

# *San Francisco State University*

## *School of Engineering*

### **Engr 301 Final Exam Project**

#### **Low-noise Headphone Amplifier**

#### Introduction

For the final project in Engr 301, you will be designing and building a Low-noise Headphone Amplifier. You will design the circuit and the circuit board, assemble components, and test and adjust the unit. Grading will be based on completion, workmanship and functionality of the Headphone Amplifier. This project is in preparation for Engr 696/697 senior project. It will also be invaluable for showing prospective employers when applying for jobs.

#### Board Details

**Enclosure** (you may want to take advantage of non-working computer power supplies for that I've collected for parts)

- Power cord attached; power-on switch
- Fuse on secondary side (12 VAC), 1A
- LED Power On Indicator
- Terminals: 12VAC, L/R Line-in (1/4" or RCA), Headphone Output (stereo 1/8" )
- Control: Dual-Ganged Potentiometer for Stereo Volume Operation

#### **Circuit Board**

- The circuit board software called Eagle is free and downloadable from the iLearn course page.
- Maximum board size: **3" x 3"**
- Labeling: **Headphone Amplifier, San Francisco State University, School of Engineering, <date>, <your name>**
- Circuit board files must be named with your last name so software can be readily identified.
- Need 4 mounting holes at the 4 corners. The size of **the hole = 4-40 (2.8 mm)** machine screw
- School of Engineering will be pay for the cost of the circuit board only if timely turned in

#### Submission on iLearn

- Two Eagle files include .sch and .brd
- Gerber files: all the files generated from Eagleware

# Headphone Amplifier Simulation

Project design can be broken into two simple circuit: Non-inverting Amplifier, and feedback circuit. In order to better understand the design and to be able to change the design's function, strongly advise you to simulate the circuit design by LTSpice to understand the design's ability and its function. It is great opportunity to understand industrial procedure from design to layout, and practice your own ability to handle a project.

Before starting board design, please download the LTSpice Template from iLearn and analyze the circuit design from simulations. The templates are named as below on iLearn:

1. Non-inverting Amplifier
2. Voltage Divider
3. Low-Noise Amplifier
4. Rectifier
5. Power Supply

Those templates are in the folder called **Project LTSpice Template**. Please download the folder and double click on the template you are going to simulate.

## I. Non-inverting Amplifier

Please open *Non\_inverting Amplifier* template and the schematic is exactly look like Fig.1

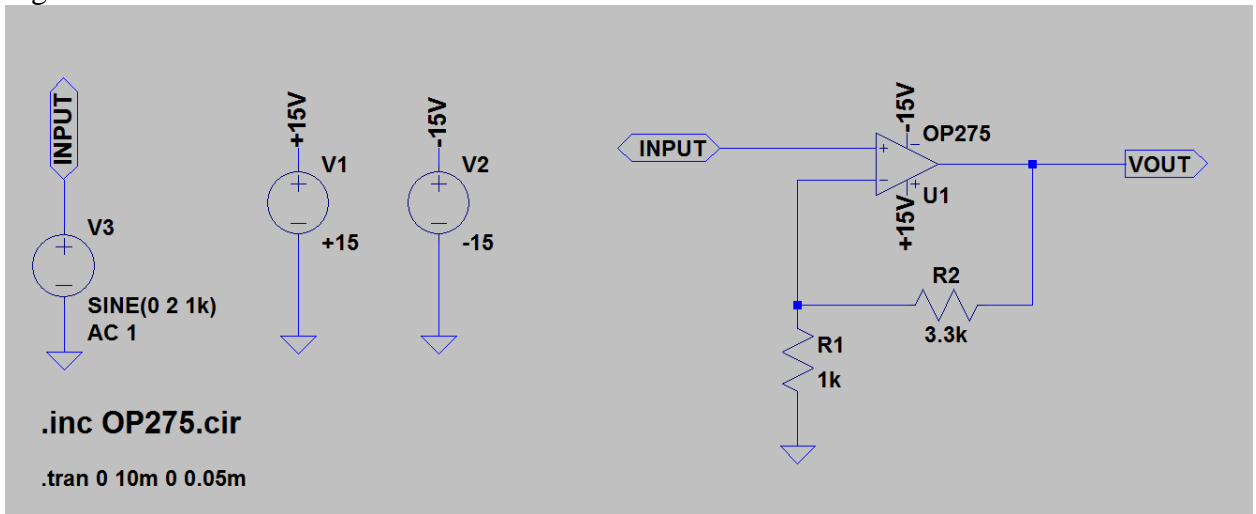


Fig.1 Non-inverting Amplifier

And click on a running-person symbol in the tool bar on the top of the window (Fig. 2)



Fig.2 Tool Bar

Click on VOUT and INPUT to get the simulation result as Fig. 3

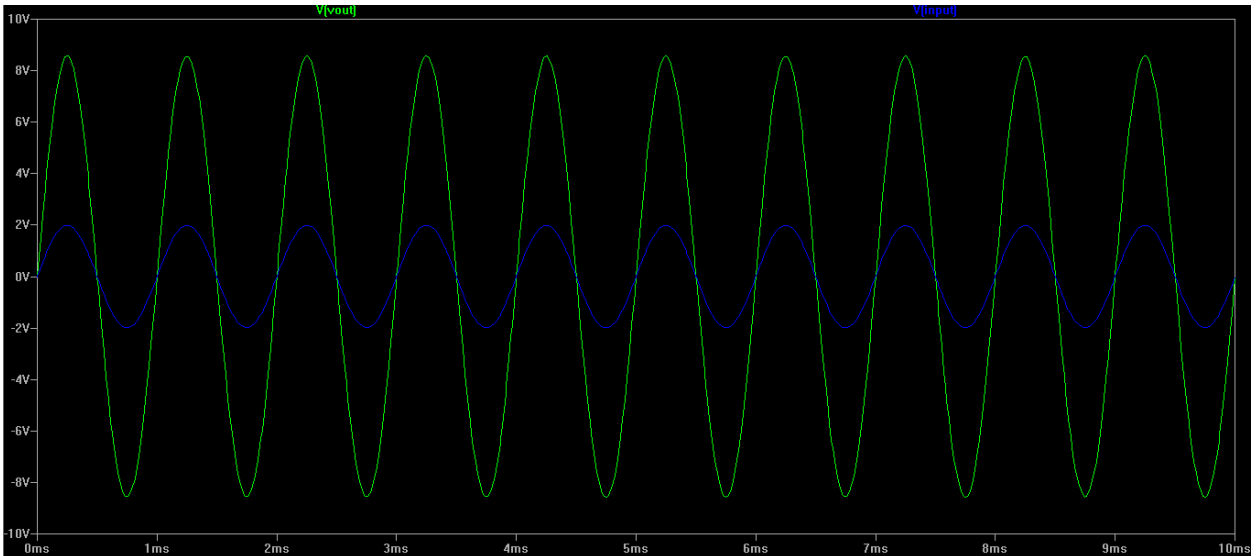


Fig.3 Non-inverting Amplifier Simulation Result

Form the equation of non-inverting amplifier:

$$DC \text{ Gain} = DC \text{ gain} = 1 + \frac{R_2}{R_1} = 1 + \frac{3.3k\Omega}{1k\Omega}$$

$$DC \text{ Gain} = 1 + \frac{3.3k\Omega}{1k\Omega} = 4.3, \text{ for } R_2 = 3.3k\Omega \text{ and } R_1 = 1.0k\Omega .$$

Questions:

Please measure maximum output voltage and maximum input voltage and find out by the

DC Gain by the equation  $DC \text{ gain} = 1 + \frac{\text{Output Voltage}}{\text{Input Voltage}}$

How much the DC gain for  $R_2 = 6.3k\Omega$  and  $R_1 = 1.5k\Omega$ ? Please compare calculation result and simulation result.

