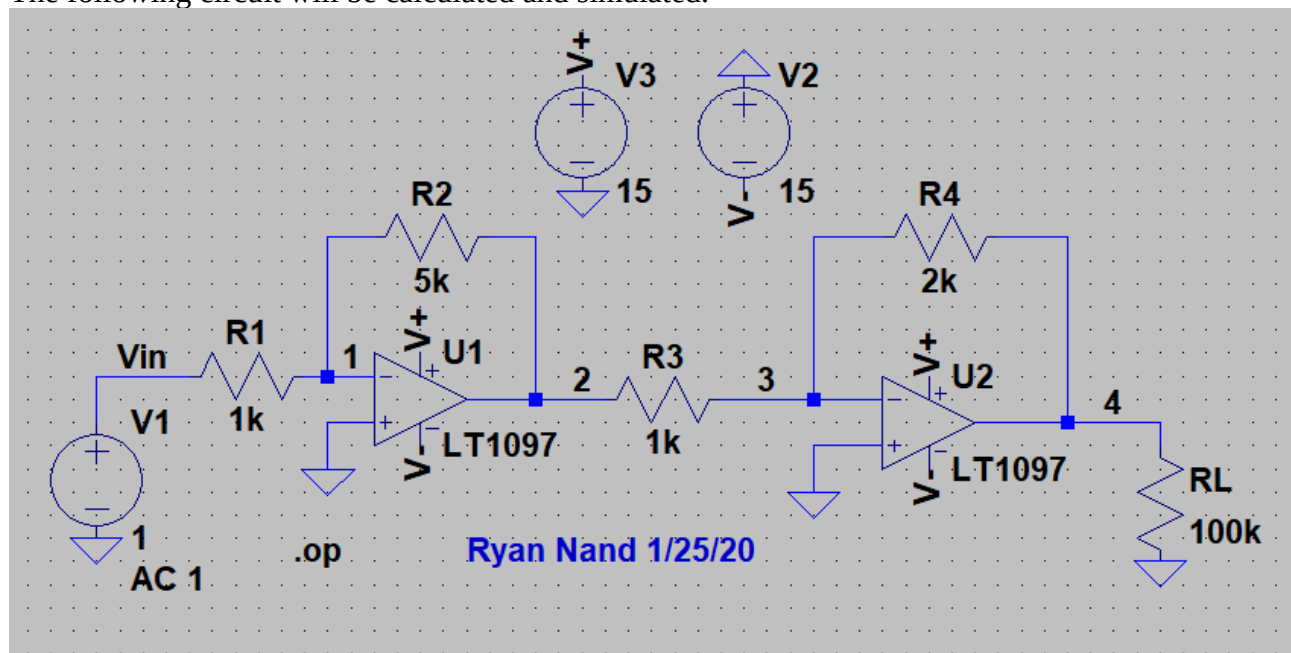
This experiment is to find similarities between the ideal models and LTSpice simulations. Comparing hand calculations and assumptions with LTSpice results. Going through this experiment we find several conforming results, along with reasons as to why simulations are a necessity to circuit design.

Introduction

The purpose of this experiment is to get familiar with the functions of LTSpice and all of its features. Several circuits throughout the experiment will be used comparing hand calculations to LTSpice simulations. Some of the circuits used will contain op amps, transistors, diodes, and MOSFETs. Both dc and ac analysis will be compared to ideal models. DC operating point, DC sweeps, AC sweeps, and variable components will also be utilized and evaluated.

Part 1: Op Amps

The following circuit will be calculated and simulated.



Calculations:

Using the ideal op amp model and looking at each op amp individually as two sections.

Section 1(op amp 1):

Since node 1 = $V_n = V_p = 0$, node 1 is zero volts.

$V_{out} = -V_{in}(R_f/R_i)$, in this case it is node 2 = $-V_1(5k/1k)$. So voltage at node 2 is -5V.

Section 2(op amp 2):

Same process as above for this op amp. However, the input for op amp 2 is the output of op amp 1.

Thus, $V_{in} = \text{node 2}$.

Node 3 = $V_n = V_p = 0$, so node 3 is zero also.

Node 4 = $-\text{node 2}(R_f/R_i)$, so node 4 is 10V.

