Active Filter Design

Cristhian Quezada Riofrio

Cristhian.michael24@gmail.com

ABSTRACT

This report provides an exhaustive revision of the different filters that are used in engineering. Using these filters provides the foundation for some applications, such as signal processing, communication, and other applications in electronics. This report aims to explain by experimentation how filters work in real life by carrying out different tests. The result can be compared with simulations to understand the difference between the theoretical and practical results.

INTRODUCTION

Filters are important in modern technology, and understanding how they work in real life is key to implementing them in real applications that require a good understanding of the theory, calculations, and practical design. This report documents the implementation of an active filter called a band-pass filter, which uses two cuts of frequencies to allow only signals contained in between those two frequencies to pass. It was possible to design an active band-pass filter using passive components such as resistors and capacitors with a combination of active components. Calculations and simulations are provided in this report to demonstrate the difference between the values obtained from them and the values obtained from the labs. This paper also comments on the practical implications of band-pass filters.

ACTIVE FILTER LAB

THEORY

Filters are circuits designed to allow input components at specific ranges to pass to the output and prevent input components at different ranges from reaching the output. A common application of these filters is separating a signal of interest from other signals or noise [1] . There are two important characteristics in a filter: the passband that allows certain frequencies to pass and the stopband that blocks certain frequencies, as shown in Fig.(1). Ideally, filters will block signals depending on the cutoff frequency, but ideal filters are not physically realizable, however, it is possible to design filters as close to the ideal ones, by increasing the order of the filter, which means that more reactive components have to be used making the circuit more complex [2], they can also be classified as passive filters or active filters.

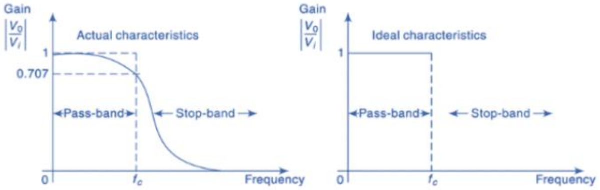


Figure 1: Ideal and real filter.

Passive Filter

Passive filters, as shown in Fig.(2) are characterized by the absence of amplification or gain, which means that passive filters are attenuators. Another characteristic of passive filters is that they cannot be cascaded because a loading effect will appear to solve these inconveniences an operational amplifier *(Op-Amp)* can be implemented in the circuit, however, the complexity of the circuit, the cost, and the number of components will increase depending on the accuracy of the filter that is desired.

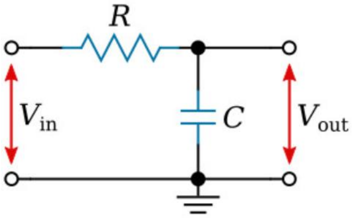


Figure 2: Passive filter configuration.

Active Filter

Active filters, as shown in Fig.(3), are characterized by the implementation of amplification components such as Op-Amps, which means that this type of filter can amplify the output signal, obtaining a gain. The complexity of these circuits will increase depending on the order of the filter; a higher-order requires more components, but it produces a more accurate filter, on the other hand, a first-order system requires only one reactive component, but the frequency response will not be as accurate compare with high-order filters.

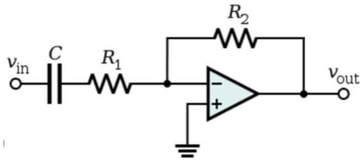
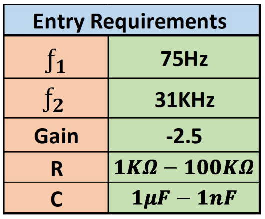


Figure 3: Active filter configuration.

CALCULATIONS

This lab aimed to design an active band pass filter with specific characteristics, as shown in Fig.(4). These initial data were used to obtain the value of the components to obtain the cutoff frequencies required (f_1, f_2).



Entry Requirements	
f_1	75Hz
f_2	31KHz
Gain	-2.5
R	$1K\Omega - 100K\Omega$
C	$1\mu F - 1nF$

Figure 4: Entry requirements.

As shown in Fig.(5), the active bandpass filter combines a low-pass filter and a high-pass filter.

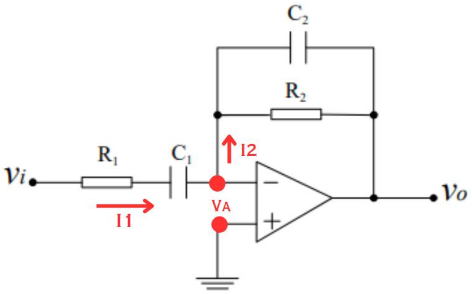
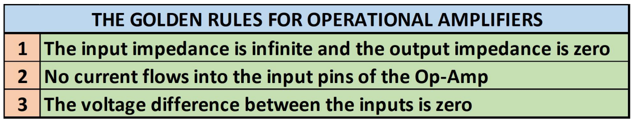


Figure 5: Active Band-Pass filter configuration.

Before starting the calculation for the band-pass filter, it is important to consider some rules that can help to simplify the calculations, as shown in Fig.(6).



THE GOLDEN RULES FOR OPERATIONAL AMPLIFIERS	
1	The input impedance is infinite and the output impedance is zero
2	No current flows into the input pins of the Op-Amp
3	The voltage difference between the inputs is zero

Figure 6: Ideal Op-Amp rules.

Considering the rules commented on above, it was possible to obtain the first equations by using Kirchoff's Current Law (*KCL*) to obtain the equation for the currents, as shown in Eq.(1), the voltage in the positive terminal of the Op-Amp is the same voltage that the one in the negative input of the Op-Amp,

$$\boxed{\begin{gathered} I_1 = I_2 \\ V_A = 0V \end{gathered}}$$

Equation 1: First considerations.

It is important to understand that this circuit is formed of two impedances. Each impedance incorporates one reactive component, in this case, this reactive component is a capacitor, as shown in Fig.(7). The first impedance (Z_1) consists of a resistor connected in series with a capacitor, the second impedance (Z_2) consists of a resistor and capacitor connected in parallel.

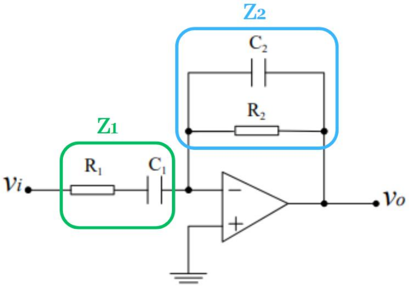


Figure 7: Input and feedback impedances.

Once the two impedances have been identified, it was possible to obtain their equations as shown in Eq.(2). The reactive component can be treated as a resister.

$$\boxed{\begin{gathered} Z_1 = R_1 + \frac{1}{SC_1} \\ Z_2 = \frac{R_2}{1 + SC_2R_2} \end{gathered}}$$

Equation 2: Impedance values.

Using the equations obtained by applying KCL, it was possible to convert it in terms of voltages and impedances as shown in Eq.(3).

$$\boxed{\begin{gathered} I_1 = I_2 \\ \frac{V_{in} - V_A}{Z_1} = \frac{V_A - V_{out}}{Z_2} \end{gathered}}$$

Equation 3: Current equations.