## II. Voltage Divider

In the design, there are two methods to change the op-amp gain:

1. By changing  $R_1$  and  $R_2$  value of the non-inverting amplifier, the value of  $gain = 1 + \frac{R_2}{R_1}$ .

2. Change the potentiometer resistance.

The potentiometer acts like a voltage divider. This is the equation for calculating output

signal of voltage divider:  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input$

For instance

1.  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = \left( \frac{2k\Omega}{2k\Omega + 8k\Omega} \right) * 2V = 0.4V$  for  $R_1 = 2k\Omega, R_2 = 8k\Omega$ , and  $input = 2V$ .
2.  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = \left( \frac{8k\Omega}{2k\Omega + 8k\Omega} \right) * 2V = 1.6V$  for  $R_1 = 2k\Omega, R_2 = 8k\Omega$ , and  $input = 2V$ .

3.  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = \left( \frac{5\text{ k}\Omega}{5\text{ k}\Omega + 5\text{ k}\Omega} \right) * 2 = 1\text{ V}$  for  $R_1 = 5\text{ k}\Omega$ ,  $R_2 = 5\text{ k}\Omega$ , and  $input = 2\text{ V}$ .

You can change the resistance of a potentiometer by following steps (LTSpice template name *Voltage Divider*):

1. Right click on a potentiometer, it will pop out a window as below

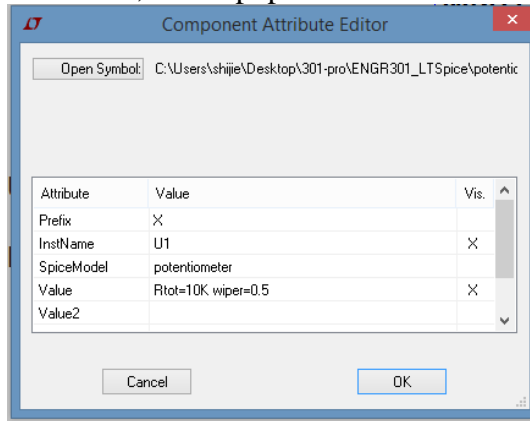


Fig.4

2. Setup the total resistance of the potentiometer and output voltage. You can set up those value on the red rectangle.  $R_{tot}$  is the total resistance of potentiometer ( $R_{tot} = 10\text{ KOhm}$  in Fig. ). Wiper = 0.5 indicates output equal to 50% of input.

For instance:

- 1)  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = wiper * input = 0.5 * 2\text{ V} = 1\text{ V}$  for  $wiper = 0.5$  and  $input = 2\text{ V}$ .

- 2)  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = wiper * input = 0.2 * 2\text{ V} = 0.4\text{ V}$  for  $wiper = 0.2$  and  $input = 2\text{ V}$ .

- 3)  $output = \left( \frac{R_1}{R_1 + R_2} \right) * input = wiper * input = 1 * 2\text{ V} = 2\text{ V}$  for  $wiper = 1$  and  $input = 2\text{ V}$ .

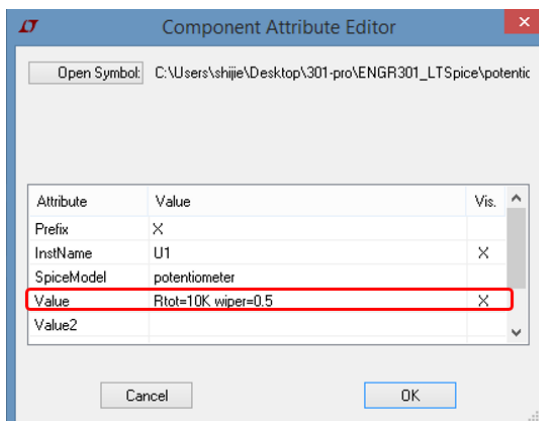


Fig. 5

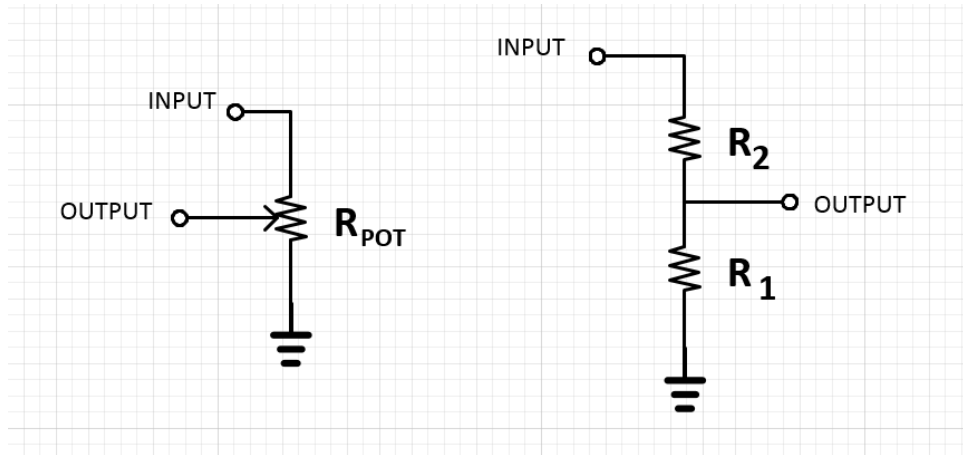


Fig. 6 Potentiometer (on the left) Acts as Voltage Divider (on the right)