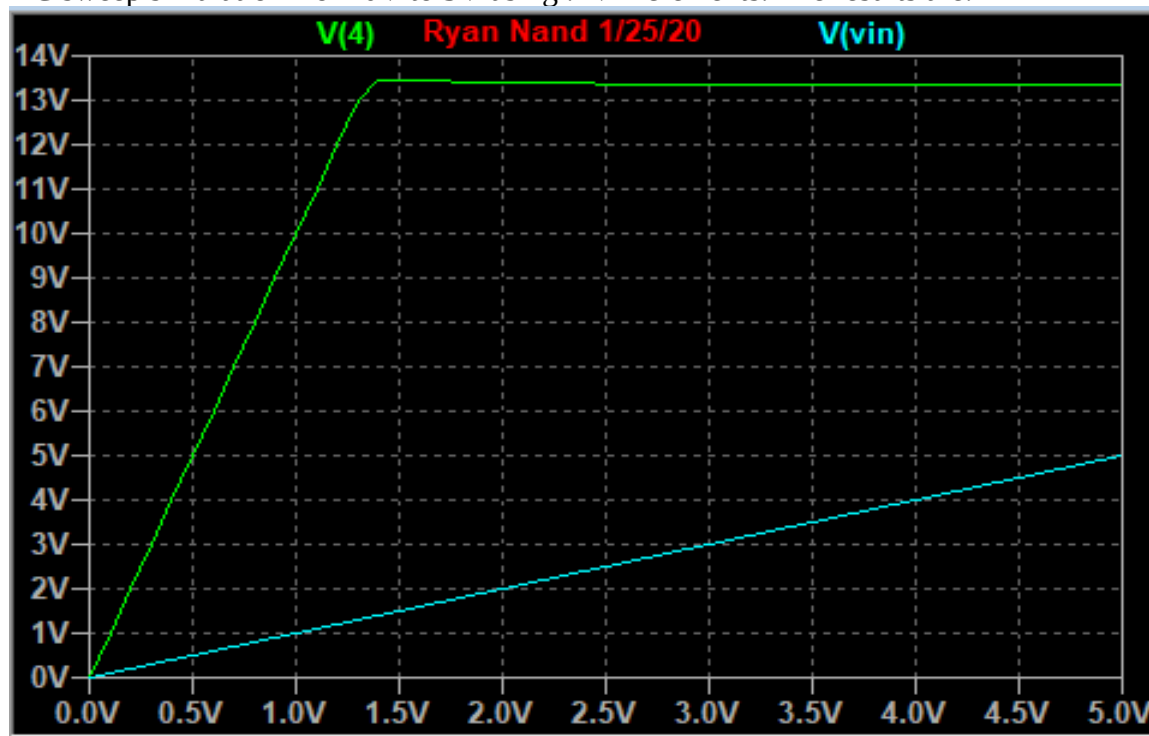
Lab 1: Further Adventures with LTSpice

Simulation results:
Using DC operating point simulation.

Node Voltages	Calculated(Volts)	Simulated(Volts)
1	0	2.459e-06
2	-5	-4.999
3	0	-4.628e-06
4	10	9.999

I would say that the results of the calculated and the simulations can define this circuit as close to ideal.

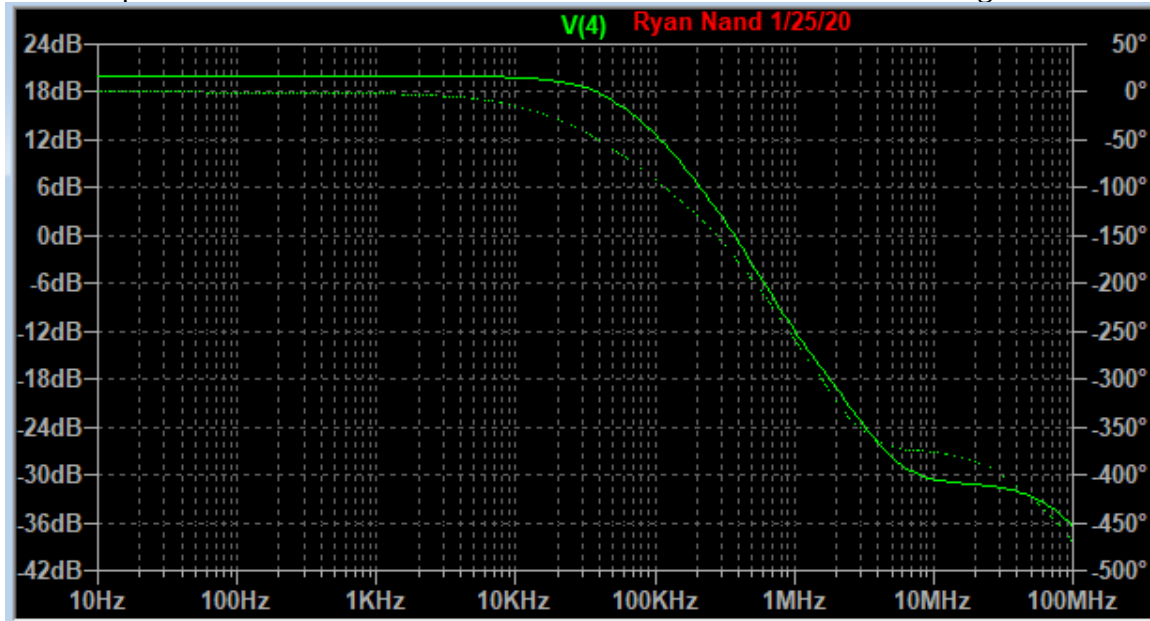
DC sweep simulation from 0V to 5V using .1V increments. The results are:



