Department of Electrical Engineering and Computer Science

Howard University
Washington, DC 20059

EECE 471 Design of Integrated Circuits

Spring 2024

Project 1
by
Dequane Nealy

Instructor: Dr. Eric Seabron

Date Assigned: 2/29/2024 Date Submitted: 3/18/2024

Analog Design Question and Steps

When designing this circuit I had to keep in mind some design considerations. Later through the design process I considered the non-idealities of the circuit, namely the resistance tolerances. With real-life components they will never function as expected in the design. Whether it be noise, it's lossy nature, or components not acting as expected, non-idealities are to be expected. Out of the 3 smaller circuits comprising this circuit, an amplifier, a low pass filter, and a high pass filter (**Fig 1.1**), I started with the amplifier. The gain had to be greater than 3 to account for the other transfer functions likely not being perfect. With this in might, the resistors 3 and 4 were set to 10 ohms and 30 ohms, respectively. As the gain is $1 + R_4/ R_3$ this sets the amplifier for a transfer function resulting in 4, just above the required value. Next the focus shifted to the filters. For this design some research was first conducted to find optimal frequencies for the respective filters. During this search I came to find low pass filters have frequencies that will pass through around 500 Hertz while high pass filters pass around 2000 Hertz or 2 kHz. With this boundary in mind, I worked backwards using the frequencies I found as a starting point. The low pass filter frequency had to be less than or equal to 500 Hertz. The high pass filter had to be greater than or equal to 2000 Hertz. I considered these frequencies as the cutoff frequency in the equation, $RC = 1/(2 * \pi * f_c$. Here this related the cutoff frequency to the values of the resistor and the capacitor. From here, I generated plausible values for each capacitor and found the resulting resistor value. With the capacitors, I utilized pico-farads or 10^e-6 farads as that seemed to be a standard when working with capacitors. The capacitors had to relate to each other, making the upcoming sensitivity analysis much easier to conduct. Capacitor 1 would be a factor of Capacitor 2, allowing N to be the multiple separating them. With these capacitor values, I then found the limits for the resistors. The low pass filter's resistor or resistor 2 had to be at least 8 ohms. The high pass filter's resistor or resistor 1 had to be no greater than 8 ohms. The constraints kept my values consistent even when I wanted to check varying capacitor values. Anytime my capacitor values, each equation I went through before then became pointless. Over this process I was constantly rechecking my values to keep everything to the standards I set. For the high pass filter (resistor and capacitor 1) the values chosen were 10 pico-farads and 5 ohms. From the equation $R_1/(Z_{C1}+R_1)$, this resulted in a transfer function of 0.5. This frequency response was shown using a Bode Plot based on the design (**Fig 2.1**). For the low pass filter (resistor and capacitor 2) the values chosen were 40 pico-farads and 30 ohms. From the equation $Z_{C2}/(R_2 + Z_{C2})$, this resulted in a transfer function of 0.49. As capacitor 2 was based on capacitor 1, the multiple N was 4. This frequency response was shown using a Bode Plot based on the design (**Fig 2.2**). The cascading transfer function, the gain of the entire circuit, was the three phases multiplied against each other. The final transfer function was 0.98, just slightly under 1. This shows the signal should be very clear going through the circuit, with little noise distorting or disrupting it. The actual transfer function was just one of the figures of merit in the design, the others being bandwidth and center frequency. With my design completed and mathematically tested, I moved onto my Python programming and sensitivity analysis.

