# 7. Bandpass Filter

<u>Chapter 7 Goals</u>

- Introduce the tank circuit as a bandpass filter
- Become familiar with bandwidth and Q of a resonant circuit
- Analyze the filter with LTspice
- Breadboard and test filters

Many frequency components are contained in the output of the mixer. It is preferred that we filter out the unwanted components, otherwise we will waste power amplifying them. These unwanted frequency components can also cause noise in the output, and can even result in more noise due to mixing in nonlinear devices in the circuit.

Filters can be made using Op Amps (so called active filters), or can be made using inductors and capacitors (passive filters). The 4 categories of filter are low pass, high pass, band pass and band stop. Here we are interested in bandpass filters (abbreviated BPF).

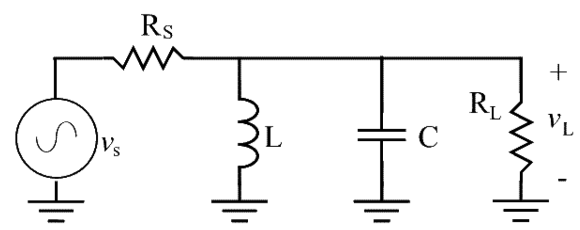


Figure 7.1: Tank circuit schematic

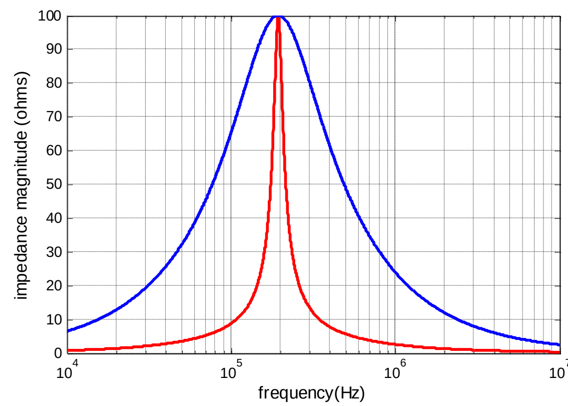


Figure 7.2: Impedance for tank circuit. The blue trace (wider bandwidth) has L = 100 µH, C = 0.0067 µF, $R_L$ = 100 Ω. The red trace (narrower bandwidth) has L = 10 µH, C = 0.067 µF, $R_L$ = 100 Ω.

## 7.1 The Tank Circuit

A simple and very useful bandpass filter, termed a *tank* circuit, is shown in Figure 7.1. The parallel inductor and capacitor combination will store energy at a resonance frequency, hence the term tank (storage tank). The impedance, *Z*, looking into the tank circuit (including the load resistance, $R_L$) is

$$Z = \frac{j\omega R_L L}{R_L\left(1-\omega^2 LC\right)+j\omega L} \qquad (7.1)$$

At resonance, the term $\left(1-\omega^2 LC\right)=0$, or

$$\omega_{res} = \frac{1}{\sqrt{LC}} = 2\pi f_{res} \qquad (7.2)$$

It is easy to show that at the resonance frequency the impedance will be equal to $R_L$. At values away from the resonance frequency, the impedance drops. This is shown in Figure 7.2 for a pair of tank circuits with a common resonance frequency but with different LC combinations. We observe that the lower value of inductance gives us a narrower bandwidth. For resonant circuits, we employ the term "Q" or "resonator Q" to define the quality of the resonant circuit. Note that this "Q" has <u>nothing</u> to do with the "Q" of "Q-point". For a tank circuit, it can be shown that

$$Q = \frac{R_L}{\omega_{res} L} = \omega_{res} R_L C \qquad (7.3)$$

Experimentally, we can measure the resonant Q by dividing the resonance frequency by the 3 dB bandwidth:

$$Q = \frac{f_{res}}{BW} \qquad (7.4)$$

Thus the narrower bandwidth result in Figure 7.2 has a higher value of Q than the broader bandwidth result.

<u>Exercise 7.1:</u> Derive (7.1).

<u>Exercise 7.2</u>: Suppose $L$ = 1 µH and $R_L$ = 1 kΩ for your tank circuit. Design the circuit to have a 600 kHz resonance frequency. What is the Q for this tank circuit?

<u>7.1a LTspice Simulation</u>

1. Build the default circuit in LTspice shown in Figure 7.3 with $L_{f1}$ = 10 µH, $C_{f1}$= 0.069 µF, $R_L$ = 100 Ω And $R_S$ = 100 Ω). For the voltage source, set the Small signal AC analysis values to 2V for AC Amplitude and an AC Phase of 0.

- Calculate the resonant frequency and resonator Q for this configuration:

  $f_{res}$ = ___________ Q = ___________

- Calculate the maximum possible value for the output voltage amplitude. This is found by removing Lf1 and Cf1.

