EE1025-Independent Project
DC AND AC CIRCUIT SIMULATION
BAXANDALL TONE CONTROL CIRCUIT

P.Srijith
EE19BTECH11041
Eshwari.J
EE19BTECH11042

April 8, 2020

Abstract

Simulation of simple voltage divider with DC source to explore LT - SPICE resources. AC simulation and understanding the graphical view of low-pass and band-pass filters. The gain and phase plots are depicted in this report.Baxandall Tone control circuit known for independently adjusting the bass and treble for high quality audio applicatons is also stated!

.

Contents

1 Aim

- Understanding LT-SPICE utility for DC and AC simulations.
- The low pass filter's graphical view with the help of gain and phase plots.
- Baxandall Tone control circuit it's importance ,the method of usage and the output.
- Analysing LCR circuits theoritically and verification through SPICE simulation.

2 Introduction

LT-SPICE:-
LTspice is a high performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits. Included in the download of LTspice are macromodels for a majority of Analog Devices switching regulators, amplifiers, as well as a library of devices for general circuit simulation.

DC simulation:-
DC simulation is set up by selecting transient analysis in the SPICE module over some fixed time and constant dc voltage source. Through DC simulation,voltages at any point ,currents through circuit elements,power dissipated through resistors and delivered by source and many more could be calculated.

AC simulation:-
AC simulation is set by selecting AC analysis in SPICE module over a range of frequencies and fixing the votlage amplitude and phase,or voltage of fixed frequency and amplitude over transient analysis results in AC simulation and yeilds us characteristics of output and input voltages .

Baxandall tone circuit:-
The Baxandall tone control circuit, illustrated below, which is an analogue circuit providing independent control of bass and treble frequencies; both bass and treble can be boosted or cut and with both controls at their mid positions, provides a relatively flat frequency response, as illustrated by the blue Level response graph.The original model used a valve (tube) amplifier and feedback as part of the circuit to reduce the considerable attenuation (about 20dB).

3 Appartus

These experiments are conducted on an open-source LT-spice.

4 Theoretical Background

4.1 DC simulation

- The LT-SPICE software is used.
- The analysis is set in DC mode which is transient for about 1ms.
- VOLTAGE DIVIDER :- Voltage division is used to calculate the voltage drop across a resistor (impedance) as a fraction of total voltage across a series string of resistors

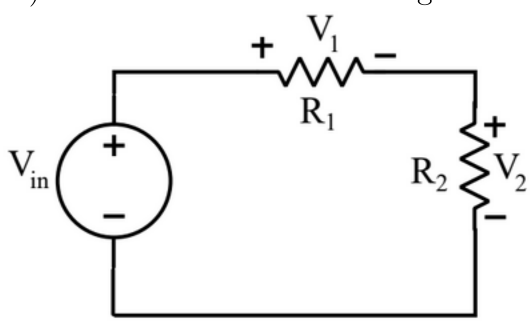


Figure 1: Voltage Divider

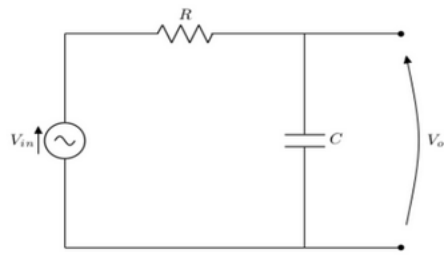
$$V_1 = V_{in}\frac{R_1}{R_1 + R_2}$$
$$V_2 = V_{in}\frac{R_2}{R_1 + R_2}$$

(impedances).

- TRANSIENT ANALYSIS OF CAPACITOR:- In an RC circuit connected to a DC voltage source, voltage on the capacitor is initially zero and rises rapidly .The drop across the capcitor is V (t) = emf (1-e$^{-t/RC}$).
The time constant for an RC circuit is defined to be RC.
The response time is $5{\times}RC$. (*which is time taken to reach* 99 *percent of the source voltage*).