Once the value of the impedances and voltages have been substituted in the previous equation, the transfer function shown in Eq.(4) was obtained.

$$\boxed{\begin{gathered} \frac{V_{out}}{V_{in}} = \frac{-R_2C_1S}{(1 + R_1C_1S)(1 + R_2C_2S)} \\ \frac{V_{out}}{V_{in}} = \frac{-Kj\omega}{(1 + \frac{j\omega}{\omega_1})(1 + \frac{j\omega}{\omega_2})} \end{gathered}}$$

Equation 4: Transfer function.

From the transfer function, the two cutoff frequencies were obtained as shown in Eq.(5). Using those equations, it is possible to obtain the value of the resistors and capacitors required.

$$f_1 = \frac{1}{2\pi R_1 C_1}$$
$$f_2 = \frac{1}{2\pi R_2 C_2}$$

Equation 5: Cut-off frequencies.

The filter's gain is determined by two resistors, as shown in Eq.(6). Using this equation and modifying the value of the components, it is possible to obtain different gains.

$$k = \frac{-R_2}{R_1}$$
$$-2.55 = \frac{-5.6K}{2.2K}$$

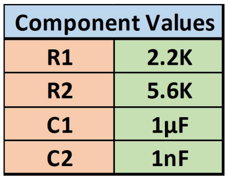
Equation 6:Gain of the filter.

Replacing the value of the capacitor in the equations of the cutoff frequency, the value of the resistors was obtained. However, it is impossible to obtain exact values for each component because they are not available in the market, so components with similar values were used, as shown in the table. (1).

Resistor Obtained	Resistor Used
$R_1 = 2122\Omega$	$R_1 = 2.2K\Omega$
$R_2 = 5134\Omega$	$R_1 = 5.6K\Omega$

Table 1: Value of resistors.

Now that all the values for each component have been obtained, as shown in Fig.(8), it is possible to recreate the filter using a simulator and then recreate the filter using actual components and see how they behave at different frequencies.



Component Values	
R1	2.2K
R2	5.6K
C1	1µF
C2	1nF

Figure 8: Components selected.

SIMULATIONS

Before recreating the circuit using actual components, it is important to simulate the circuit in a safe environment, and this can be achieved by using simulator software such as Proteus to simulate the signals in the oscilloscope, as shown in Fig.(9)

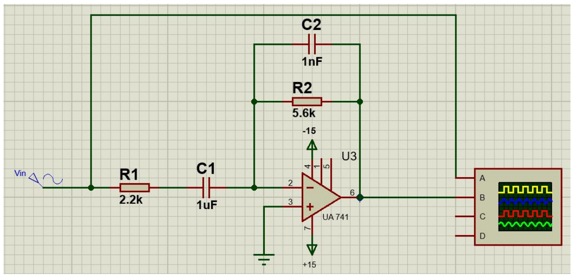


Figure 9: Proteus simulations of the bandpass filter.

Recreating the bandpass filter in proteus and using an oscilloscope to simulate the output signal was used, as shown in Fig.(10). Using a sinusoidal input signal of 1V peak-to-peak, it is possible to observe the conductance of the filter when a frequency of 10Hz that is below the first cutoff frequency of 75Hz is applied, the bandpass filter almost rejects the signal of 10Hz.

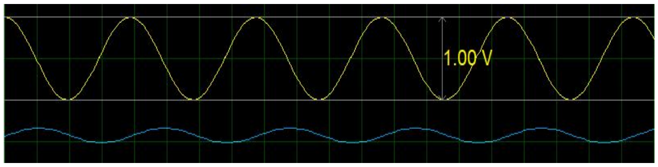


Figure 10: Representation of the input signal in Yellow and the output signal in Blue at 10Hz.

The filter's gain was obtained using proteus, as shown in Fig.(11). The gain obtained is the same from Eq.(6), which verifies that the components used are suitable for a bandpass filter with the selected characteristics.

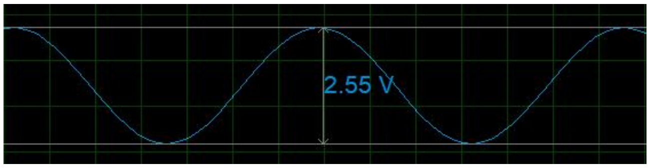


Figure 11: Gain in the midband

With the gain obtained, it is possible to predict the amplitude for the cutoff frequency, as shown in Eq.(7)

$$f_1 = \frac{2.55}{\sqrt{2}} = 1.8$$
$$f_2 = \frac{2.55}{\sqrt{2}} = 1.8$$

Equation 7: Amplitudes at the cutoff frequency.

Simulating the output signal with an input of 75Hz is expected to obtain an amplitude of 1.8V according to previously obtained values. By using Proteus, the amplitude is quite close, obtaining a value of 1.83V as shown in Fig.(12). The reason for this can be the components used, according to Table.(1), the values obtained are not the values used for this simulation, simply because there are not resistors with those specific values, so resistors with similar values were used, another reason could be the human error at the moment of measure the amplitude of the signal.

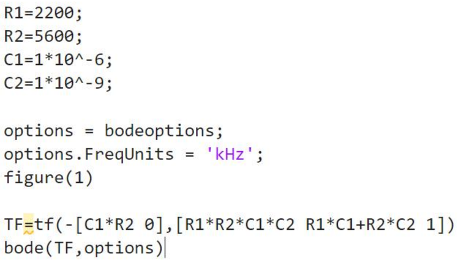
Figure 12: Output signal simulation at 75Hz.

Using the amplitude obtained from the calculations, it is possible to obtain the gain in decibels (dB), as shown in Eq.(8).

$$\boxed{\begin{aligned} K &= 20\log(2.5) = 7.96dB \\ f_1 &= 20\log(1.8) = 5.10dB \\ f_2 &= 20\log(1.8) = 5.10dB \end{aligned}}$$

Equation 8: Gain value in dB of the amplitude of the Gain, the lower cutoff frequency, and the higher cutoff frequency.

Using MATLAB, it is possible to obtain the bode plot regarding the gain in dB and the frequency. The bode plot of the bandpass filter was obtained using the code as shown in Fig.(13).



```
R1=2200;
R2=5600;
C1=1*10^-6;
C2=1*10^-9;

options = bodeoptions;
options.FreqUnits = 'kHz';
figure(1)

TF=tf(-[C1*R2 0],[R1*R2*C1*C2 R1*C1+R2*C2 1])
bode(TF,options)
```

Figure 13: MATLAB Code for the transfer function.

Using the bode plot, it was possible to verify that the gain obtained in Eq.(8) is quite close to the ones obtained from the graph, as shown in Fig.(14), the lower cutoff frequency is around 75Hz, and the higher cutoff frequency is almost 31KHz.

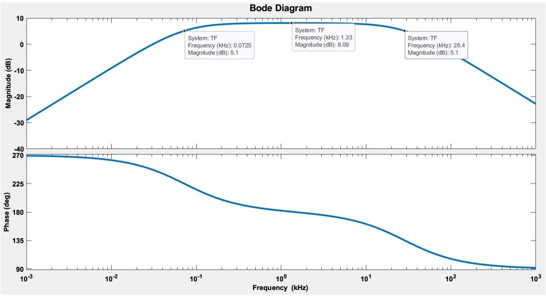


Figure 14: MATLAB Bode plot.

PROCEDURE

Components and Devices.

Once all the simulations and calculations have been done, recreating the circuit using real components is secure.

A combination of resistors and capacitors have been used for this project, as shown in Table.(2).