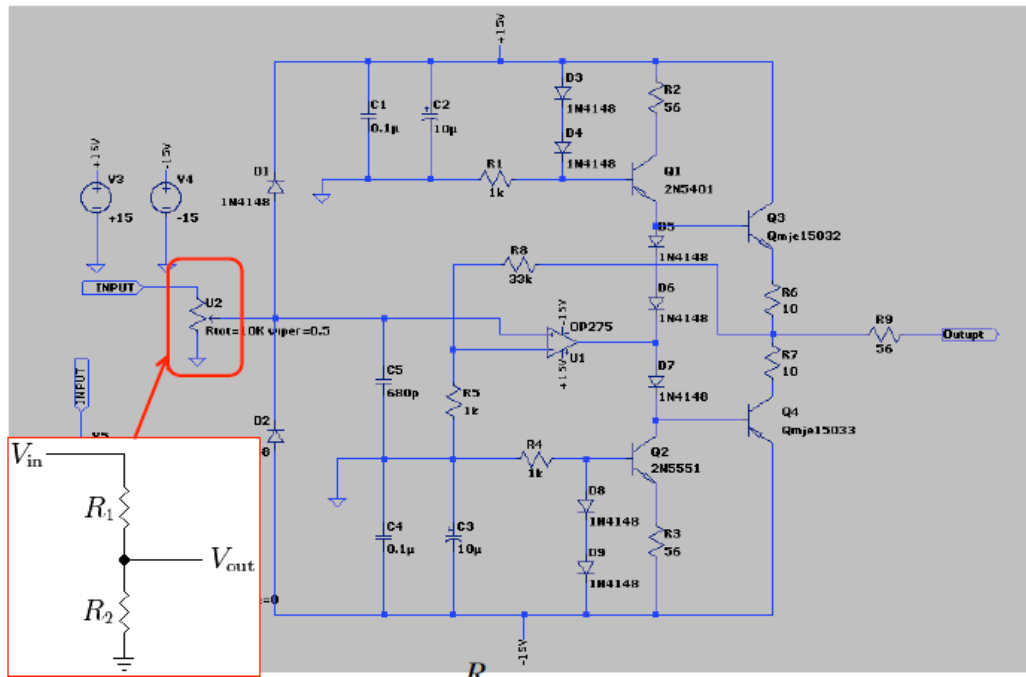


Fig. 7 Potentiometer Act as Voltage Divider

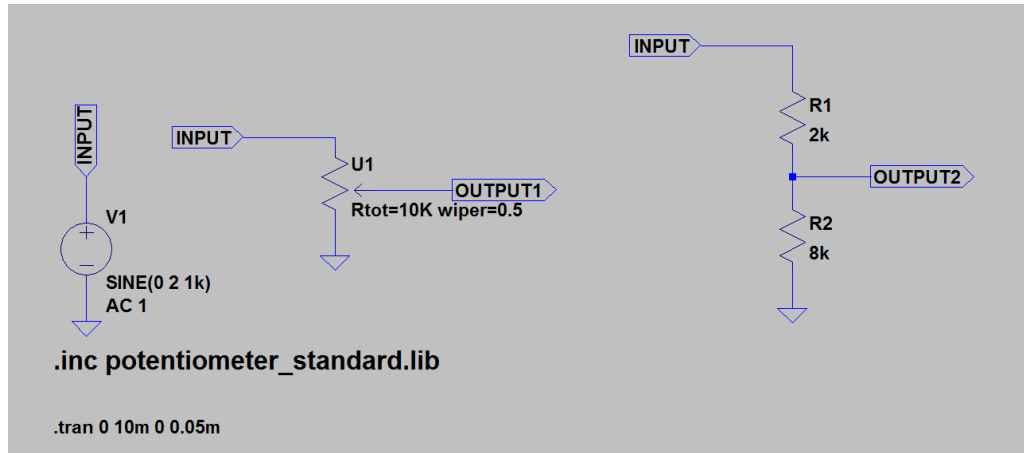


Fig.8 Potentiometer LTSpice setup

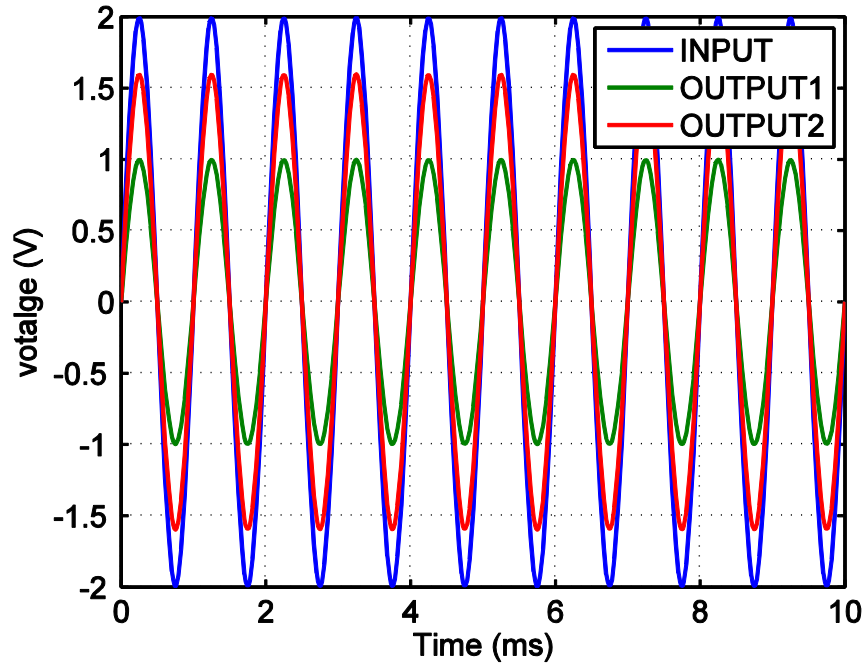


Fig. 9 Simulation Result of Fig.5 Setup (wiper = 0.5,  $R_1 = 2 \text{ KOhm}$ ,  $R_2 = 8 \text{ KOhm}$ )

The gain of ENGR301 final project is

$$\text{opamp Gain} = \frac{\text{output voltage}}{\text{input voltage}} = \text{wiper} * \text{DC Gain}$$

$$\text{output} = \text{wiper} * \text{Noninverting Amplifier DC Gain} * \text{input}$$

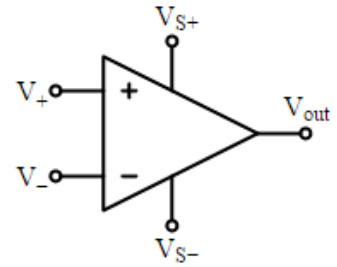
For instance, the setup of op-amp design likes that:  $\text{wiper} = 0.5$ ,  $R_1 = 2 \text{ k}\Omega$ ,  $R_2 = 8 \text{ k}\Omega$ , and  $\text{input} = 2 \text{ V}$ . The gain and output voltage is

$$\begin{aligned} \text{Noninverting Amplifier DC Gain} &= \left( \frac{R_1}{R_1 + R_2} \right) * \text{input} = \left( \frac{2 \text{ k}\Omega}{2 \text{ k}\Omega + 8 \text{ k}\Omega} \right) * 2\text{V} \\ &= 0.4 \text{ V} \end{aligned}$$

$$\begin{aligned} \text{output} &= \text{wiper} * \text{NonInverting Amplifier DC Gain} * \text{input} = 0.5 * 0.4 * 2\text{V} \\ &= 0.2 \text{ V} \end{aligned}$$

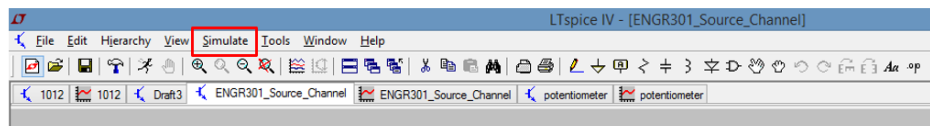
## Voltage Swing

Even though we can modify the value of  $R_1$  and  $R_2$  to get a high DC gain and AC gain, operational amplifier has its certain output range depended on source voltage  $V_{S+}$  and  $V_{S-}$ . In the our design, the voltage sources is equal to  $\pm 15V$  ( $V_{S+} = 15V$  and  $V_{S-} = -15V$ ), therefore, the output of the amplifier is within the scale:  $-15V \leq \text{output} \leq +15V$ .