The main point of this simulation is to find the peak results. Due to the supply voltages of +15V and -15V the ideal op amp would peak at +15V or -15V. With calculation, one could find that the peak would be around 1.5V for the input (if gain 2 = -2 and gain 1 = -5, $15V/2 = 7.5$, $7.5/5 = 1.5V$ for V_{in}). The plots show the realistic results where the peak would be very close below 15V and not exactly 15V. Also, the input would very close below to 1.5V but not exactly 1.5V.

Lab 1: Further Adventures with LTSpice

AC sweep simulation from 10Hz to 100MHz in a decade scale. Small AC signal of 1V.



This results shows the gain vs. frequency. In other words, the gain vs. bandwidth limitations for this particular op amp(LT1097). The 20dB represents the total gain of 10 for this circuit (gain 1 = 5, gain 2 = 2, $A_{total} = 5 \times 2 = 10$, $20\log_{10} 10 = 20\text{dB}$). The plot shows that the bandwidth limitation for this op amp is around 50kHz for a gain of 20dB. Otherwise, this op amp will not work for higher frequencies.

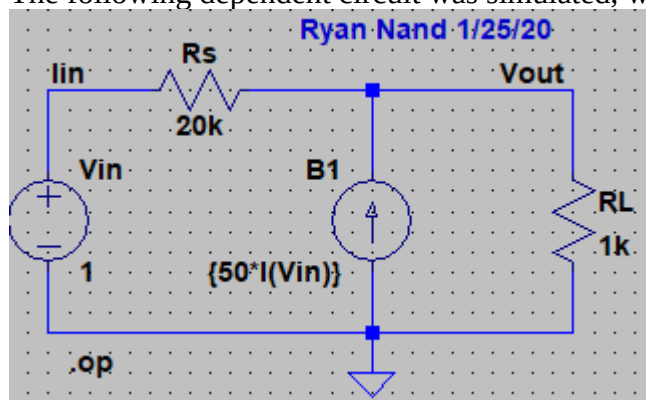
DC operating point simulation with variable load resistor from 100k to 1 ohms.

RL Values (ohms)	Vout(4)
100k-1	9.999V

Since the output of the DC op amp is not dependent on the load resistor, the Vout will still be 10V(just below 10V for non-ideal) for all values of RL.

Part 2: DC Analysis of Dependent Sources

The following dependent circuit was simulated, with varying alterations throughout simulation.