Resistor Name	*Manufactures value*	*Measured Value*
R1	**2.2kΩ**	**2.19kΩ**
R2	**5.6kΩ**	**5.55kΩ**
RL	**4.7kΩ**	**4.66kΩ**

Table 2: Value of the resistors.

One of the most important components used in this project is the UA741 operational amplifier, shown in Fig (15). It is easier to understand the connections that must be done using its pin configurations. The Op-Amp configuration is inverted, meaning that the input signal has to be connected to pin 2, and pin 3 has to be connected to the ground.

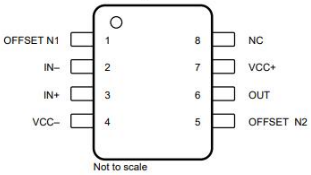


Figure 15: UA471 Op-Amp configuration pins.

Pins number 4 and 7 need a constant voltage of 15V. This voltage will be supplied using a power supply device, as shown in Fig.(16).

Figure 16: Voltage supply device.

The input signal in pin 2 was generated using a signal generator, and pin number 6 of the Op-Amp was connected to the oscilloscope, as shown in Fig.(17).

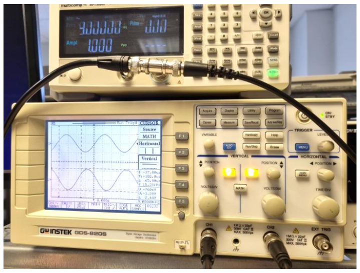


Figure 17: Equipment used to produce the input signal and display the output signal.

Using these two devices, it was possible to change the frequency of the input signal provided to the Op-Amp's input pin by adjusting the signal generator's frequency. The oscilloscope was connected to the signal generator directly by using a coaxial cable. The reason for this connection is to display the signal generator's signal on the oscilloscope and use it as a reference signal. As commented before, pin number 2 of the Op-Amp is connected directly to the signal generator. Using a T-junction, channel 1 of the oscilloscope and pin 2 of the Op-Amp are connected to the signal generator receiving the same input frequency, and channel 2 of the oscilloscope is connected to pin 6 of the Op-Amp.

Using the initial circuit diagram for an active bandpass filter, the circuit was recreated using the capacitors and the resistor obtained from the previous calculations, as shown in Fig.(18). It is possible to observe the connections with one capacitor connected in series with a resistor connected to pin 2 of the Op-Amp, from that node another capacitor and resistor are connected in parallel to pin 6 of the Op-Amp.

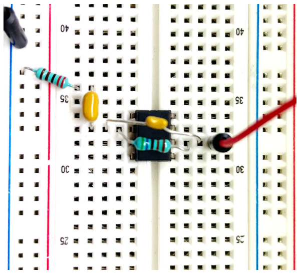


Figure 18: Bandpass filter using actual components.

The signal generator shown in Fig.(19) was used to generate sinusoidal frequencies from 0 to 80KHz with an input voltage of 1V peak-to-peak.

Figure 19: Signal generator.

Two tests were carried out to study the behaviour of the filter. The first test was carried out without a load resistor, and the second test was carried out by including a load resistor of 4.7KΩ, connecting one pin to pin 6 of the Op-Amp and the other pin connected to the ground, as shown in Fig.(20).

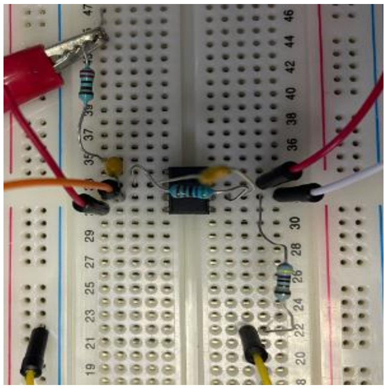


Figure 20: Bandpass filter with a load resistor.

RESULTS

Bandpass filter without a load resistor.

Using different frequencies from 0Hz to 85KHz, it was possible to obtain the Amplitude obtained at each frequency and store it in a table (see Appendix 1). The data collected was represented in a bode plot showing the amplitude in dB per frequency, as shown in Fig.(21).

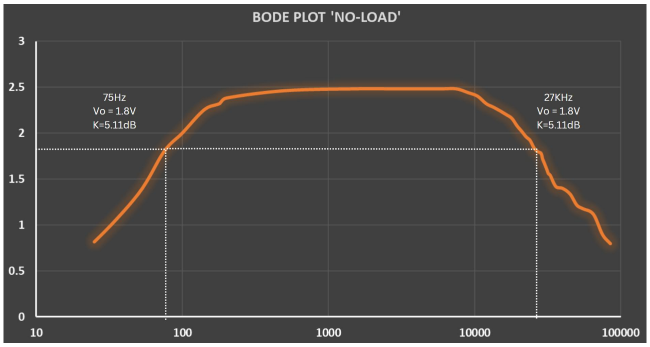


Figure 21: Bode Plot with no Load.

This graph shows the bandpass filter and its characteristics, for example, using the values obtained from Eq.(8), where the amplitude gain obtained for the cutoff frequency was 5.10dB, using this value in the graph obtained, a lower cutoff frequency of 75Hz was obtained and high cutoff frequency of 27KHz was obtained, these values are quite close to the expected values for the cutoff frequencies, a possible reason of having a significant difference in the high cutoff frequency is because from the calculations the value of R_2 obtained was 5.1KΩ, but the measured value of the component used was 5.55KΩ this increment in the resistance produce a smaller cutoff frequency, in order to obtain a more accurate value, more than one resistor can be used in series or parallel to achieve the value of 5.1KΩ. From the graph, a midband gain of 2.48V or 7.89dB was obtained. The expected gain was 2.5V this is a small difference that has been caused by the value of R_1 and R_2, the values used to obtain the midband gain were the manufacturing values of the resistors, however, the measured values are slightly different, producing a different but similar gain.

Bandpass filter with a load resistor.

A new test was carried out, including a load resistor with a value of 4.7KΩ. The bode plot obtained is quite similar to the bode plot obtained previously, as shown in Fig.(22). The cutoff frequencies of the filter are the same or similar, the load resistor is connected in parallel with the oscilloscope, and that is one of the reasons to obtain a similar bode plot because the voltage out from the Op-Amp is still the same.

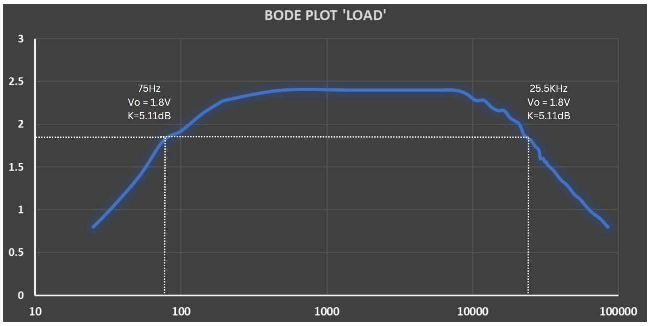
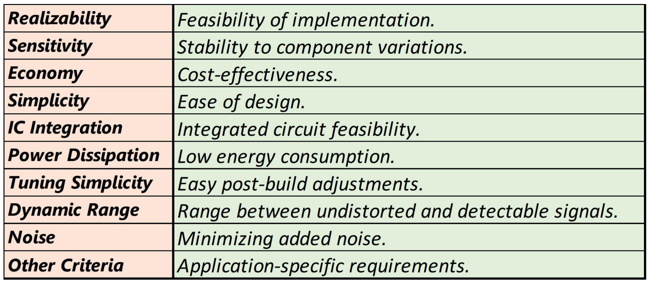


Figure 22: Bode Plot with Load

RESEARCH

ACTIVE FILTERS IN TELECOMMUNICATIONS

Active filters are essential in some industries, for example, in telecommunication systems, higher-order filters are required, in order to achieve such high-order filters [3] , a combination of active components such as Op-Amps with passive components such as resistors or capacitors, however, design these filters can be complex and it is important to have some considerations before starting the design as shown in Fig.(23).