4.2 AC simulation

- The LT-SPICE software is used.
- The analysis is set in AC mode which takes range of frequencies into account.
- CUT -OFF FREQUENCY:- A cutoff frequency, corner frequency, or break frequency is a boundary in a system's frequency response at which energy flowing through the system begins to be reduced rather than passing through
- LOW PASS FILTER:- These are filters which allow only the low frequency signal and



blocks the higher frequency signal.

The cutoff frequency is defined as at which $|H(\omega)|$ falls to 0.7 times of it's maximum gain. The transfer function for the above circuit is :

$$H(s) = \frac{V_o(s)}{V_{in}(s)} \tag{1}$$

Calculations for the above circuits:

For RC network:

$$V_o(s) = V_{in}(s)\frac{Z_C}{R + Z_C} \tag{2}$$

- BAND PASS FILTER:- This filter allows the band of freuencies (pass band) and block the frequency which falls outside the pass band.
- GAIN AND PHASE PLOTS:- The Gain plot depicts the ratio of output voltage to the input voltage in decibels . In other words, the gain margin is 1/g if g is the gain at the 180 phase frequency. Similarly, the phase margin is the difference between the phase of the response and 180 when the loop gain is 1.0.

Voltage gain [edit]

The power gain can be calculated using voltage instead of power using Joule's first law $P = V^2/R$; the formula is:

$$\text{gain-db} = 10\log\frac{\frac{V_\text{out}^2}{R_\text{out}}}{\frac{V_\text{in}^2}{R_\text{in}}}\ \text{dB}.$$

In many cases, the input impedance R_in and output impedance R_out are equal, so the above equation can be simplified to:

$$\text{gain-db} = 10\log\left(\frac{V_\text{out}}{V_\text{in}}\right)^2\ \text{dB},$$

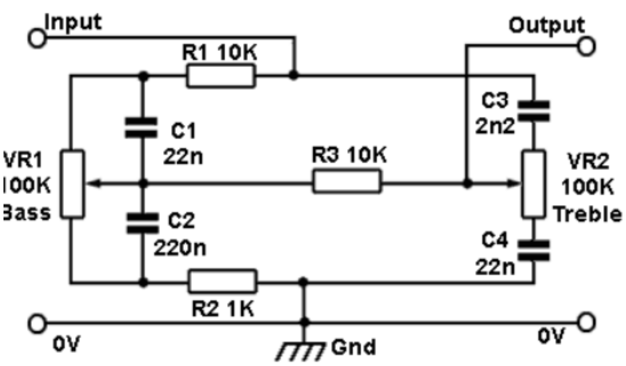
$$\text{gain-db} = 20\log\left(\frac{V_\text{out}}{V_\text{in}}\right)\ \text{dB}.$$

This simplified formula, the 20 log rule, is used to calculate a **voltage gain** in decibels and is equivalent to a power gain only if the impedances at i

4.3 Baxandall tone control circuit

- Tone Control, provides a simple means of regulating the amount of higher frequencies present in the output signal fed to the loudspeakers. A simple method of achieving this is to place a variable CR network between the voltage amplifier and the power amplifier stages.

- LT-spice is used to create potentiometers for the circuit with the help of resistors and sliding over it for about 100 divisions and selecting the linear line for the best audio response.
- This is of passive version,though bass and treble boost occur the circuit gives a picture of frequency reduction(attenuation).