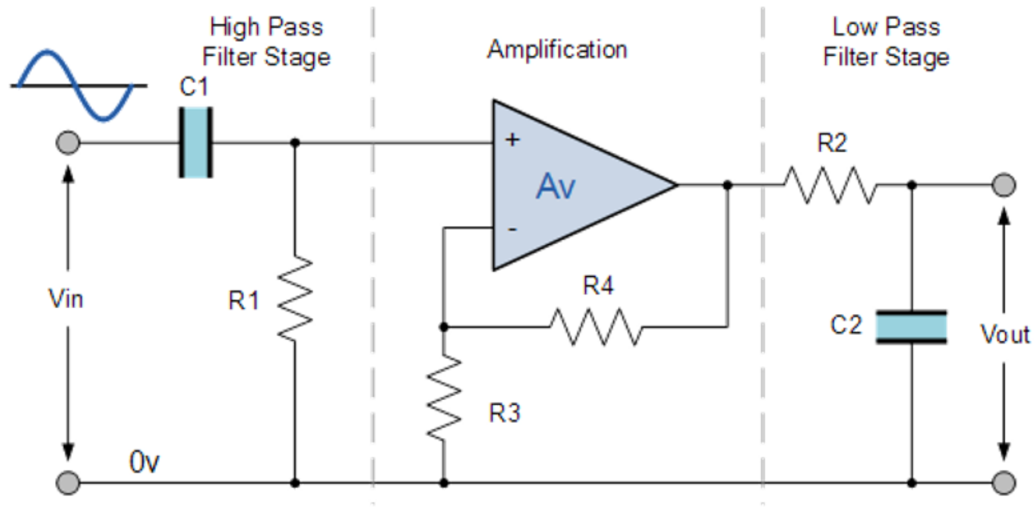

Fig 1.1 – Circuit Layout

Bode Plots

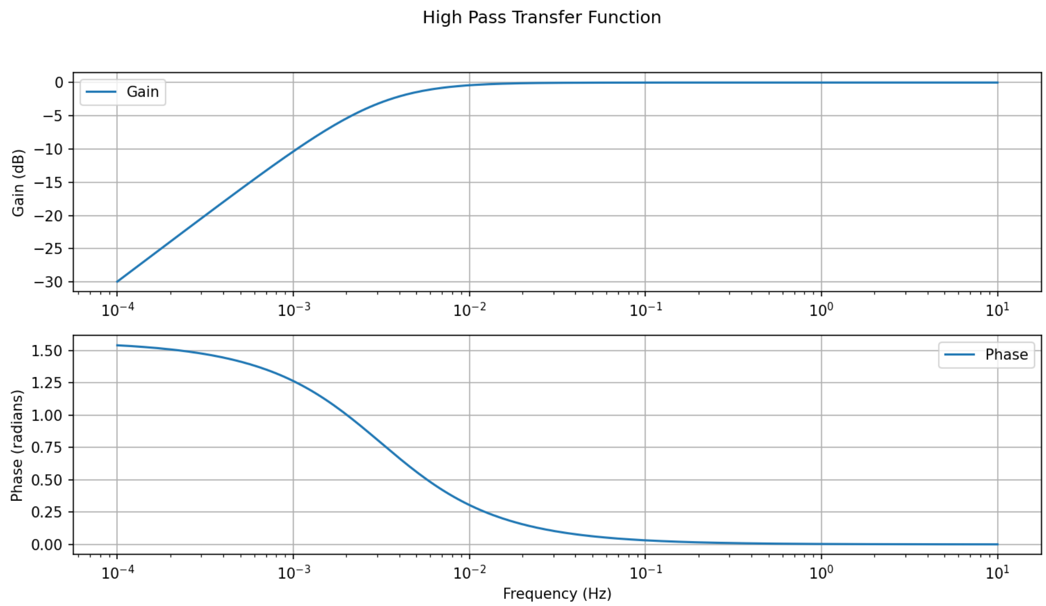

Fig 2.1 – High Pass Filter Response

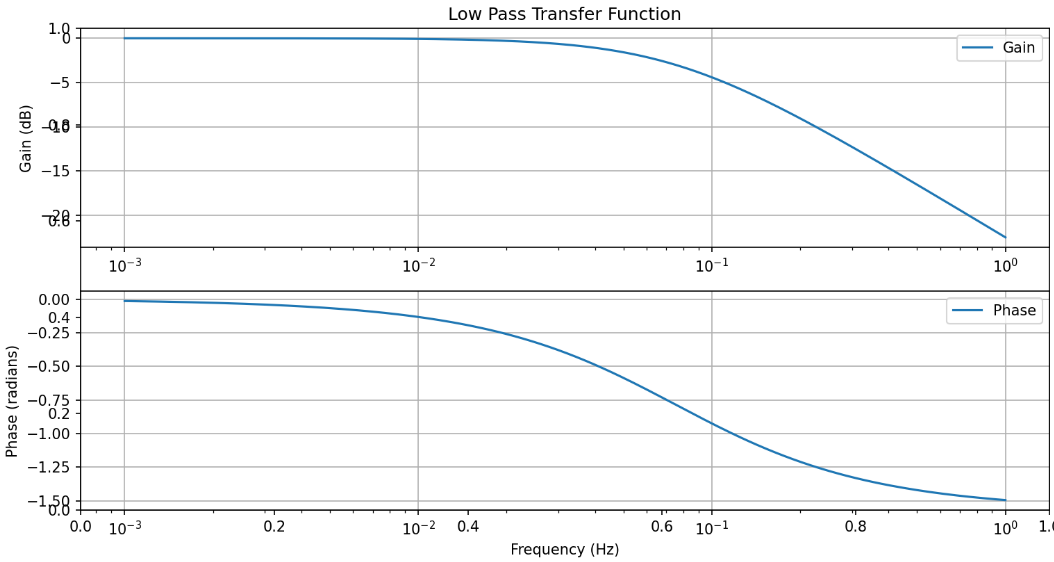

Fig 2.2 – Low Pass Filter Response

Analog Design Sensitivity Analysis

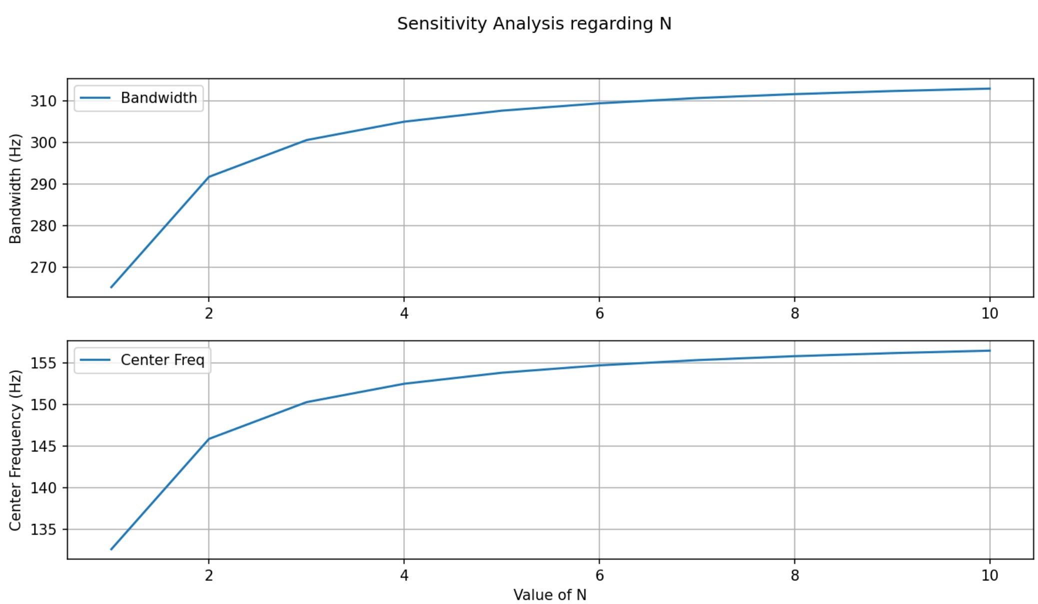

Fig 3.1 – N Sensitivity Analysis

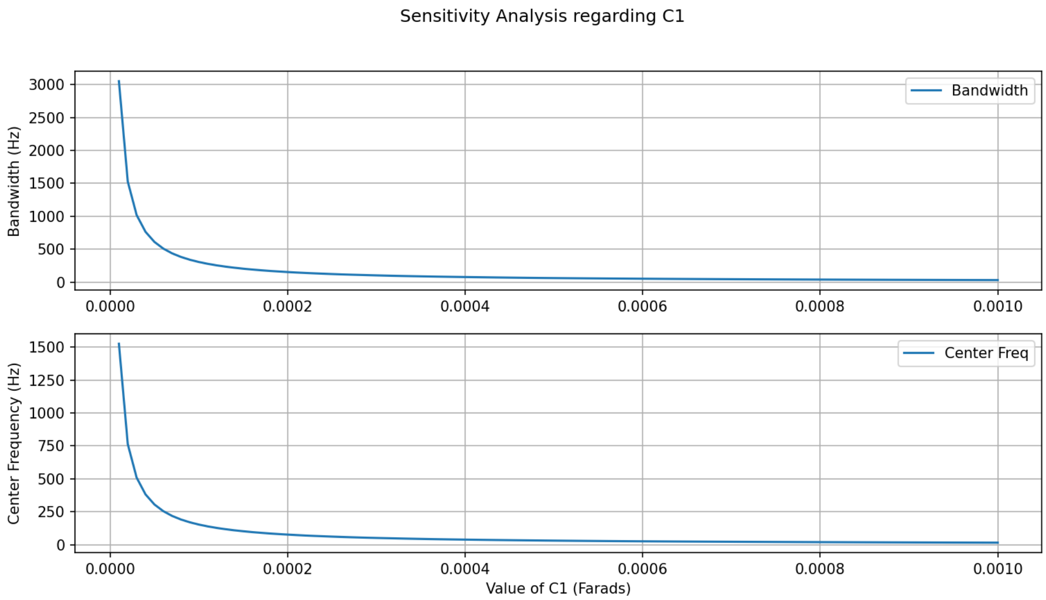

Fig 3.2 – C1 Sensitivity Analysis

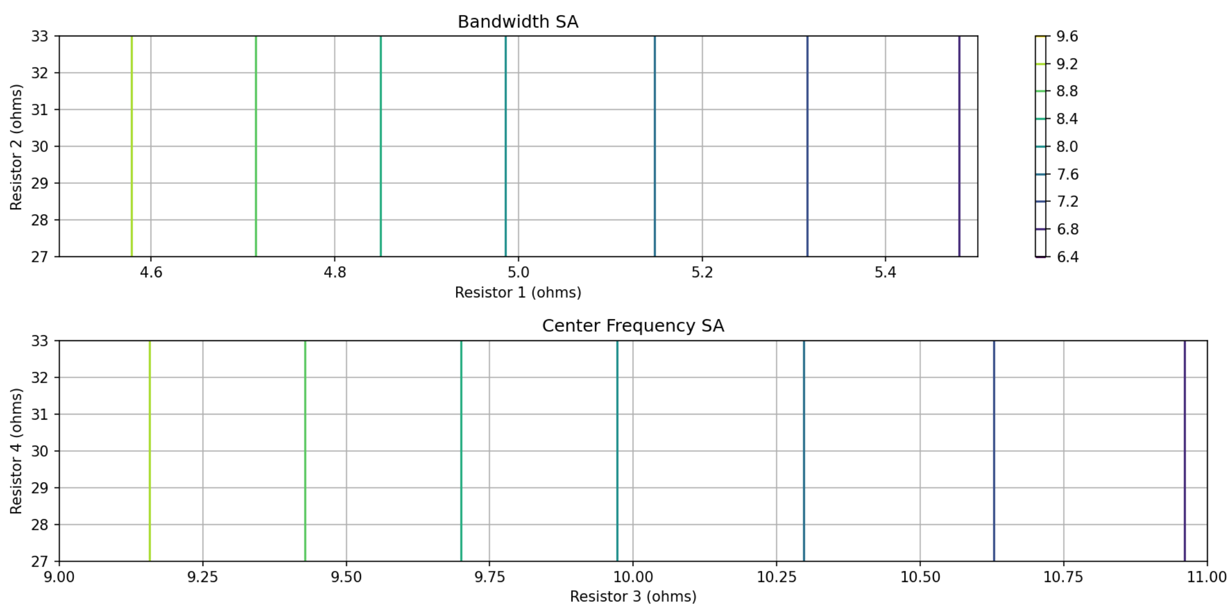

Fig 3.3 – Resistor Sensitivity Analysis

Analog Design SA Discussion

The sensitivity analysis for a circuit judges the effect or performance a specific parameter has on a quantitative metric. The metrics to be considered here are the actual transfer function, the bandwidth, and the center frequency. These are called the Figures of Merit. For the circuit there were 3 sensitivity analyses to be performed. The first analysis is over the effect N has on the figures of merit (**Fig 3.1)**. I found the larger N is, the more the bandwidth and center frequency would increase. This relation was steady until N=8, shortly coming to a flat band around that time. After N surpasses 