  $V_{outmax}$=____________

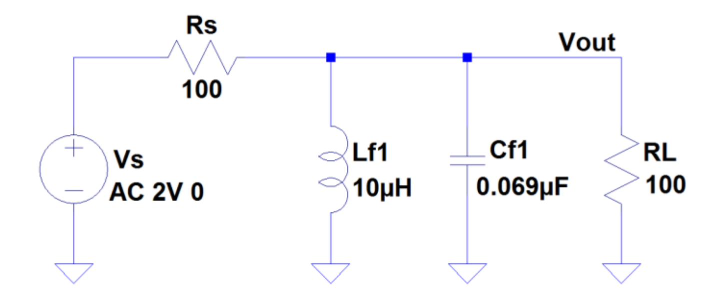


Figure 7.3: the default LTspice circuit

2. Now, in the Simulate menu, select Edit Simulation Command with the following settings:

| | |
|---|---|
| Type of Sweep: | Decade |
| Number of points per decade: | 101 |
| Start Frequency: | 1e3 |
| Stop Frequency: | 1e7 |

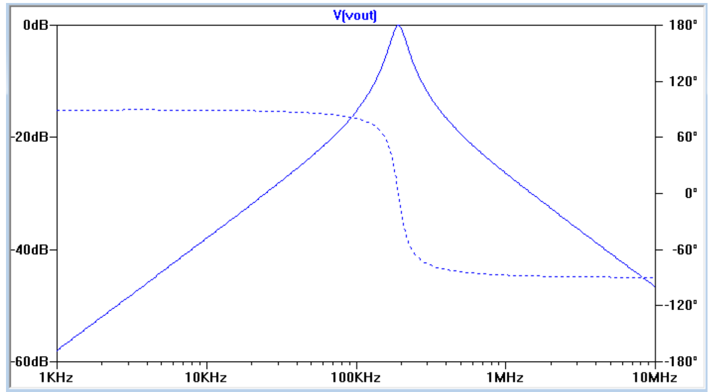


Figure 7.4 Voltage output for the default circuit

- In order to get voltage in dB, in step 1 we set the AC amplitude to 2 V and then the maximum possible output amplitude for the circuit will be 1 V, corresponding to 0 dB.
- After simulation, your output should appear as in Figure 7.4.

3. To better see the resonance and Q, zoom in on the peak, for instance changing the start and stop frequency range to 100 kHz and 1 MHz. Use display information about the cursor location to fill in Table 7.1.

4. Run another AC Sweep and collect the data for the case where $L_{f1}$ = 100 µH, $C_{f1}$ = 0.0069 µF, $R_L$ = 100 Ω and $R_S$ = 100 Ω. Change the Start Frequency to 10 kHz for this simulation. Record the data in Table 7.1.

5. Return to the default circuit. Change $R_L$ and $R_S$ to 10 Ω and run an AC sweep to find data for Table 7.1. Repeat for $R_L$ and $R_S$ equal to 1 kΩ. For these runs, adjust the simulation frequency as needed. How does Q vary with the changes in resistance?

Table 7.1 – LTspice AC sweep data

| $L_{f1}$ | 10 µH | 100 µH | 10 µH | 10 µH |
|---|---|---|---|---|
| $C_{f1}$ | 0.069 µF | 0.0069 µF | 0.069 µF | 0.069 µF |
| $R_L$, $R_S$ | 100 Ω | 100 Ω | 10 Ω | 1 kΩ |
| $f_{res}$ | | | | |
| vdB(@ $f_{res}$) | | | | |
| $f_{(-3\text{ dB, low})}$ | | | | |
| $f_{(-3\text{ dB, high})}$ | | | | |
| BW | | | | |
| Q = $f_{res}$/BW | | | | |

Table 7.2 – measurement data

| | | | | |
|---|---|---|---|---|
| $L_{f1}$ | 10 μH | 10 μH | | |
| $C_{f1}$ | 0.069 μF | 0.069 μF | | |
| $R_L$, $R_S$ | 100 Ω | 1 kΩ | | |
| $f_{res}$ | | | | |
| $f_{(-3\ dB,\ low)}$ | | | | |
| $f_{(-3\ dB,\ high)}$ | | | | |
| BW | | | | |
| $Q = f_{res}/BW$ | | | | |

7.1b Assemble the Circuit

1. Assemble the default circuit of Figure 7.3 (L = 10 μH, C = 0.069 μF, $R_L$ = 100 Ω and $R_S$ = 100 Ω). Here you will use a parallel combination of capacitors, a 0.047 μF and a 0.022 μF, to achieve 0.069 μF.

2. Use the generator to provide a variable frequency input signal, and use the dual trace feature of your oscilloscope to observe the input and output signals simultaneously. Maintain constant amplitude on the input signal. Use Table 7.2 to assist your calculations and record your results for $f_{res}$ and $Q$.

   *Comment: Rather than performing a manual sweep of frequency, a network analyzer may be used to automatically sweep frequency and can display an output similar to what is observed in Figures 7.2 and 7.4 (the output in this case is a transmission parameter, termed $S_{21}$, a topic beyond our present expertise). One limitation is that network analyzers typically have fixed 50 Ω impedance terminations, which would be the values of $R_S$ and $R_L$.*

3. Replace $R_S$ and $R_L$ with 1 kΩ. Find the new fres and Q for Table 7.2.

   *Comment: If possible, retain your bandpass filter circuit for use at the end of chapter 8.*

**7.2 Additional Filter Projects**

7.2a Op Amp Filter

A low pass filter can be realized using an Op Amp, and this can have the benefit of sharper rolloff and even amplification of the signal. Care must be taken to find a high speed Op Amp if the intent is to build a filter that will operate at RF frequencies. A good resource is Richard Jaeger's Microelectronics Circuit Design.

**7.2b Chebyshev, Butterworth, T or pi filter**

Filters using more than two elements can be constructed which will have sharper rolloff. T and pi filters have three elements. Chebyshev and Butterworth filters can have many elements.

(notes)