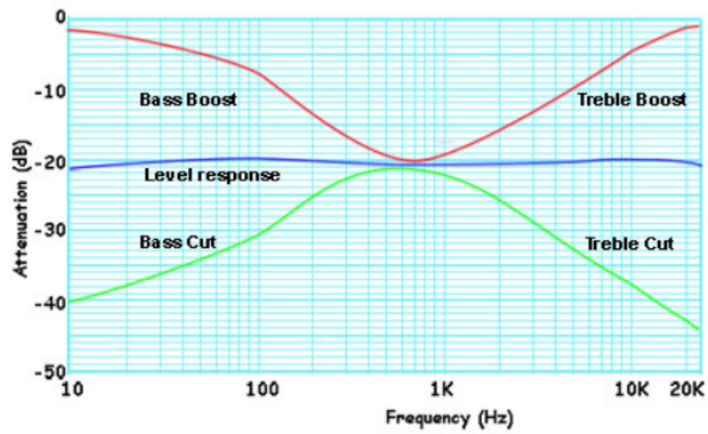


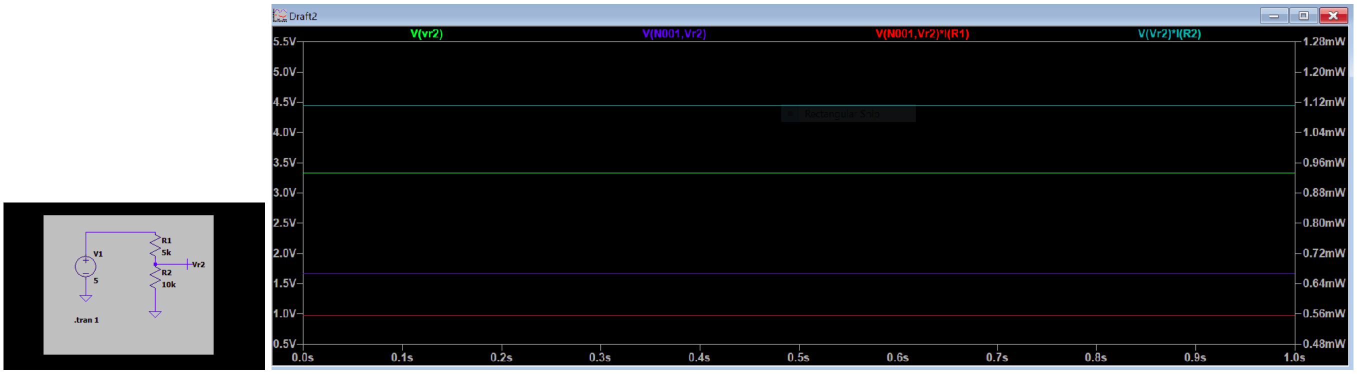
Fig. 4.2.5 The Baxandall Modified Response Curve

5 Procedures & Analysis

5.1 TASK 1 :VOLTAGE DIVIDER (DC SIMULATION)

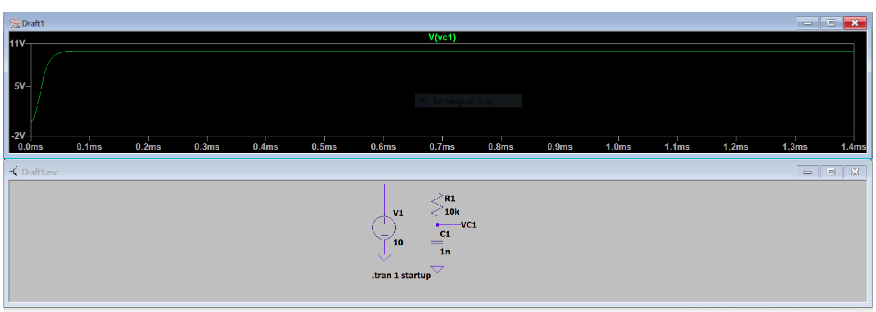
- Step 1: Selecting the transient analysis in run simulation with stop time as 1s.
- Step 2: Placing 5k ohm and 10k ohm resisotrs in series with 5V source.
- Step 3: Calculating the voltages across 5k and 10k by placing the cursor at respective points in the circuit.
- Step 4: Simulating the circuit gives us the voltage across the resistors as 1.667 V and 3.33V respectively .