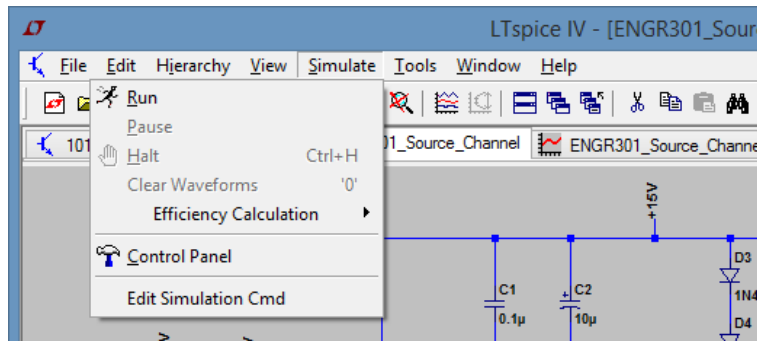


LTSpice allow you to find out the voltage swing by DC Sweep Demand. Please follow steps by steps to set up the simulation:

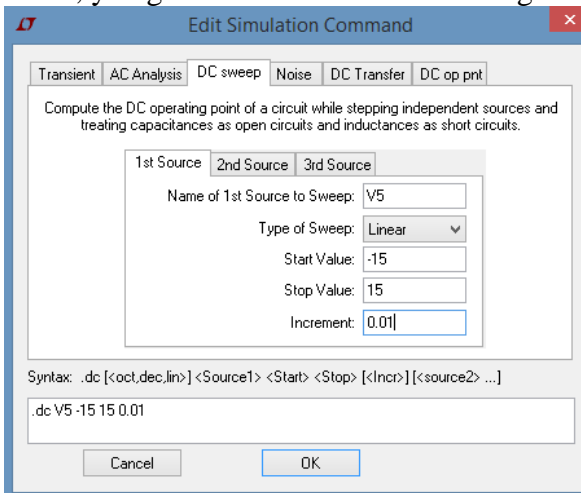
1. Click on **Simulation** on the tool bar



2. Choose **Edit Simulation Cmd**



3. Click on **Edit Simulation Cmd** to pop out an **Edit Simulation Command** window, choose DC sweep and setup the parameter exactly like Fig. Run the simulation, you get the simulation result as figure below



Waveform View (Fig.10) shows the output voltage swing in the scale:  $-13.69V \leq \text{output} \leq +13.69V$ . This value indicates that the max voltage of output signal  $\pm 13.69$  V. The slope of the op-amp output is equal to its DC gain. If the output voltage excess the max scale, the output signal is clipped off (Fig.11).

There is an online tutorial in Youtube called LTSpice Waveform Viewer. Can you copy the link in index of this manual and watch the video in browser.

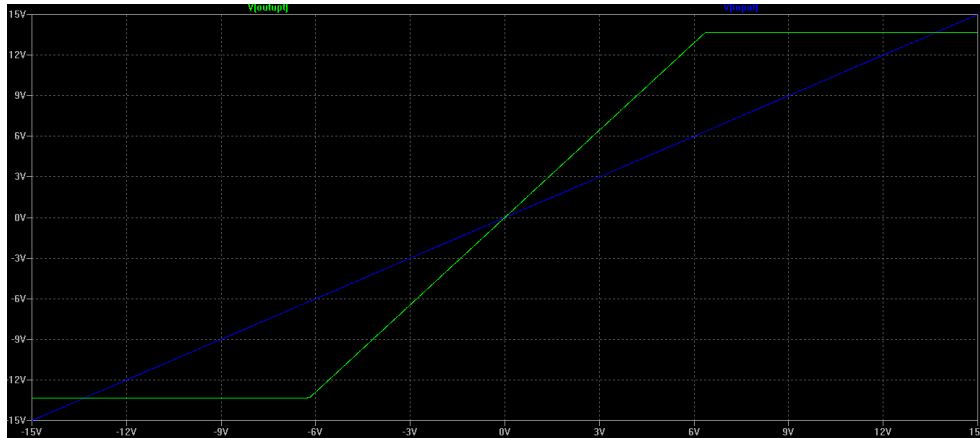


Fig. 10 Voltage Swing



Fig. 11 Clip off The Peak of Output Signal (Green Line)

## AC Gain

While we are used the amplifier to enlarge the value of a music song, the input signal is a sound wave in frequency 100-4000 Hz. That is necessary to know AC Gain of amplifier. Fig. is bode plot diagram. Magnitude from 0 to 35 KHz is a straight line (approximate 6dB) is DC Gain, or Low-Frequency Gain. We can use the equations below to convert DC Gain:

$$XdB = 20 \log_{10} Y$$

$$Y = 10^{\frac{X}{20}}$$



We get 6 dB in simulation result, we able to find out Y for X = 6, wiper = 0.5, R<sub>1</sub> = 1 KOhm, R<sub>2</sub> = 3.3 KOhm

$$Y = 10^{\frac{X}{20}} = 10^{\frac{6}{20}} \cong 2$$

We get DC Gain by the equations:

$$\text{Noninverting Amplifier DC Gain} = \left(1 + \frac{R_2}{R_1}\right) * \text{input} = \left(1 + \frac{3.3 \text{ k}\Omega}{21 \text{ k}\Omega}\right) = 4.3 \text{ V}$$

$$\begin{aligned} \text{DC Gain} &= \frac{\text{OUTPUT}}{\text{INPUT}} = \text{wiper} * \text{NonInverting Amplifier DC Gain} * \text{input} \\ &= 0.5 * 4.3 = \mathbf{2.15} \end{aligned}$$

This result indicates Low-frequency Gain is closed to DC Gain.

The frequency at 3dB lower than DC gain is called bandwidth. AC Gain is decreased with frequency increased. Gain BandWidth Product (GBWP) is the method to calculate approximately AC Gain at high frequency.

For instance, we got DC Gain = 2 and bandwidth = 104.9 KHz in previous simulation.

We can get GBWP by the equation:

$$\text{GBWP} = \text{DC Gain} * \text{Bandwidth} = 2 * 104.9 \text{ KHz} = 209.8 \text{ KHz}$$

If input signal is at 150 KHz, we are able to assume AC Gain by the equation:

$$\text{AC Gain} = \frac{\text{GBWP}}{\text{Frequency}} = \frac{209.8 \text{ KHz}}{150 \text{ KHz}} \cong 1.39 \cong 2.73 \text{ dB}$$

So you can assume the amplifier has 2.73 dB at 150 KHz. The frequency is an approximately value. We can get the exactly value (about 1.08dB) in simulation.