10, that value loses its importance. Even though N outwardly seems to play a large role in the success of the circuit, the sensitivity analysis argues differently, with the very minimal change shown in the plot. The next analysis is how much the first capacitor values affect the performance of the circuit (**Fig 3.2**). The capacitor value had an inverse effect. The higher the capacitor, the worse the circuit began to perform. Very quickly both the bandwidth and the center frequency drop to 0 on the plot. Where the higher N drops the performance or the circuit, it would seem the smaller the Capacitor 1 value is the same will occur. How large it can be before it degrades was not measured but with the asymptote appearing the closer C1 is to 0, the first capacitor needs to be smaller. The final analysis was the effect each resistor has on the circuit (**Fig 3.3**). When it comes to the resistors, I was facing trouble with the contour plot. I tried to judge the effect of all the resistors at once, not giving myself the easiest time to figure out the impact on each other. I tried evaluating all four resistors within one plot, but it was deemed impossible by my Python interpreter. From there, I wanted to look at the impact of 2 resistors, slowly comparing each in that fashion. That method, while functional, still yielding no results during my investigation. It is likely an error with my setup that could have been overlooked, but an answer could not be found. Initially I utilized resistors 3 and 4 for my evaluation, as they were only used together for the amplifier. As those were the only resistors intertwined, they did yield much in terms of response. Each line plotted was a vertical line with a small difference between each notch. The same came to be true with the other resistors. I felt stumped but as resistors are constant, assuming ideal circumstances, there would be no need for the resistor to vary besides the tolerances I applied. In my comparison to of resistors to the figures of merit, I ran into more and more errors. Plotting my resistors seemed to be the only option as I could not uncover this final mistake. Outside of this, the effect of each tested parameter was very clear, with the plots serving as a visual aid for the work conducted.