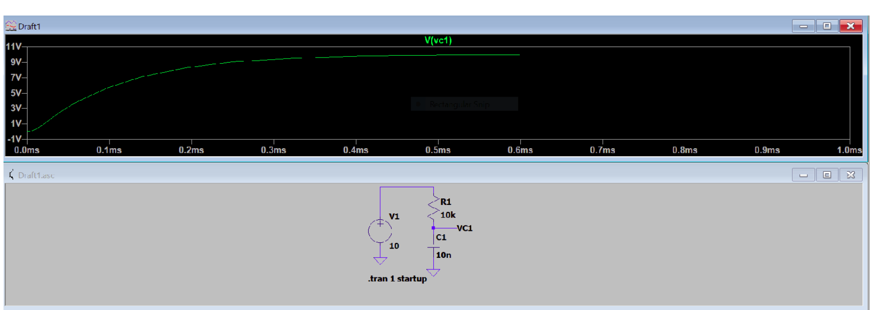
- Step 5: The power can be directly calculated by placing the cursor+Ctrl on the circuit elements.The LT-SPICE effectively calculates the power using the (voltage drop)$\times(thecurrent)$

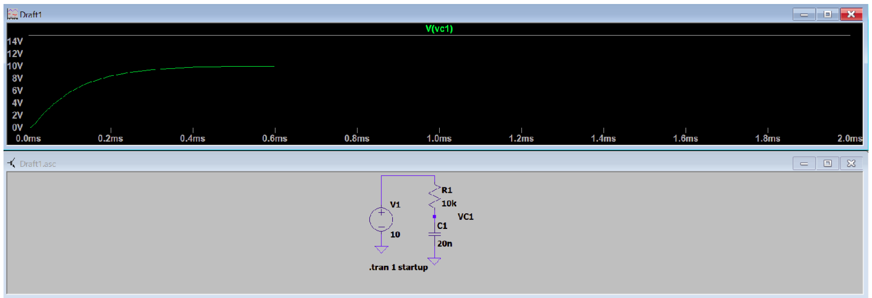


5.2 TASK 2: CAPACITOR (DC SIMULATION)

- Step 1:Selecting the transient analysis in run simulation with stop time as 10ms and start time as 0.
- Step 2:Placing the 10k ohm resistor with various capacitors with capacitances 1n,10n,20n in series with voltage source 10V.
- Step 3: Caluclating the voltage across the capacitor by placing the cursor at point before the capacitor .
- Step 4 :Simulating the circuit results in the transient voltage across the capacitor.
- Step 5 : The observation is that the voltage becomes constant after a particular value that is the response time which is 0.05ms ,0.5ms ,1ms respectively which $5 \times RC$ *is clearly depicted in the graphs below.*