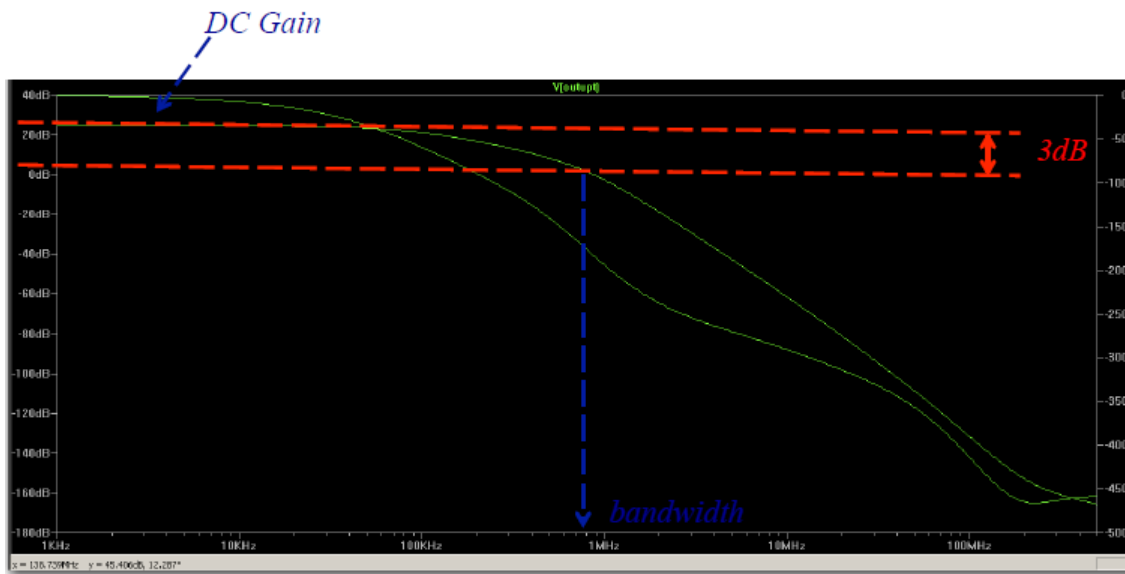


Fig.12 Bandwidth and Bandwidth Product

### III. Power Supply

Project circuit (you can download the circuit diagram from iLearn) indicates that the power supply circuit is connected with an external power source (wall adaptor with 12VAC, 60 Hz). For better understanding power supply circuit, the circuit can be simplified as Fig.13. The circuit rectifies input voltage by diode. You can read the lab manual with tile *DIODE CHARACTERISTICS AND APPLICATIONS* to understand rectifier concept.

Please run LTSpice template named *rectifier* for Fig.13 simulation and run the LTSpice template named power supply Fig.14 to simulation the entire power supply circuit from the project.

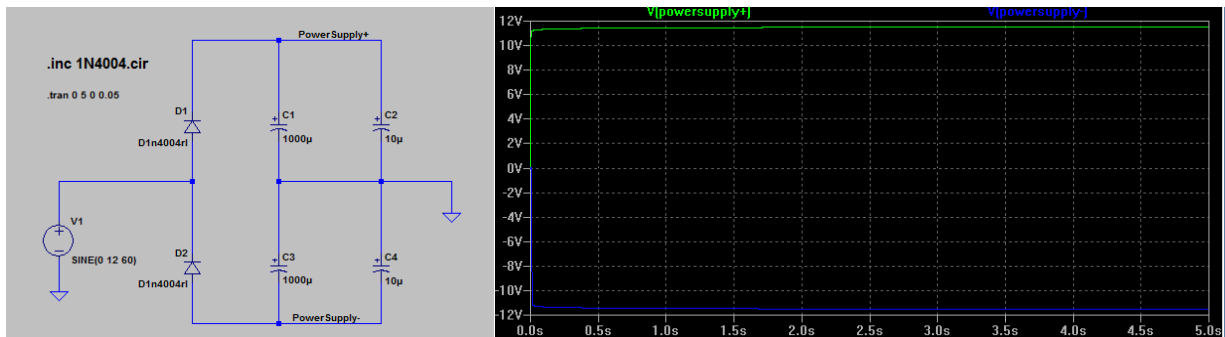


Fig. 13 Simplified Power Supply Circuit from ENGR301 Project

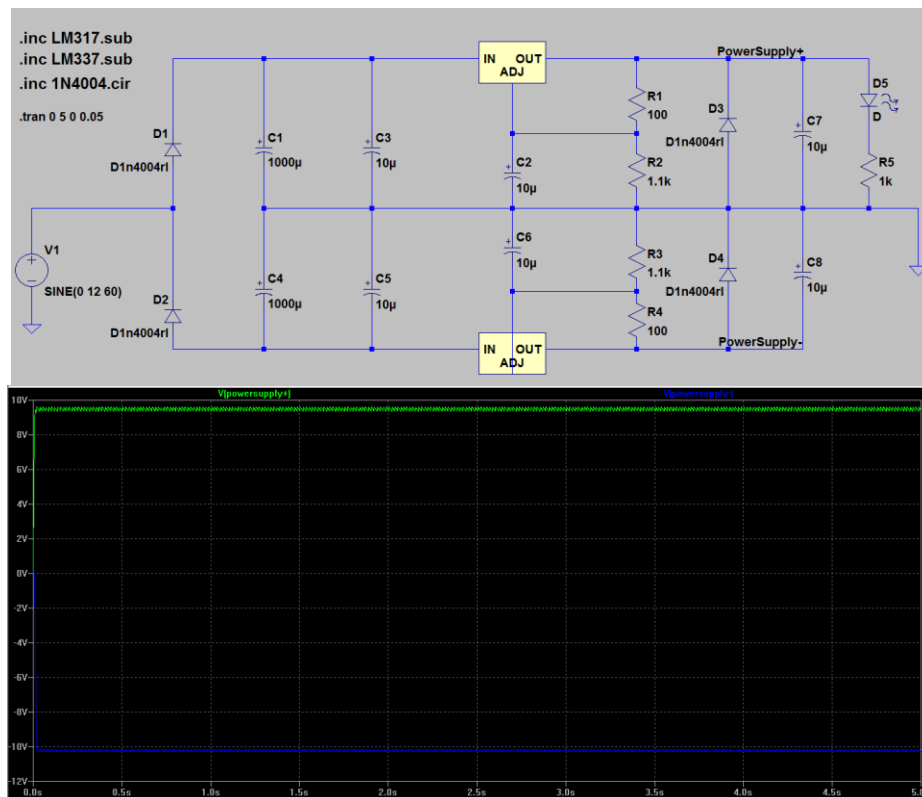


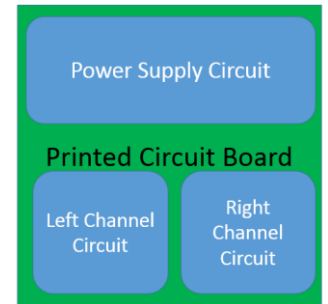
Fig.14 Power Supply Simulation

## Board Level Design

You should generate Printed-Circuit Board (PCB) files by eagle and Gerber files by Gerber View.

### Usage of Components

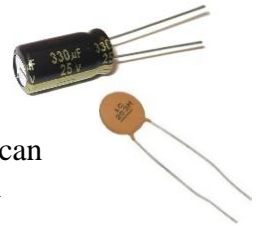
The components selection is based on through-hole soldering skill. Through-hole soldering is easiest method for new learners. The design of component placement on PCB should be **followed schematic**; that reduce copper trace routing on PCB and avoid the noise generated from the component to affect the other components nearby. The Fig on the right is the example of component placement plan.