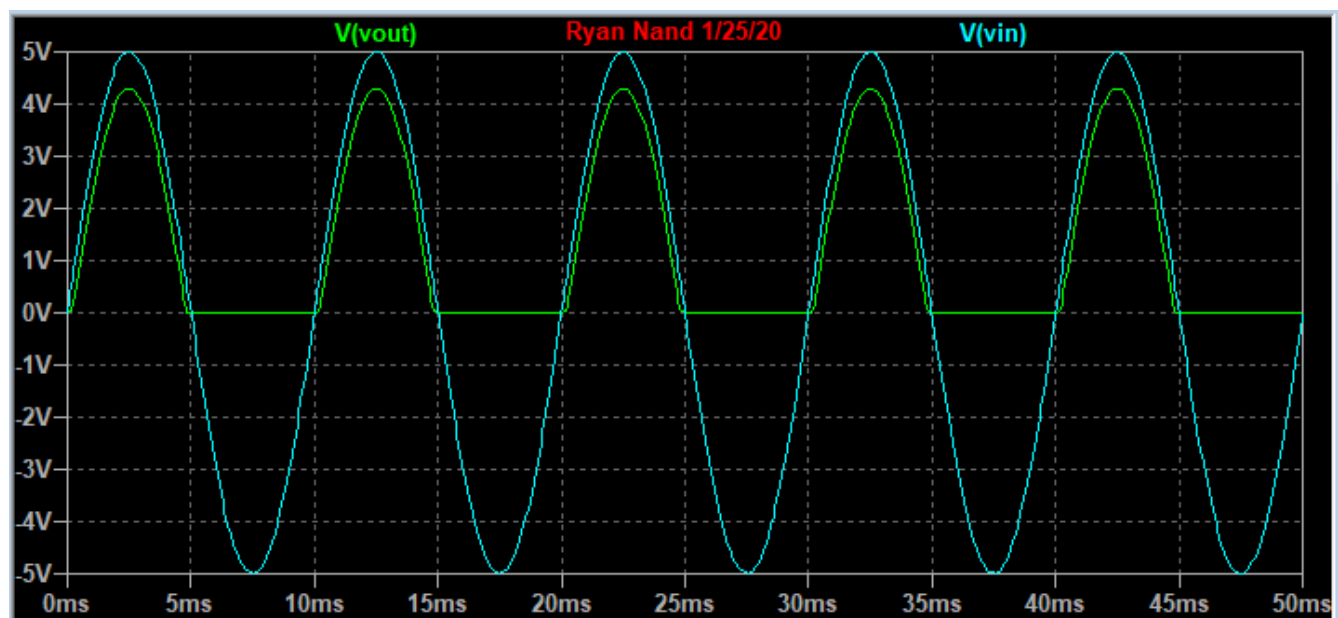
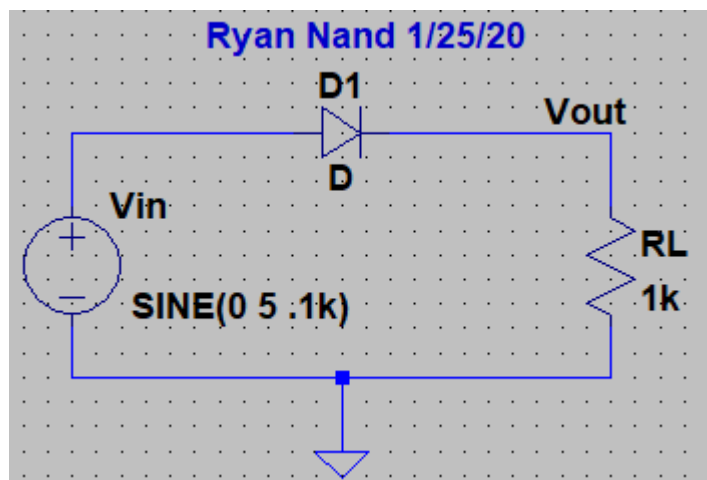


Lab 1: Further Adventures with LTSpice

Example	Calculated	Simulated
1.1	$V_{th} = .718v_i$	$V_{th} = 1.689$
1.2	$R_{th} = 282$	$R_{th} = 689.65$ ($V_x = 1$, $I_x = 1.45e03$)
1.3	$I_n = (2.55 \text{ mS})V_s$	$2.45e-03$
1.4	$R_{th} = 605k$	$R_{th} = 659.65$

Part 3: Diodes

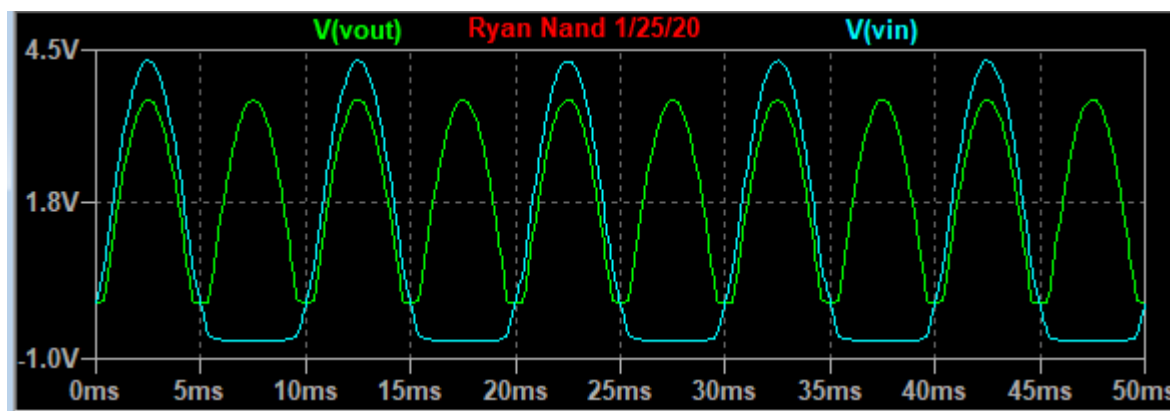
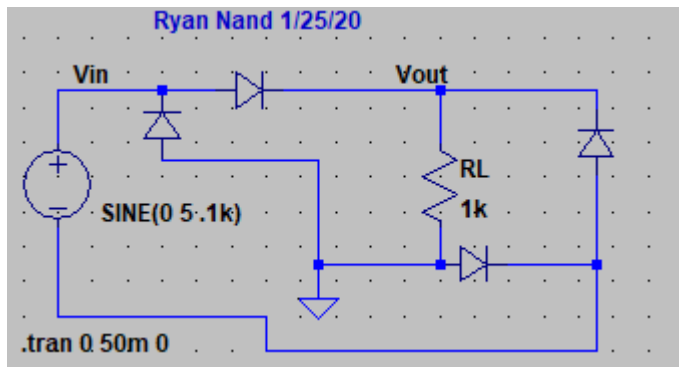
Circuit 1 Half Wave Rectifier



The half wave rectifier does just that. Converts a full 5V peak to peak sinusoidal wave to a 5V half wave like shown above.