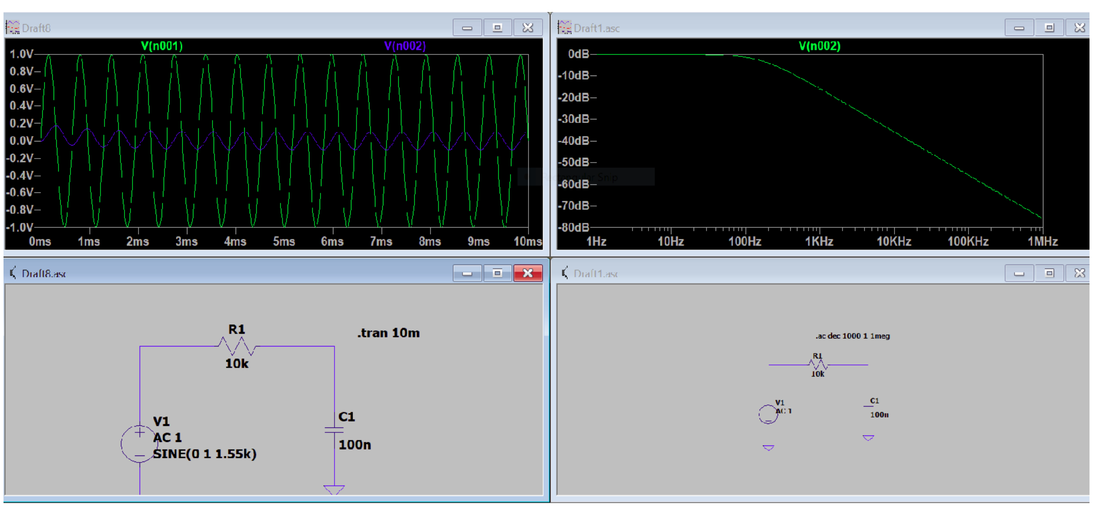






5.3 TASK 3: LOW PASS FILTER (AC SIMULATION)

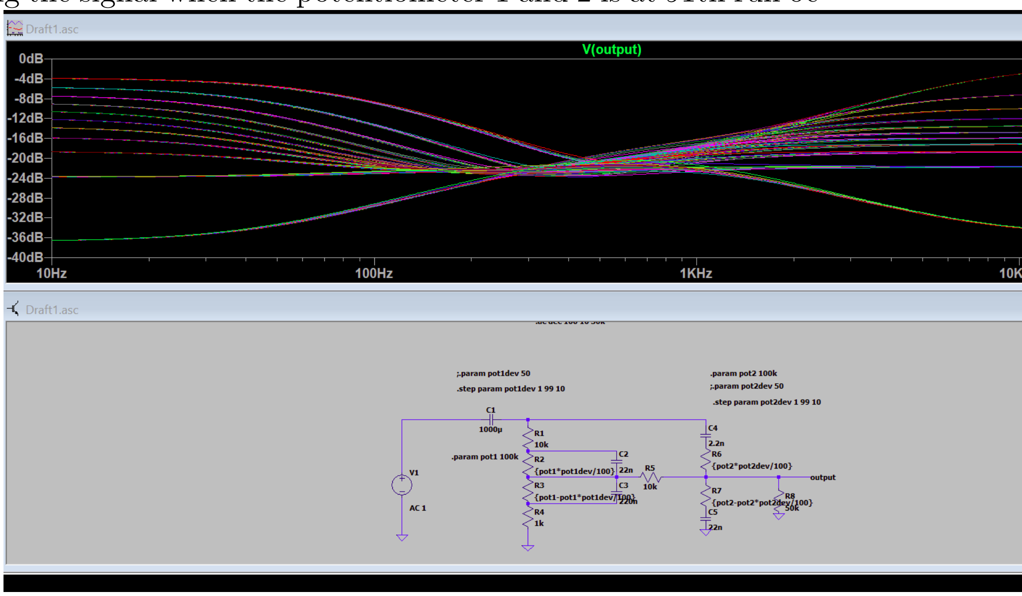
- Step 1: Selecting the AC analysis in run simulation with frequency range between 1Hz to 1MHz with step size 1000.
- Step 2:Placing the 10k ohm resistor in series with 100nC and voltage source of amplitude 1V and phase 0.
- Step 3: Caluclating the voltage across the capacitor by placing the cursor at point before the capacitor .
- Step 4 :Simulating the circuit results in the low pass filter with increase in the frequency leads to attenuation of output signal.
- Step 5: To understand the clear picture of output versus input signal ,frequency is calcuated at -20dB where the output signals amplitude is 10 times lower than input signal which is 1,55kHz.
- Step 6: Another circuit with change in run simulation as transient and the voltage as a sinusodial function with amplitude 1 and with frequency 1.55kHz yeilds output voltage amplitude 10 times lower than input voltage .



5.4 TASK 4: BAXANDALL TONE CONTROL CIRCUIT

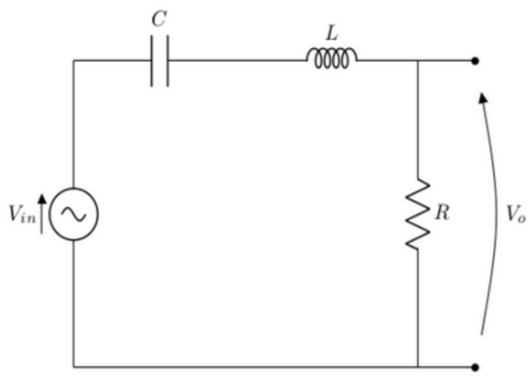
- Step 1: The circuit features two potentiometers ,with four resistors.
- Step 2: Parameter potentiometer 1,2 with resistances 100k ohm and next parameter,potentiometer deviation 1,2 starting from 1,99 with steps of 99.
- Step 3:R2=pot1$\times pot1dev/100$
  $R3 = pot1 - pot1 \times pot1dev/100. similarly R6, R7.$

- Step 4:The output voltage is plotted as shown below, depicting all possible values of gain that the circuit is producing .
- Step 5:The flat line (linear response) is observed at -20dB .
- Step 6: The filter is not affecting the signal when the potentiometer 1 and 2 is at 91th run occuring at the flat line response.