Realizability	*Feasibility of implementation.*
Sensitivity	*Stability to component variations.*
Economy	*Cost-effectiveness.*
Simplicity	*Ease of design.*
IC Integration	*Integrated circuit feasibility.*
Power Dissipation	*Low energy consumption.*
Tuning Simplicity	*Easy post-build adjustments.*
Dynamic Range	*Range between undistorted and detectable signals.*
Noise	*Minimizing added noise.*
Other Criteria	*Application-specific requirements.*

Figure 23: Higher-Order filter criteria.

In the latest years many innovations have occurred in the field of filters. Some of these advances included the development of dual and triple-mode waveguide filters with arbitrary amplitude and phase response, dielectric resonator filters, contiguous and non-contiguous multiplexing networks, surface acoustic wave (SAW) filters, high-temperature superconductor (HTS) filters, and a variety of coaxial, finline and microstrip filters [4].

In communication systems, is important to transmit and attenuate in a specific frequency band and at the same time with the minimum distortion and loss of energy of the transmitted signal. The design and development of high-performance and small-size filters on chips operating in GHz frequency are required for wireless communication systems. Radiofrequency (RF) filters operate from 100MHz to 10GHz, if the operation range is higher than 10GHz, the filter can be catalogued as a microwave filter [5].

When transmitting data in telecommunications, it is important to identify the five different stages that the information must pass until it reaches its destination, as shown in Fig.(24). The first stage is the information source this can be a voice signal that incorporates a large number of individual signals to produce the sound of the voice signal. This signal is transmitted in the transmitter stage, where the input signal is combined with a higher intermediate carrier frequency (IF). The next stage for this signal would be the communication channel it refers to the medium used to transmit the signal, in this case, it is the free space.

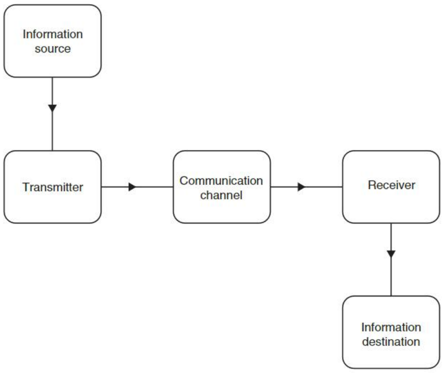
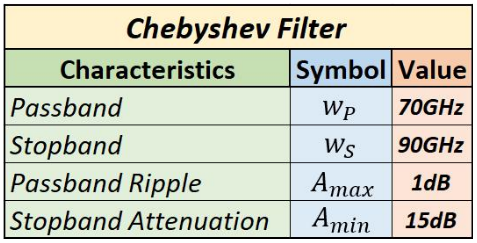


Figure 24: Stages for communication transmission.

In the receiver stage, the antenna receives the signal, and using filters, the original signal can be obtained after a process called demodulation. Without implementing wireless communication filters, obtaining the RF carrier's input signal would be complicated.

60 GHz BANDPASS FILTER DESIGN

In order to design a bandpass filter with a 60GHz centre frequency, the first step is to design a low-pass filter with the characteristics shown in Fig.(24).



Chebyshev Filter		
Characteristics	**Symbol**	**Value**
Passband	w_P	***70GHz***
Stopband	w_S	***90GHz***
Passband Ripple	A_{max}	***1dB***
Stopband Attenuation	A_{min}	***15dB***

Figure 25: Filter characteristics

The type of filter selected for this task is a Chebyshev filter with a passband ripple of 1dB and a stopband attenuation of 15 dB, as shown in Fig (26).

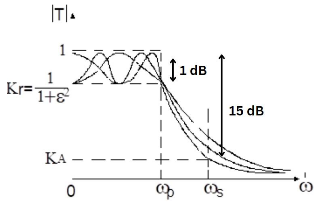


Figure 26: Chebyshev Low-Pass filter

Two important parameters need to be obtained, and these parameters can be obtained using the equations shown in Eq.(9).

$$K = \frac{1}{1+\varepsilon^2}$$

$$\varepsilon = \sqrt{10^{\frac{A}{10}} - 1}$$

Equation 9: K parameters for Chebyshev filters.

The first value obtained is epsilon, as shown in Eq.(10). There are two values for epsilon.

$$\varepsilon_R = \sqrt{10^{\frac{1}{10}} - 1}\ ;\ \varepsilon_R = 0.509$$

$$\varepsilon_A = \sqrt{10^{\frac{15}{10}} - 1}\ ;\ \varepsilon_A = 5.534$$

Equation 10: Epsilon component for each K parameter.

Each epsilon will be used to obtain the K values, as shown in Eq.(11).

$$K_R = \frac{1}{1+0.509^2}\ ;\ K_R = 0.794$$

$$K_A = \frac{1}{1+5.534^2}\ ;\ K_A = 0.032$$

Equation 11: K parameter values

Once the values for the K parameters have been obtained, the next step is to calculate the selectivity factor, as shown in Eq.(12).

$$\Omega = \frac{w_s}{w_p}$$

$$\Omega = \frac{90GHz}{70GHz}\ ;\ \Omega = 1.286$$

Equation 12: Selectivity factor.

Once the K values and the selectivity factor have been found, the next required parameter is represented with an M, as shown in Eq.(13).

$$M = \sqrt{\frac{K_A^{-1} - 1}{K_R^{-1} - 1}}\ ;\ M = \sqrt{\frac{0.032^{-1} - 1}{0.794^{-1} - 1}}$$

$$M = 10.798$$

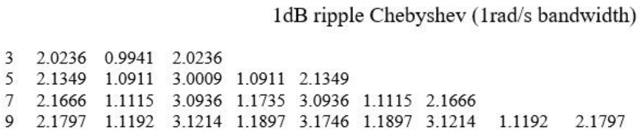
Equation 13: M parameter value.

Using the selectivity factor and the M value, as shown in Eq.(14), it is possible to obtain the order of the filter that will be used to design the low-pass filter.

$$N_c = \frac{\ln\left(M + \sqrt{M^2 - 1}\right)}{\ln\left(\Omega + \sqrt{\Omega^2 - 1}\right)}\ ;\ N_c = 4.15$$

Equation 14: Order of the filter.

The value obtained must be a whole number, so for this filter, the value obtained was 4.15, which means that the order of the filter will be rounded to 5. Using the values from Fig.(27), it is possible to obtain the values that are going to be used to obtain the Inductors and capacitors used in the circuit.