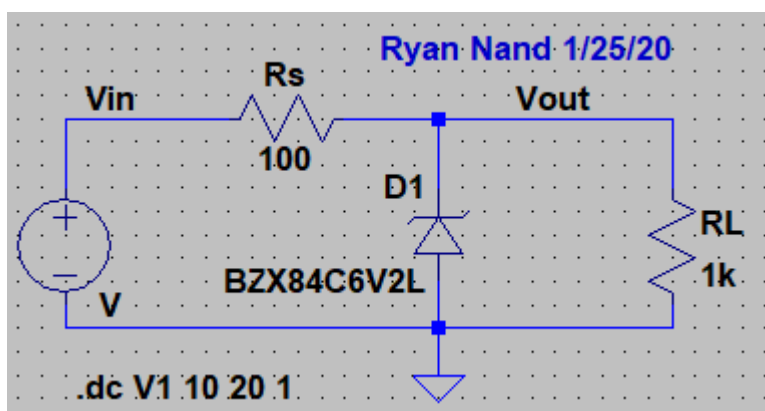
Lab 1: Further Adventures with LTSpice

Circuit 2 Full Wave Rectifier

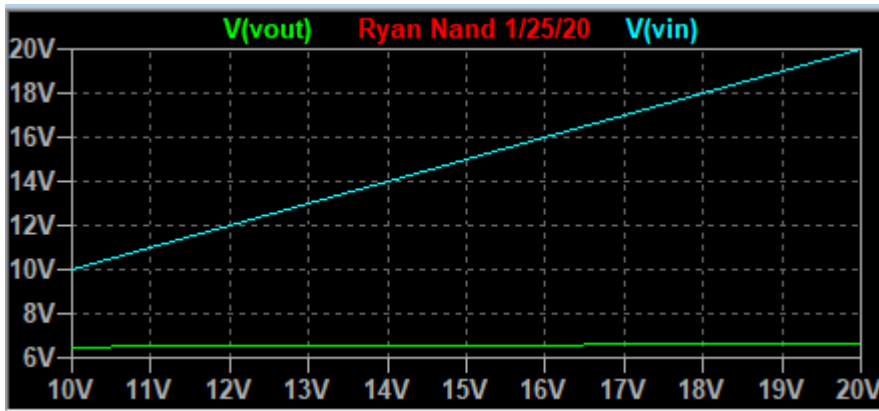


The purpose of a full wave rectifier is to convert ac to dc. Which is what the above plot shows, with non-idealistic results. However, the average of the wave is now around 2V with the offset and with twice the frequency. Each diode has about a .7V drop in voltage. For each peak the circuit is designed so that the current flows through two diodes before it reaches the output. Thus, $5V - .7V - .7V = 3.6V$ for the peak.

Circuit 3 Voltage Regulator

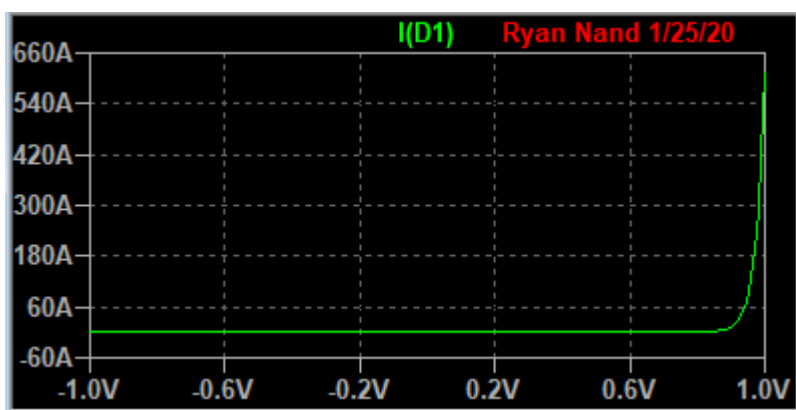
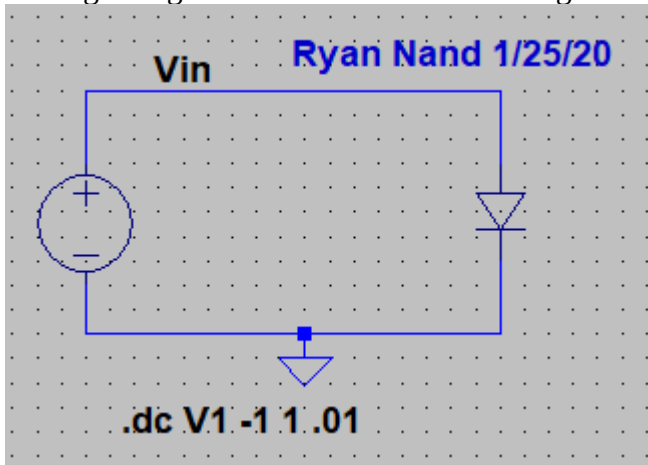