5.5 TASK 5: BAND PASS FILTER (AC SIMULATION)

- Step 1: Selecting the AC analysis in run simulation with frequency range between 1Hz to 1MHz with step size 100.
- Step 2:Placing the 10k ohm ,20k ohm in series with 10nC ,5H and voltage source of amplitude 1V and phase 0.



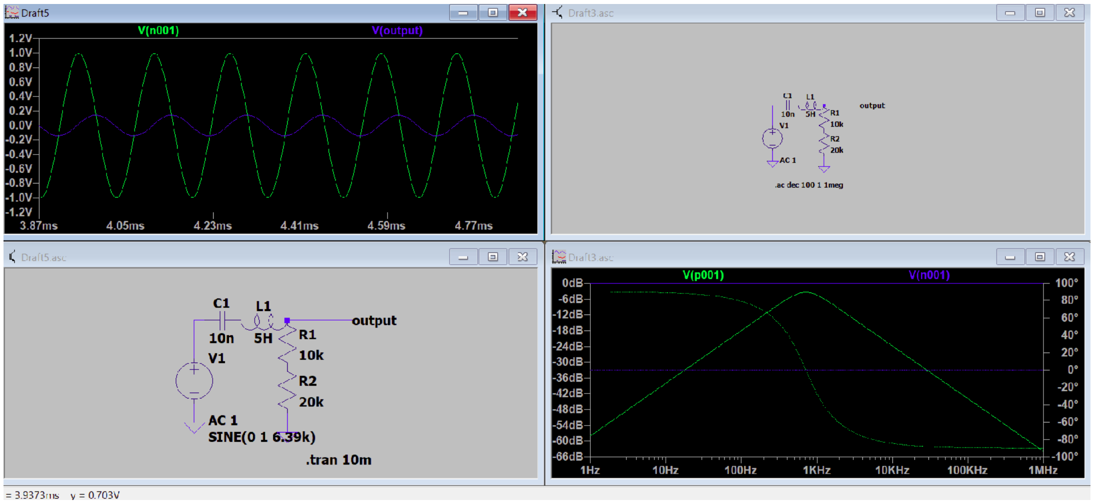
The transfer function for bandpass filter is

$$H(s) = \frac{R}{R + Z_C + Z_L}$$

where, $Z_C = \dfrac{1}{j\omega C}$ and $Z_L = j\omega L$.

The cutoff frequency for the band-pass filter is $\omega_c = \dfrac{1}{\sqrt{LC}}$

- Step 3: Caluclating the voltage across the both resistances .
- Step 4 :Simulating the circuit results in the band pass filter , for a certain range of frequencies a resonable output signal is observed at very low and high frequencies the signal attenautes.
- Step 5: To understand the clear picture of output versus input signal ,frequency is calcuated at -20dB where the output signals amplitude is 10 times lower than input signal which is 6.39kHz.
- Step 6: Another circuit with change in run simulation as transient and the voltage as a sinusodial function with amplitude 1 and with frequency 6.39Khz yeilds output voltage amplitude 10 times lower than input voltage .
- Step 7:The above steps are verified with LT-SPICE resource as shown in the below fig-

ures . 

6 Conclusions

- LT-SPICE is user-friendly module to calculate voltages,powers,understanding the behaviour of filters etc.
- All kinds of analysis can be done on LT-SPICE.
- Baxandall tone control circuit balances the treble and bass independtly using two potentiometers and output which is flat response among all responses gives the best audu=io quality.
- The Gain plots can be converted into output -input voltages at different values of frequencies and those frequencies are taken up by the voltage source.

7 References

https://learnabout-electronics.org/Amplifiers/amplifiers42.php https://en.wikipedia.org/wiki/Gain$_{(electr}$