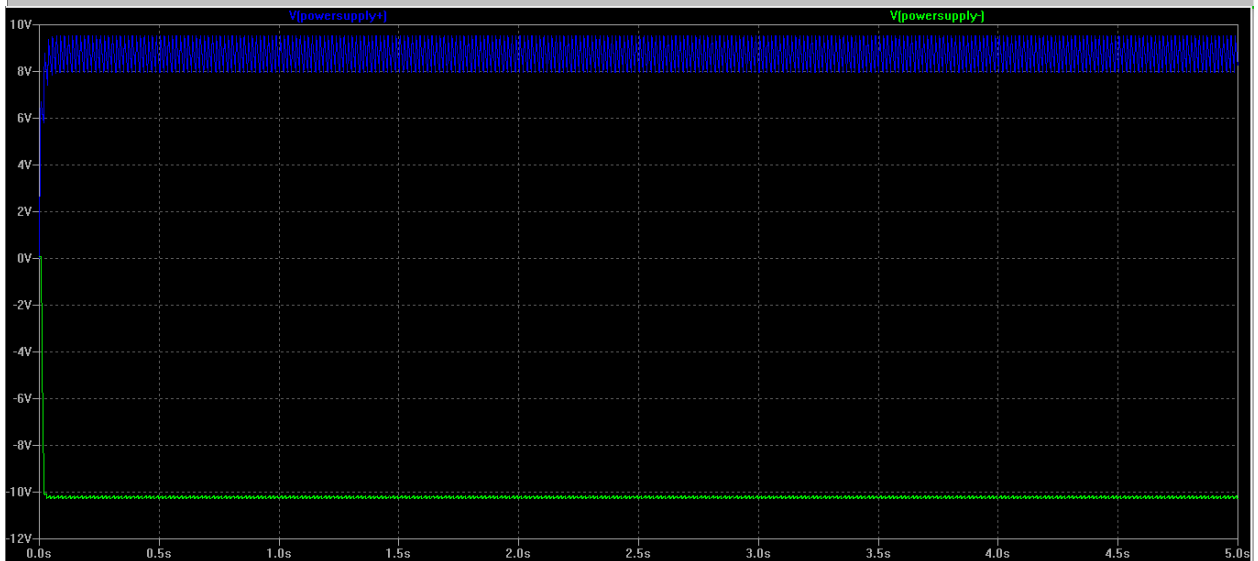
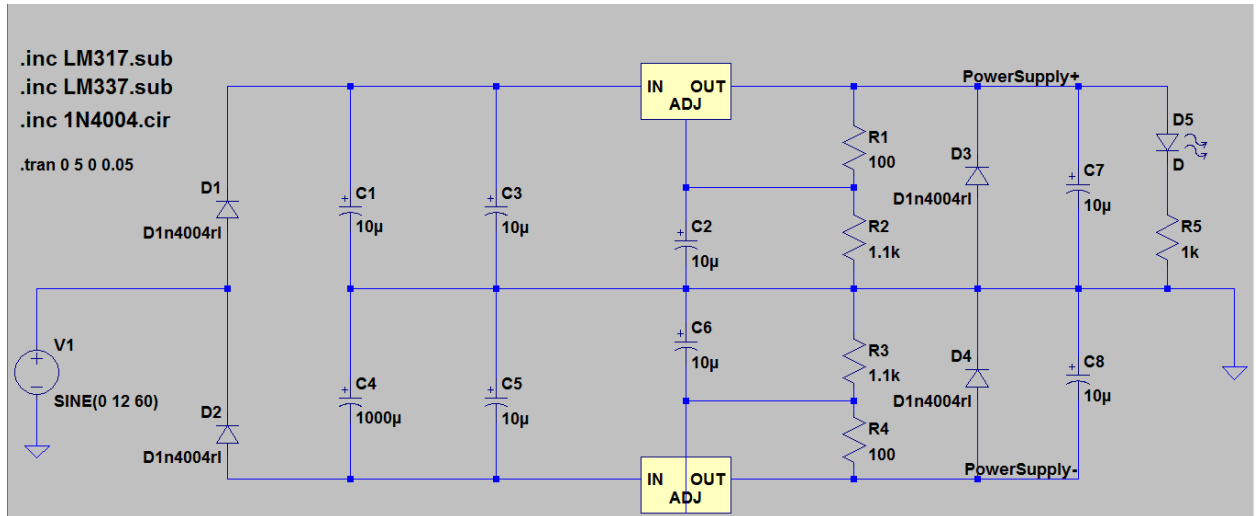


#### 1. Capacitor

There are polarity and non-polarity capacitor in our design. Please don't mess up positive and negative pin of polarity capacitors. Also there are some polarity capacitors for video device.

The capacitors in the design hold output voltage in certain value. The larger capacitance a capacitor has, the smoother the output voltage is. You can change the capacitance of C1 from 1000uF to 10uF (in LTSpice simulation template named **Power Supply**) to compare the output (PowerSupply+, PowerSupply-) and to figure out their difference.



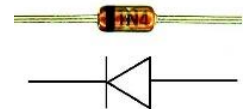


## 2. Diode

The node having a black strip is cathode.

1N4004 is used for rectifiers in power supply circuit because its reverse voltage is as high as 500V.

1N4148 is used to let input signal into amplifier without power loss because 1N4148 is connected in reverse voltage model; so input signal isn't able to flow through diode to deflect other parts of the circuit.



## 3. Resistors



4. Transistors (BJT)

Please check datasheet to confirm collector, base, emitter of a BJT. Generally, vendors supply datasheet of their product. There are also a lots of information online. Please check it if you have any questions on components.



5. Op-amp

There are several Op-amps satisfied the usage of project design. Advise to select op275 because this IC chip is cheap and save space in space-limited PCB. The chip contains two amplifiers in one chip. OP275 has large scale dual power supply (5-18V); that allow it operated properly under the power supply of the design (approximate 10V).



6. IC Socket

Advise solder IC socket on PCB instead of soldering IC chip directly to avoid to burn the chip.



7. LED

An indicator for indicating power supply working properly or not.



8. Transistor (MJE)

When you place components on PCB, it is better to place transistor near the end of board and DO NOT close to IC chip. Transistors generate a lot of heat. When you design the placement of transistor, you should consider the location of the transistors being placed not over heat the others component nearby. If necessary, each transistor be added a sink to emit heat.



9. Regulator

Supply certain value of voltage and protect source chance



10. Audio Jack 3.5mm (1/8")



This jacks are used to be connected to a headphone (output signal) and to MP3 (input signal). You can test your headphone amplifier with a cable with two headphone plug-in (male) and to connect input signal audio jack (female). After you connect output audio jack with headphone, you are able to head the sound enhanced if your PCB design and soldering work are correct.