1dB ripple Chebyshev (1rad/s bandwidth)

3	2.0236	0.9941	2.0236						
5	2.1349	1.0911	3.0009	1.0911	2.1349				
7	2.1666	1.1115	3.0936	1.1735	3.0936	1.1115	2.1666		
9	2.1797	1.1192	3.1214	1.1897	3.1746	1.1897	3.1214	1.1192	2.1797

Figure 27: Chebyshev table for 1dB ripple

The values obtained from the table made it possible to obtain the values of the inductors and capacitors used for the low-pass filter using the formulas shown in Eq.(15).

$$L = \frac{L_T * R_L}{w_p}$$

$$C = \frac{C_T}{w_p * R_L}$$

Equation 15: Inductors and Capacitor formulas.

The component values obtained in the circuit diagram used for a 5th-order low-pass filter are shown in Fig.(28).

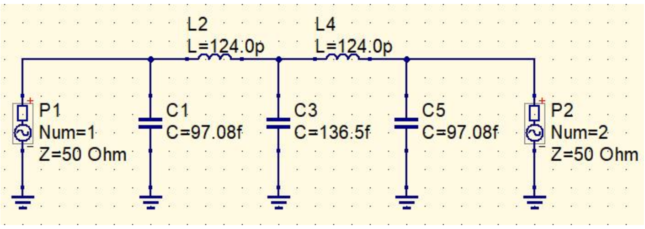


Figure 28: Low-Pass filter circuit diagram.

Once the low-pass filter has been simulated, as shown in Fig.(29), it is possible to verify that the stopband and passband correspond to the values selected at the beginning of the design.

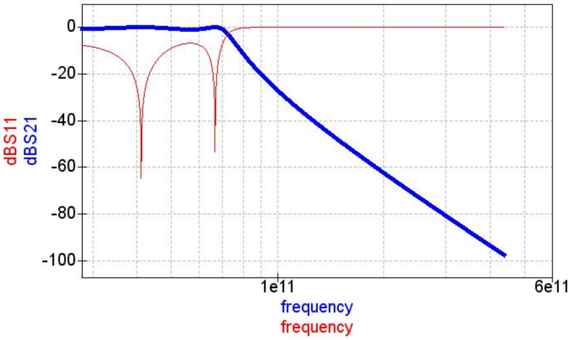
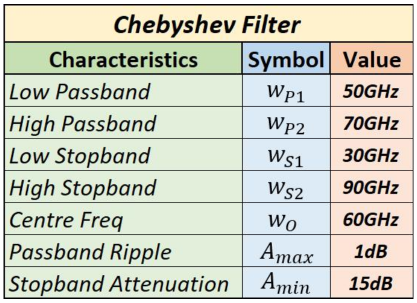


Figure 29: Low-Pass filter simulation

From the low-pass filter circuit diagram, it is possible to achieve a bandpass filter, but first, it is necessary to select some initial conditions that are desired to achieve with this filter, as shown in Fig.(30) the bandpass filter that is going to design has a centre frequency of 60GHz.



Chebyshev Filter		
Characteristics	Symbol	Value
Low Passband	w_{P1}	50GHz
High Passband	w_{P2}	70GHz
Low Stopband	w_{S1}	30GHz
High Stopband	w_{S2}	90GHz
Centre Freq	w_O	60GHz
Passband Ripple	A_{max}	1dB
Stopband Attenuation	A_{min}	15dB

Figure 30: Filter Characteristics

It is possible to obtain the centre frequency using the low passband and the high passband, as shown in Eq.(16).

$$w_0 = \sqrt{w_{P1} * w_{P2}}$$

Equation 16: Centre Frequency

The fractional bandwidth can be calculated using the centre frequency and low and high passband, as shown in Eq.(17). [6]

$$\Delta = \frac{w_2 - w_1}{w_0}$$

$$\Delta = \frac{70GHz - 50GHz}{60GHz} \; ; \Delta = 0.333$$

Equation 17: Fractional Bandwidth

Using the centre of frequency and the fractional bandwidth, the values for the capacitor and inductor can be obtained, but before obtaining these values, it is necessary to convert the low pass filter into a bandpass filter, as shown in Fig.(31)

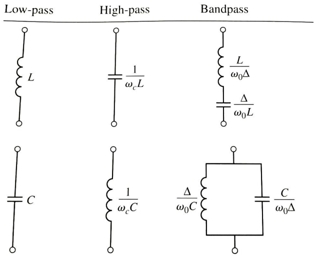


Figure 31: Low-Pass filter to Bandpass Filter transformation.

When the inductor and capacitor are connected in parallel, it is possible to obtain their values by using the centre frequency and the fractional bandwidth, as shown in Eq.(18)

$$L = \frac{\Delta * R_s}{w_0 * C_T}$$

$$C = \frac{C_T}{w_0 * R_s * \Delta}$$

Equation 18: Components connected in parallel.

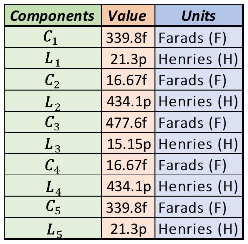
When the components are connected in series, their values can be obtained, as shown in Eq.(19).

$$L = \frac{L_T * R_s}{w_0 * \Delta}$$

$$C = \frac{\Delta}{w_0 * R_s * C_T}$$

Equation 19: Components connected in series.

The following component values have been obtained using previously used formulas, as shown in Fig.(32).



Components	Value	Units
C_1	339.8f	Farads (F)
L_1	21.3p	Henries (H)
C_2	16.67f	Farads (F)
L_2	434.1p	Henries (H)
C_3	477.6f	Farads (F)
L_3	15.15p	Henries (H)
C_4	16.67f	Farads (F)
L_4	434.1p	Henries (H)
C_5	339.8f	Farads (F)
L_5	21.3p	Henries (H)

Figure 32: Component values

Using these components, it was possible to build the circuit diagram for the bandpass filter, as shown in Fig.(33)

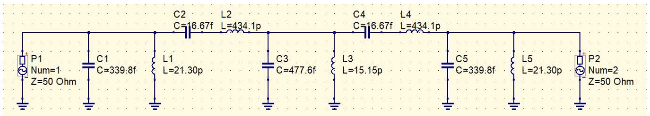


Figure 33: Bandpass filter circuit diagram.

Using a simulator called QUCS, the bandpass filter with a centre frequency of 60GHz was simulated, obtaining the bandpass filter shown in Fig.(34)

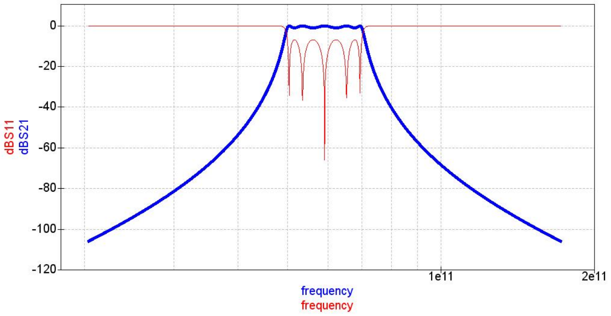


Figure 34: Bandpass filter simulation.

Even though the simulation shows a bandpass filter with a centre frequency of 60GHz, this filter is not physically possible to achieve using the component values obtained from the previous equations because there are no capacitors or inductors available in the market with those values, so a better method to use and achieved the centre frequency required is using a microstrip filter.

ULTRA-LOW-POWER ELECTRONICS

Ultra-low power electronics refer to electronic devices and circuits that operate with minimal energy consumption. This advantage is key in wearable technology, such as wearable devices for health applications, including heart rate, blood oxygen level and many others [7].