Lab 1: Further Adventures with LTSpice



This voltage regulator will regulate the voltage to about 6V. Does not let it go any higher which is what the purpose of the regulator is.

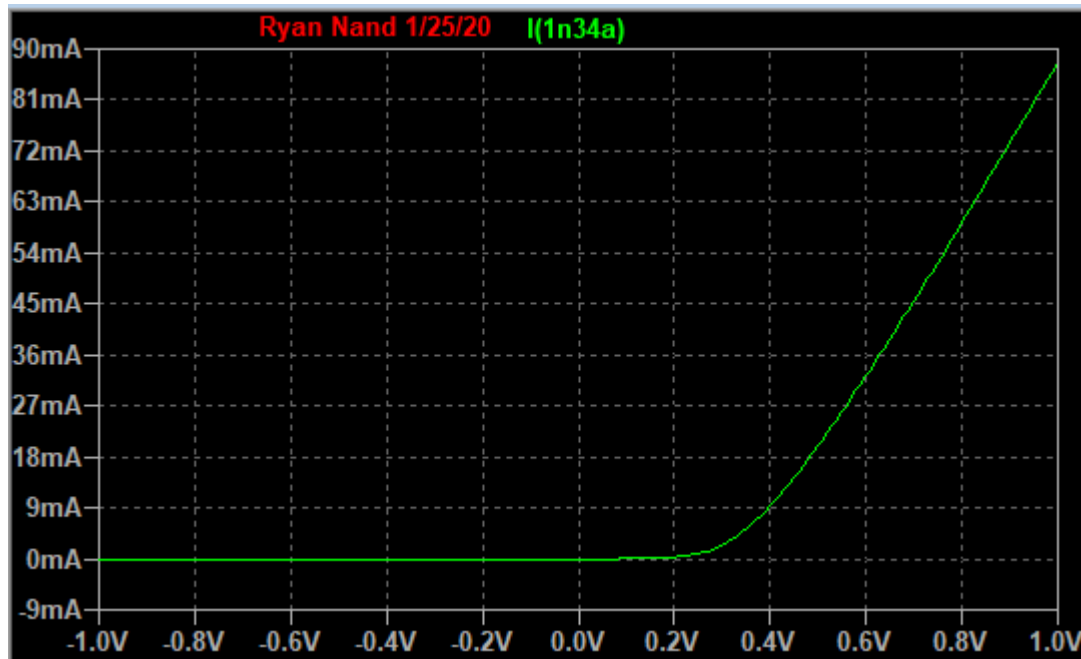
Testing a single diode circuit. The following circuit was used.



V_D about .9V, because current starts to flow through the diode around that voltage. According to the previous simulations the diode created a .7V drop. So it makes sense for this diode to be operating with a voltage higher than .7V which is around .9V.

Lab 1: Further Adventures with LTSpice

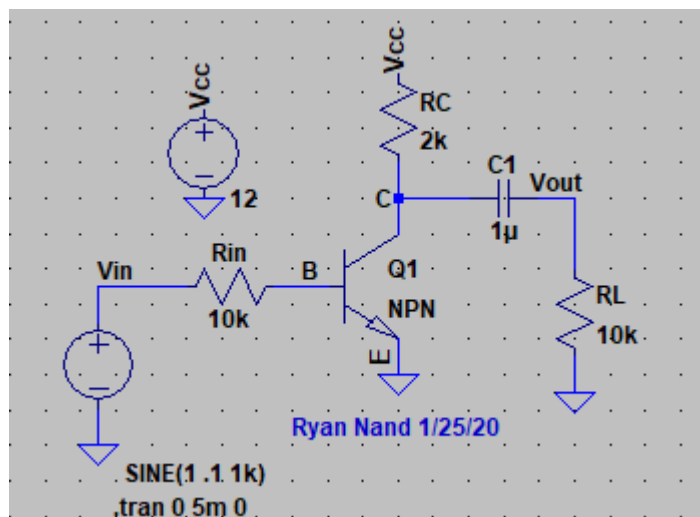
Using the same circuit, however, using a different diode model. Model 1N34A. The results are as follows.