### **Eagleware and Create Gerber file**

Please download the files name *Eagle Schematic and Layout Example* and *Create Gerber File from EAGLE* from iLearn. You can follow tutorial Eagle Schematic and Layout Example step by step to practice and be familiar with Eagle. There are also a lots of online material on the skill of Eagle usage. *Create Gerber File from EAGLE* is the tutorial to generate Gerber file. After you complete board level design on Eagle, you should generate Gerber files. These files is send to PCB manufactory to generate your unique PCB.

# Appendix

## A. Soldering Equipment



*Solder*



*Soldering Station*



*Soldering Magnifying Glass*

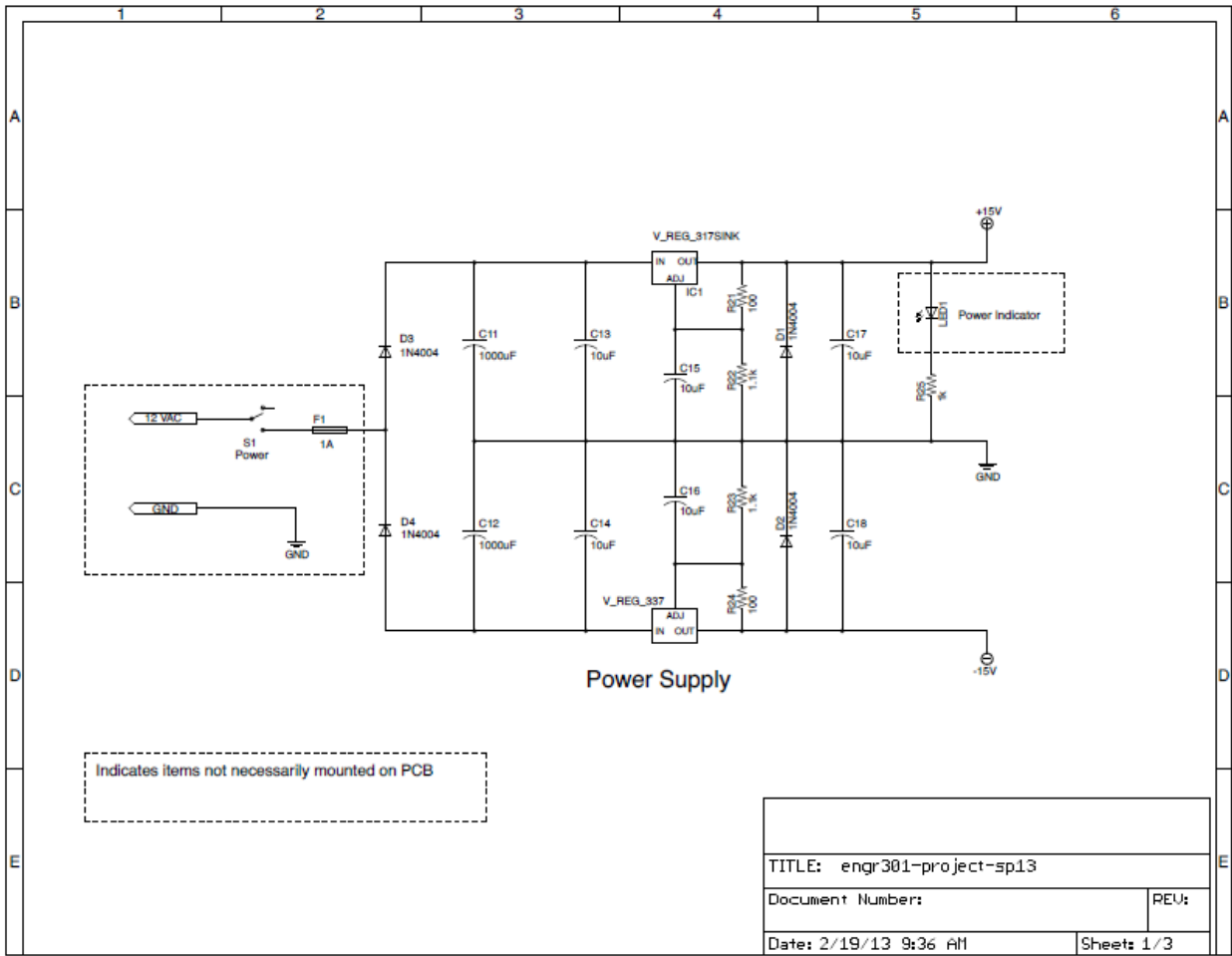


*Desoldering Pump*

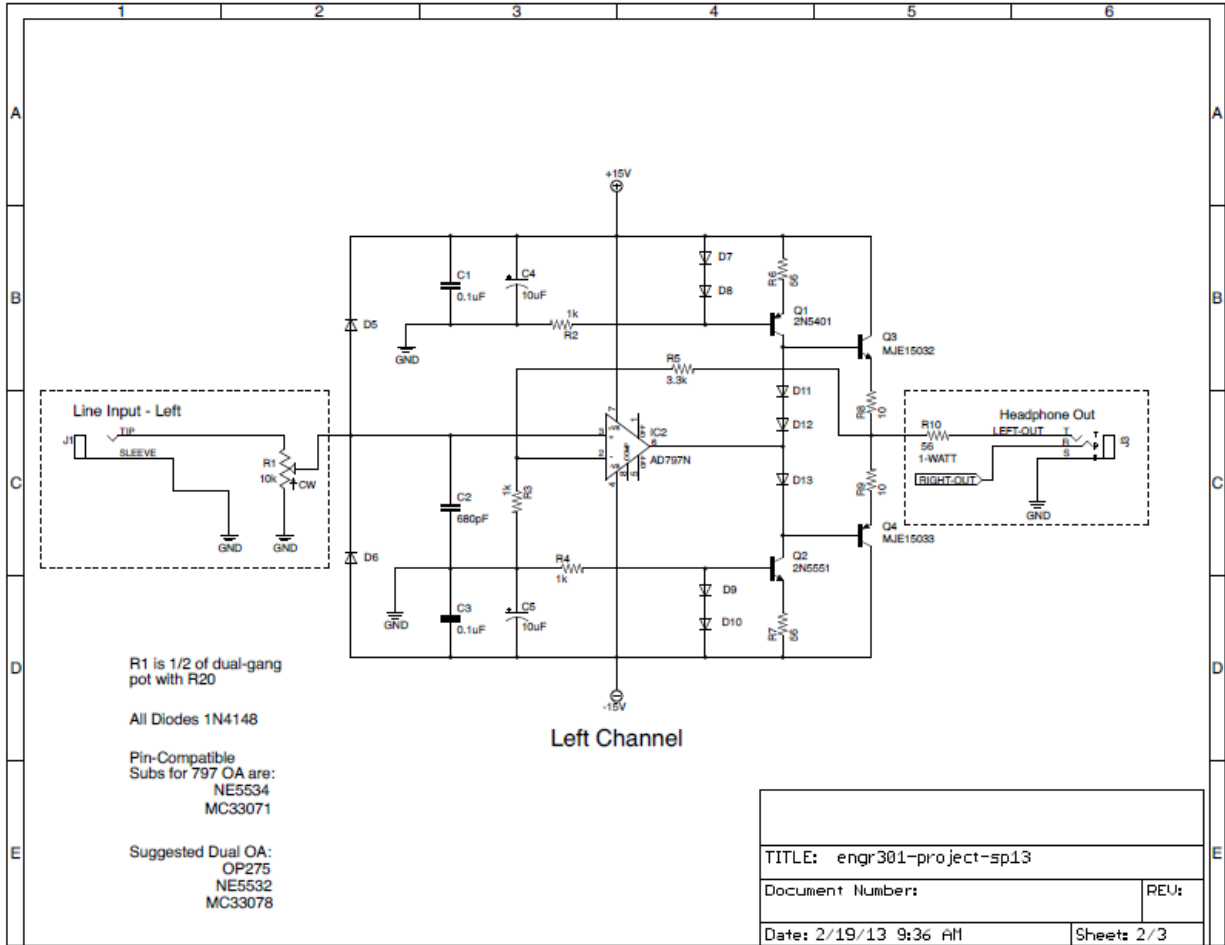


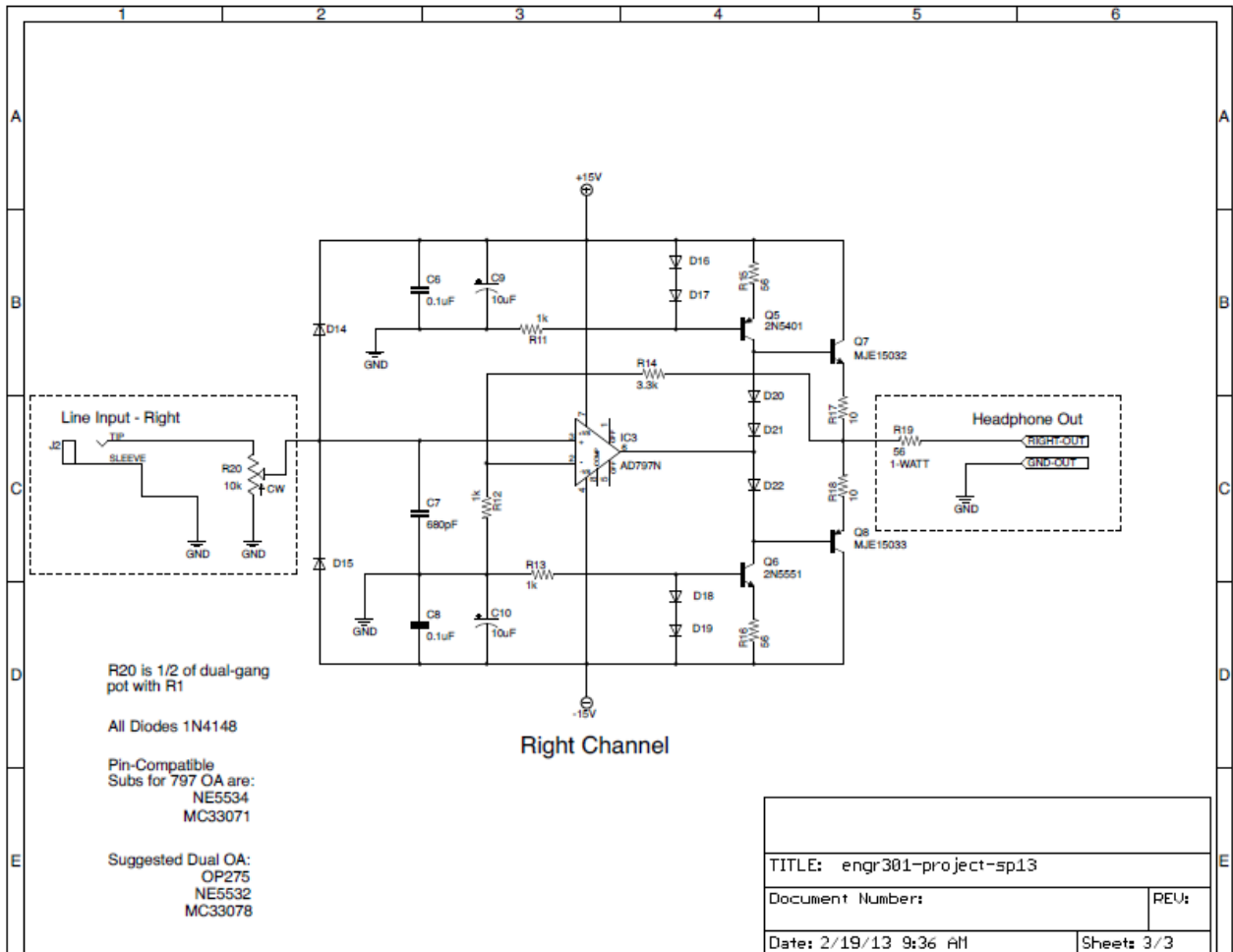
*Clipper*

## B. Circuit Design









## C. Software Downloads

Software Downloads Source

- Eagleware: <http://www.cadsoftusa.com/download-eagle/>
- LT-spice: <http://www.linear.com/designtools/software/>

Gerber Viewer:

- Mac OS X: <http://www.mcn-audio.com/sharewares/index.html>
- Window: [http://graphiccode.com/GC-Prevue\\_Gerber\\_Viewer](http://graphiccode.com/GC-Prevue_Gerber_Viewer)

Components Purchase Vendor:

- Jameco: <http://www.jameco.com/>