The importance of designing new electronic devices that consume less energy is a huge challenge that engineers are facing. The use of batteries can have disadvantages, such as the size of the batteries used to power electronic devices or the lifespan. In collaboration with other institutions, Cambridge University is designing new methods to implement in the circuits, such as using ambipolar semiconductor materials from their test, so the printed electronics meet the real power and voltage required for real-world applications. According to their research, they see it as a possibility to implement and develop this technology to power electronic devices for their entire lifetime, reducing the need to charge the devices continuously [8].

There are a few important concepts that are important to know when dealing with low-power electronics the first one is power efficiency. This concept is focused on the power consumption during the active mode and when it is in standby mode. The second concept is energy harvesting, which emphasizes on energy collection, such as the integration of solar cells or thermal energy. The third concept is the subthreshold operation, which involves using transistors in the subthreshold region to achieve ultra-low power consumption. Finally, the last concept is the energy-aware design that is focused on the performance and power consumption of the device that has been designed.

Low-power electronics can be considered as a new technology, but there is still a long journey in order to produce low-power electronic devices that can be used with the same efficiency as standard batteries. The challenges in this sector increase because the technology scaling of traditional methods are reaching its limits. The need to develop new materials and technologies is crucial to innovate in new methods to reduce power consumption,

CONCLUSION

In conclusion, this report includes the design of one of the most common types of filters used in the industry. By the implementation of simulation, tests, and calculations, it was possible to demonstrate how a bandpass filter works and how this type of filter can be used at high frequencies by using different methods, one of the most common applications commented in this report was how the filters are used in telecommunications.

REFERENCES

[1] A. R. Hambley, Electrical Engineering, Principles and Applications, New Jersey: Pearson Education, 2011.

[2] M. Nahvi, Electric Circuits, Fourth ed., McGraw-Hill, 2002.

[3] Y. Sun, Continuous Time Active Filter Design, London: CRC Press, 1999.

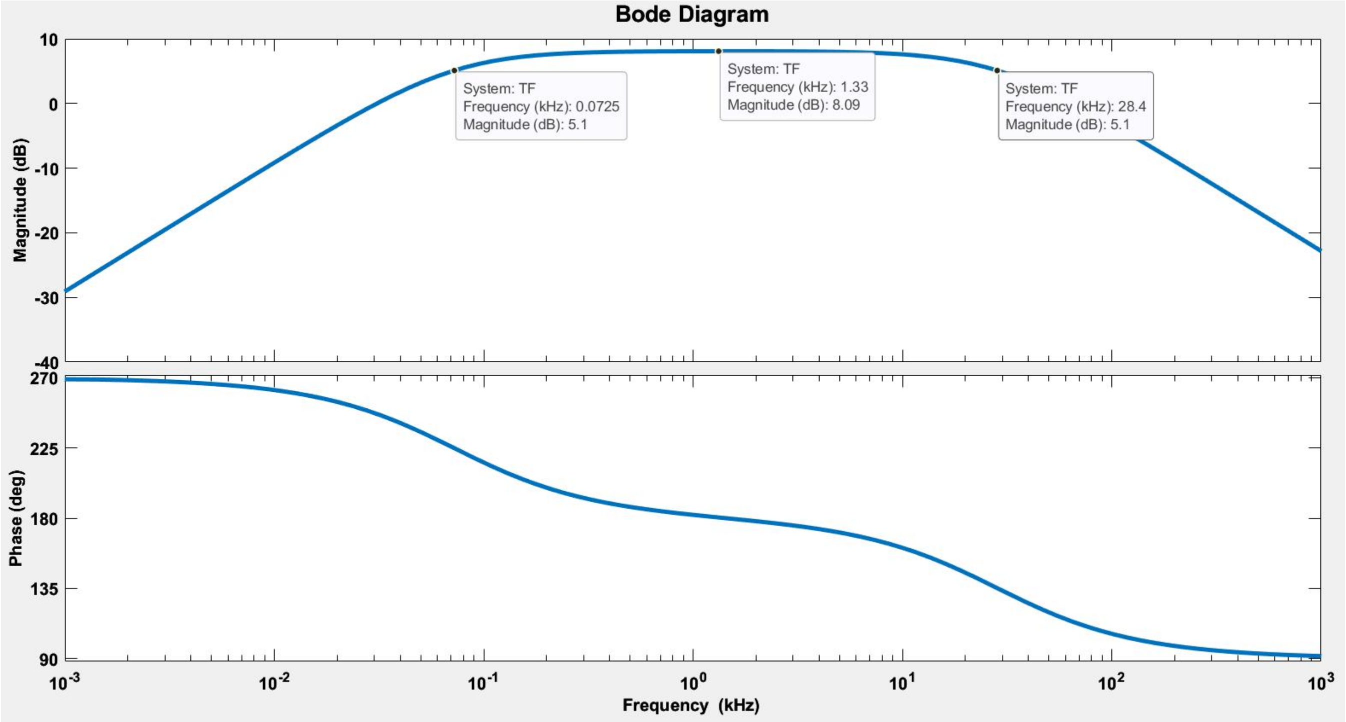
[4] R. J. Cameron, Microwave Filters for Communication Systems, New Jersey: John Wiley & Sons, Inc., 2007.

[5] M. H. Memon, Film Bulk Acoustic Wave Resonator in RF Filters, Hefei, China: IEEE, 2018.

[6] D. M. Pozar, Microwave Engineering, Third ed., New Jersey: John Wiley & Sons, 2005.

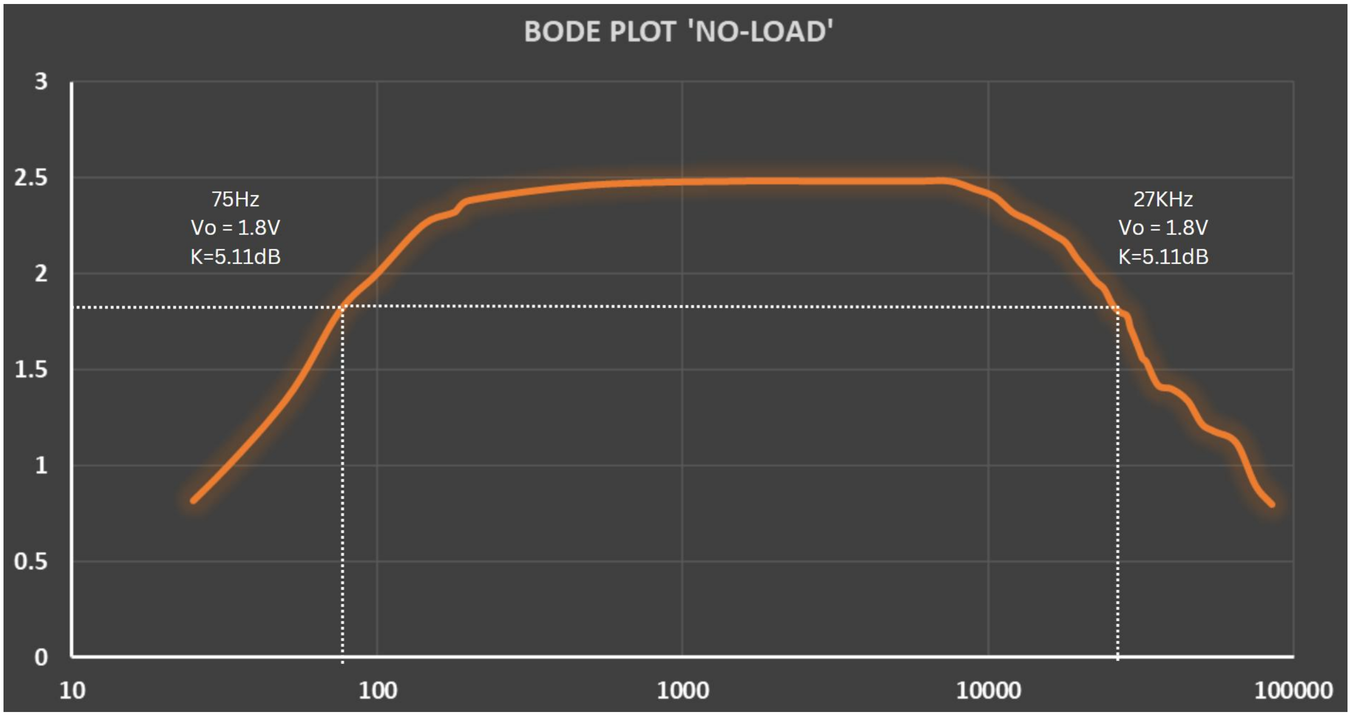
[7] T. Basaklar, “Wearable Devices and Low-Power Design for Smart Health Applications.,” IEEE, Wisconsin, 2021.