The V_D is about .3V, where the current start to flow through the diode.

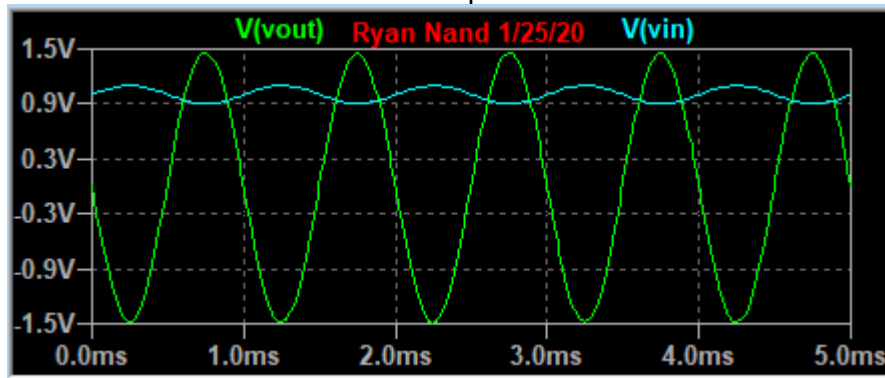
Part 4: Transistors

The following circuit is simulated:



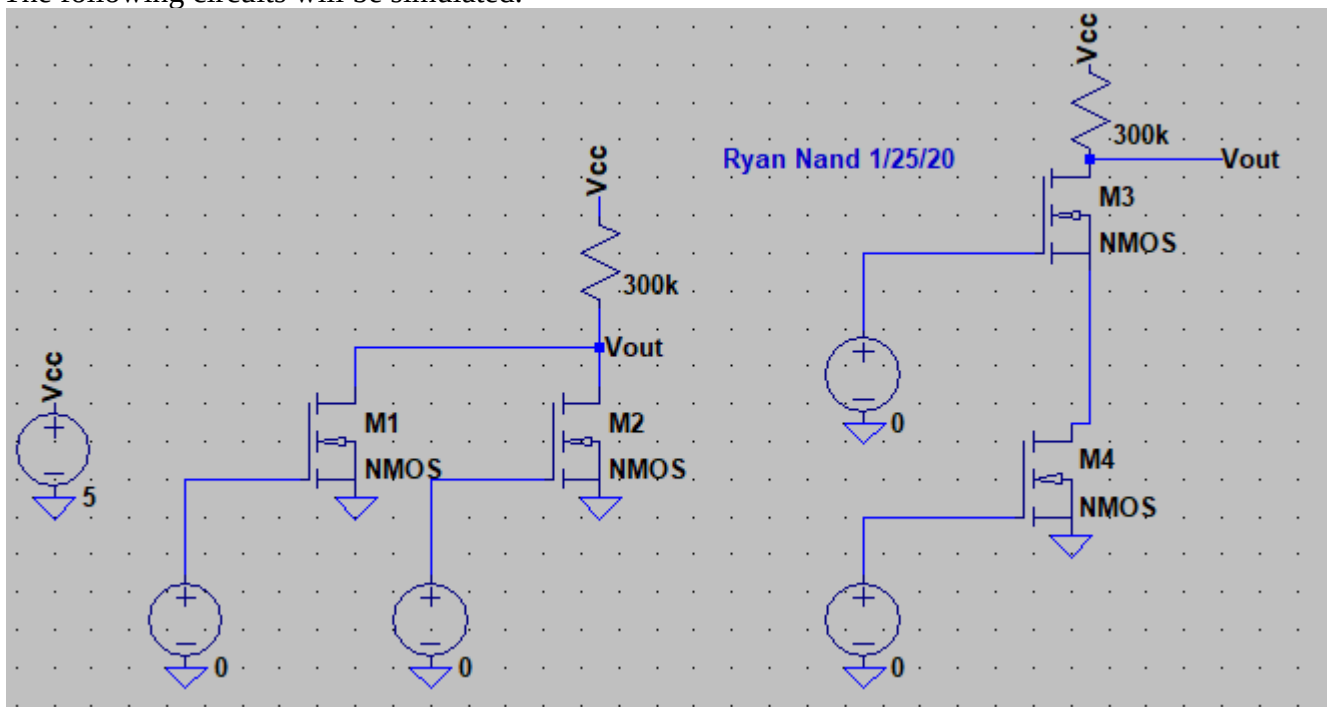
Lab 1: Further Adventures with LTSpice

Simulation results with transient response.