- Mouser: <http://www.mouser.com/>
- Digkey: <http://www.digikey.com/>

## D. Online Tutorials

Basic Soldering instruction on Solder in Youtube:

1. <https://www.youtube.com/watch?v=vIT4ra6Mo0s&index=1&list=PL926EC0F1F93C1837>
2. <https://www.youtube.com/watch?v=Mrhg5A1a1mU&index=2&list=PL926EC0F1F93C1837>
3. [https://www.youtube.com/watch?v=\\_GLcT\\_u3U8&list=PL926EC0F1F93C1837&index=3](https://www.youtube.com/watch?v=_GLcT_u3U8&list=PL926EC0F1F93C1837&index=3)
4. <https://www.youtube.com/watch?v=hvTiql-ED4A&index=4&list=PL926EC0F1F93C1837>
5. <https://www.youtube.com/watch?v=sN3V8hMiUb4&list=PL926EC0F1F93C1837&index=5>
6. <https://www.youtube.com/watch?v=AY5M-IGxvzo&index=6&list=PL926EC0F1F93C1837>
7. <https://www.youtube.com/watch?v=VgcPxdnjw4&index=7&list=PL926EC0F1F93C1837>
8. <https://www.youtube.com/watch?v=sTv3gK9tAKA&index=8&list=PL926EC0F1F93C1837>
9. <https://www.youtube.com/watch?v=Nq5ngauITsw&index=9&list=PL926EC0F1F93C1837>

LTSpice IV Waveform View

<https://www.youtube.com/watch?v=xFLD5wblX7U>

LTSpice Solution (this link is the general information of the usage of LTSpice)

<http://www.linear.com/solutions/LTspice>

Add Third-Party Models to LTSpice IV

<https://www.youtube.com/watch?v=ajcYYwoHF0g>

Noise Simulation Websites:

<http://www.linear.com/solutions/1148>