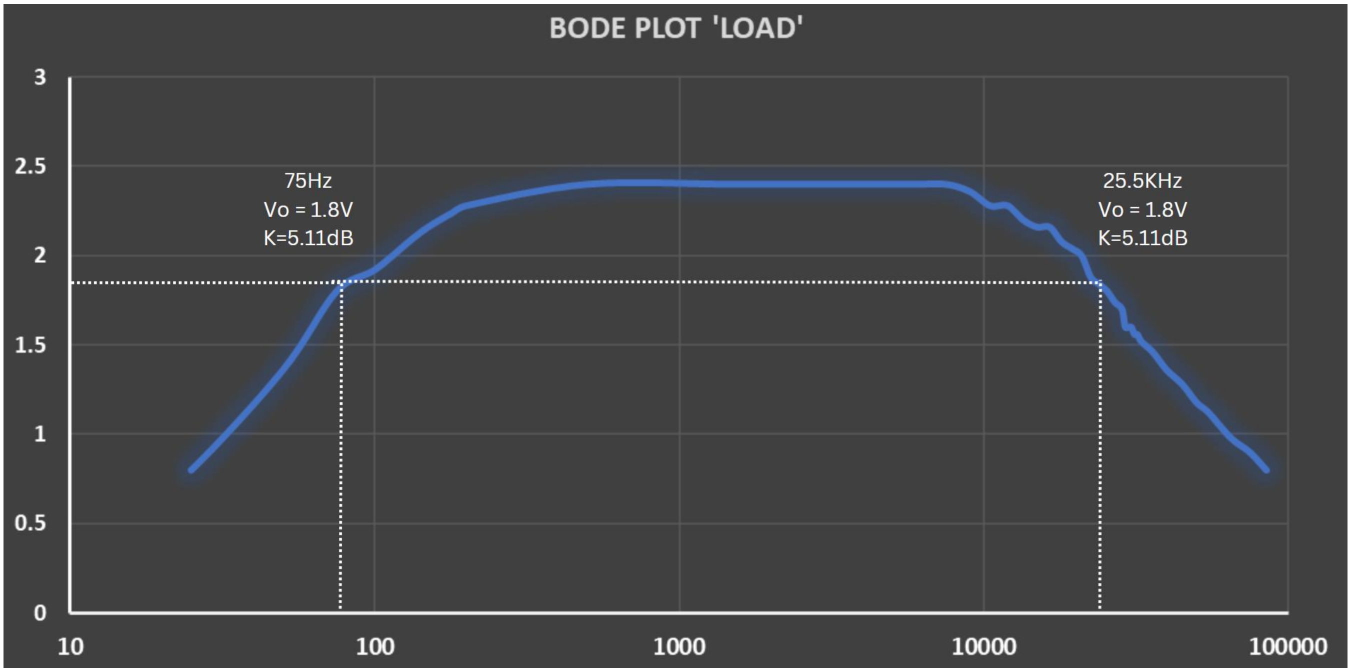
[8] L. Portilla, “Ambipolar Deep-Subthreshold Printed-Carbon-Nanotube Transistor,” *ACS Nano,* 2020.