If the amplitude of the input is .1V to 1.5V. Then the gain is $1.5/.1 =$ a gain of 15. The purpose of the capacitor is to get rid off any offset at the output.

The following circuits will be simulated.



Lab 1: Further Adventures with LTSpice

For the series NMOS circuit.

V1	V2	Output
Low	Low	High
Low	High	Low
High	Low	Low
High	High	Low

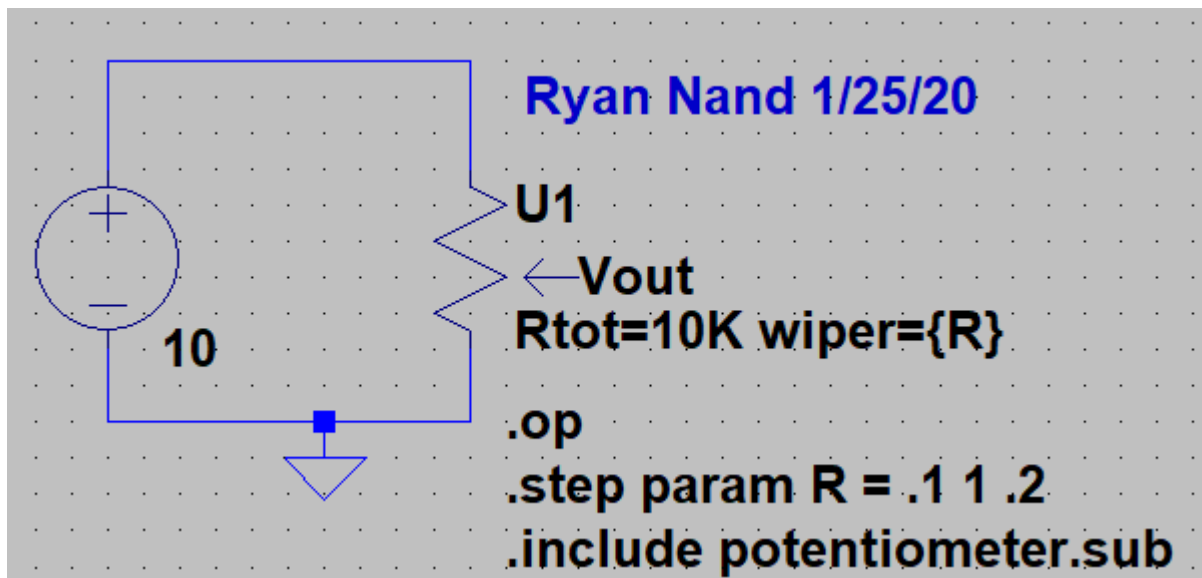
For the parallel NMOS circuit.

V4	V3	Output
Low	Low	High
Low	High	High
High	Low	High
High	High	Low

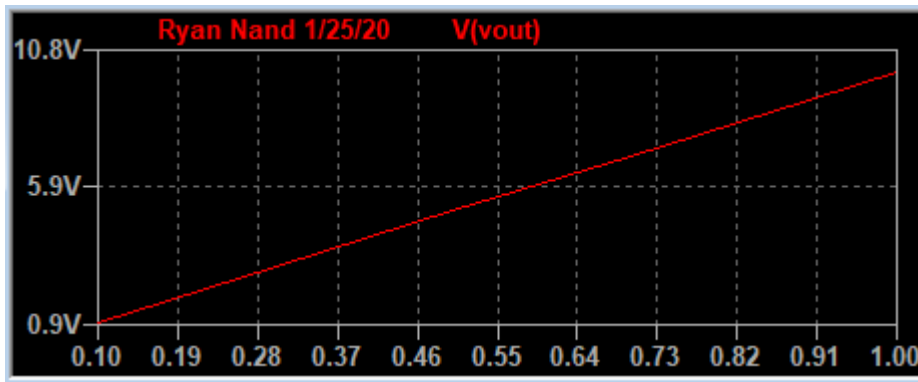
The series circuit is NOR logic and the parallel circuit is NAND logic

Part 5: Adding New Components and Variable Parameters

A variable potentiometer will be added to LTSpice and simulated. The following circuit will be used.



Lab 1: Further Adventures with LTSpice



A Pot ranging from .1 to 1 ohms.

Conclusion

With the following results above, it can be concluded that having simulations along with the hand calculations is very necessary. The results of the simulations can pinpoint human error, gives a second run through of the design, and has more realistic results. This is because ideal models do not take into account various real world problems like power dissipation, or real world components (limitation values). In the end, having simulations in the design process is a very good idea.