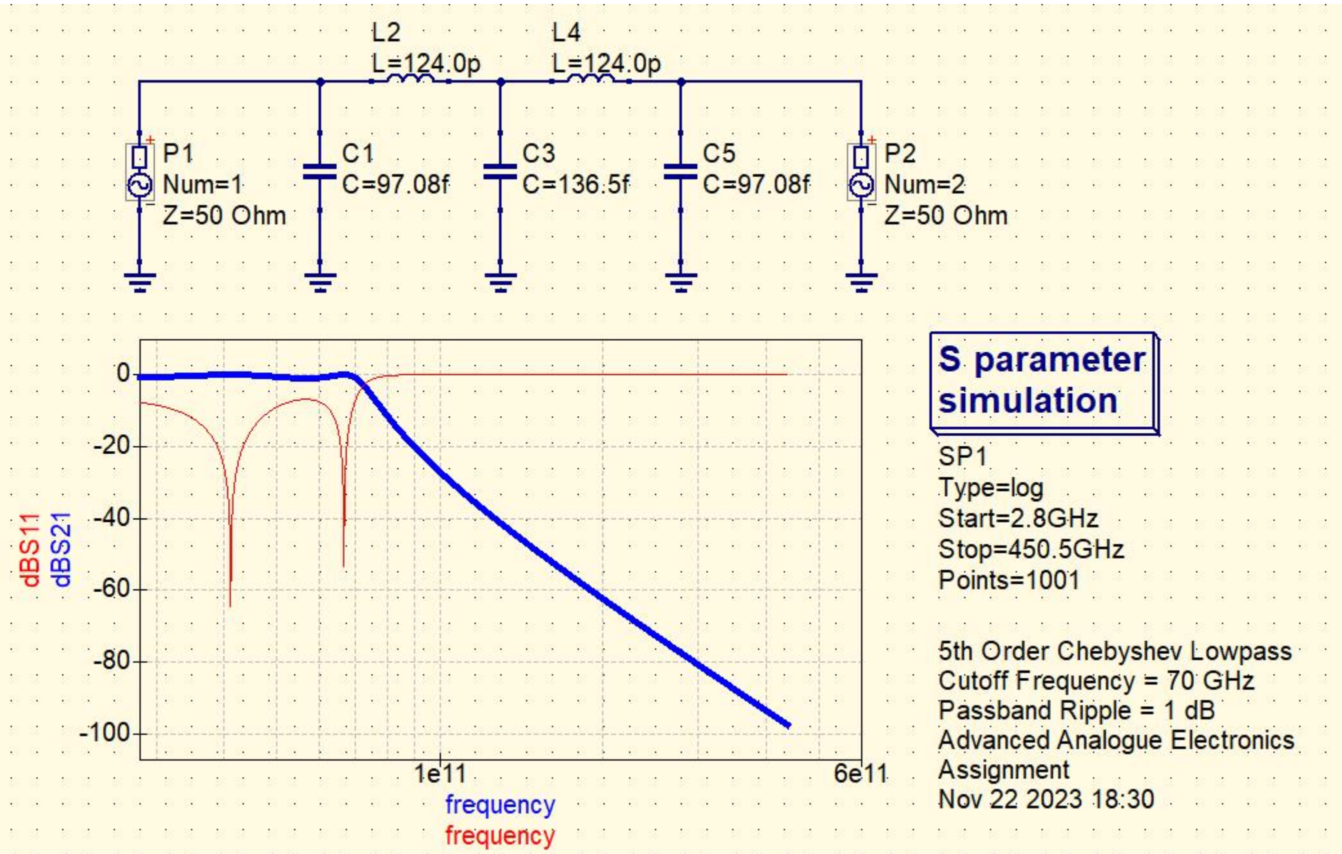


Bode Diagram
Magnitude (dB)
10
0
-10
-20
-30
-40
System: TF
Frequency (kHz): 0.0725
Magnitude (dB): 5.1
System: TF
Frequency (kHz): 1.33
Magnitude (dB): 8.09
System: TF
Frequency (kHz): 28.4
Magnitude (dB): 5.1
Phase (deg)
270
225
180
135
90
10^{-3}
10^{-2}
10^{-1}
10^{0}
10^{1}
10^{2}
10^{3}
Frequency (kHz)

FREQUENCY	Vout	Vin	GAIN
0	0	1	
15	0.55	1	-5.19
25	0.82	1	-1.72
50	1.34	1	2.54
75	1.8	1	5.11
100	2	1	6.02
130	2.2	1	6.85
150	2.28	1	7.16
180	2.32	1	7.31
200	2.38	1	7.53
500	2.46	1	7.82
1500	2.48	1	7.89
3000	2.48	1	7.89
4500	2.48	1	7.89
6000	2.48	1	7.89
7500	2.48	1	7.89
9000	2.44	1	7.75
10500	2.4	1	7.60
12000	2.32	1	7.31
13500	2.28	1	7.16
15000	2.24	1	7.00
16500	2.2	1	6.85
18000	2.16	1	6.69
19500	2.08	1	6.36
21000	2.02	1	6.11
22500	1.96	1	5.85
24000	1.92	1	5.67
25500	1.84	1	5.30
27000	1.8	1	5.11
28500	1.78	1	5.01
29200	1.72	1	4.71
29900	1.68	1	4.51
30600	1.64	1	4.30
31300	1.62	1	4.19
32000	1.58	1	3.97
33000	1.54	1	3.75
36000	1.42	1	3.05
40000	1.4	1	2.92
45000	1.34	1	2.54
50000	1.22	1	1.73
55000	1.18	1	1.44
65000	1.12	1	0.98
75000	0.9	1	-0.92
85000	0.8	1	-1.94


BODE PLOT 'NO-LOAD'
3
2.5
2
1.5
1
0.5
0
10
100
1000
10000
100000
75Hz
Vo = 1.8V
K=5.11dB
27KHz
Vo = 1.8V
K=5.11dB

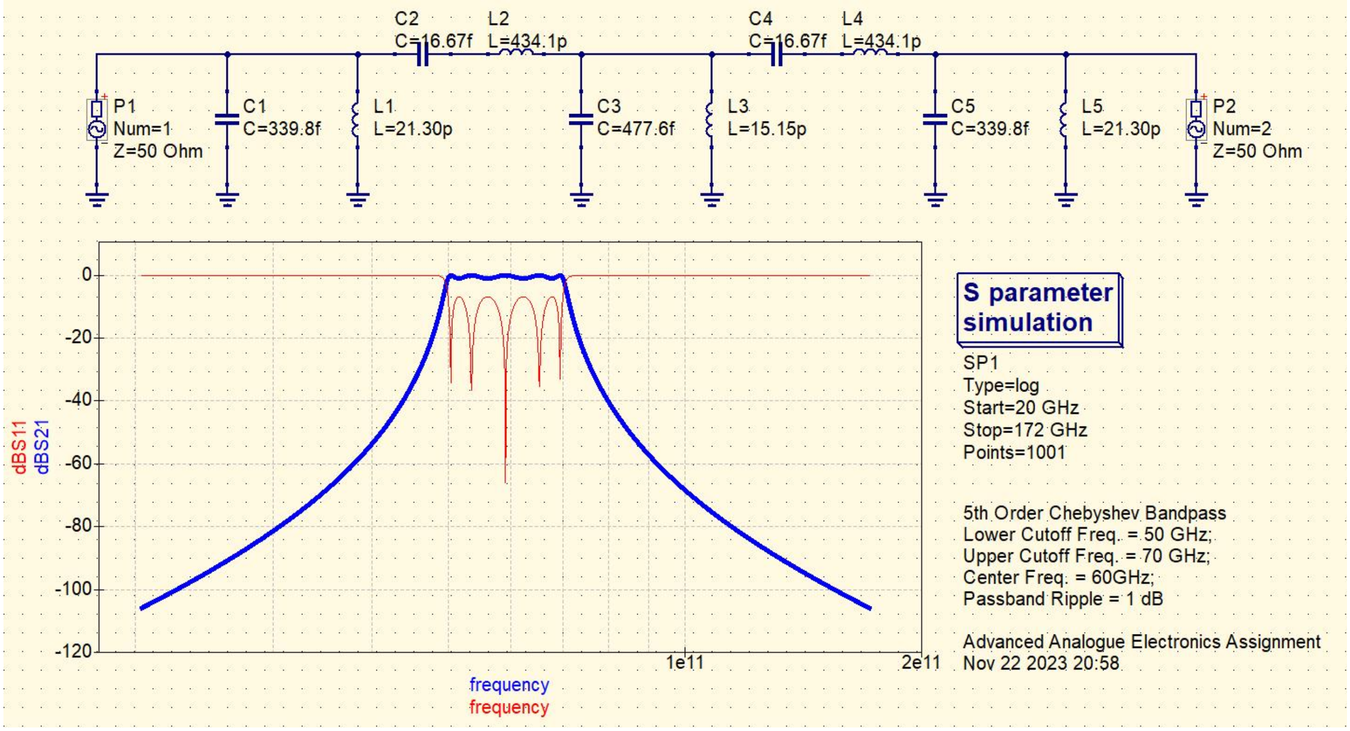

BODE PLOT 'LOAD'
3
2.5
2
1.5
1
0.5
0
10
100
1000
10000
100000
75Hz
Vo = 1.8V
K=5.11dB
25.5KHz
Vo = 1.8V
K=5.11dB



S parameter simulation

SP1
Type=log
Start=2.8GHz
Stop=450.5GHz
Points=1001

5th Order Chebyshev Lowpass
Cutoff Frequency = 70 GHz
Passband Ripple = 1 dB
Advanced Analogue Electronics Assignment
Nov 22 2023 18:30



S parameter simulation

SP1
Type=log
Start=20 GHz
Stop=172 GHz
Points=1001

5th Order Chebyshev Bandpass
Lower Cutoff Freq. = 50 GHz;
Upper Cutoff Freq. = 70 GHz;
Center Freq. = 60GHz;
Passband Ripple = 1 dB

Advanced Analogue Electronics Assignment
Nov 22 2023 20:58