5

Filters

So far the filters we have made have had only two elements: a capacitor and a resistor or inductor. We can improve the response of our filters by adding more elements. This allows us to make the pass band flatter and the roll-off steeper. Multielement filters behave somewhat like transmission lines, and we need to have the right input and output resistance to avoid problems with reflections. Analyzing these filters by hand is quite difficult, but the calculations are easy on a computer. For this we will use a computer program called *Puff*, which is included with this book. Instructions for running the program are given in Appendix C.

5.1 Ladder Filters

We will consider ladder networks with alternating series and shunt elements like the discrete transmission line we studied in Problem 11. If the series elements are inductors and the shunt elements are capacitors, then the circuit acts as a low-pass filter (Figure 5.1a, b). At low frequencies, the impedance of the inductors and the admittance of the capacitors are small, and the input signal passes through to the output with little loss. In contrast, at high frequencies the inductors begin to act as voltage dividers and the capacitors as current dividers. This reduces the power transmitted to the load. We can also make high-pass filters with series capacitors and shunt inductors (Figure 5.1c, d).

Many different filters have been developed, giving a wide choice of amplitude, phase, pass-band, and stop-band characteristics. We will focus on amplitude and consider two different types – the Butterworth filter, which gives a flat pass band, and the Chebyshev filter, which gives an excellent roll-off. Mathematically, we write the loss factor L for the Butterworth low-pass filter as

$$L = P_i/P = 1 + (f/f_c)^{2n}, \tag{5.1}$$

where the input power P_i is the available power from the source, P is the output power delivered to the load, and f_c is the 3-dB cut-off frequency. This is shown in Figure 5.2a. The loss characteristic is quite flat in the pass band. In fact, the first $2n - 1$ frequency derivatives are zero at $f = 0$. For this reason, people call these *maximally flat* filters. When we are well into the stop band, the loss factor can be

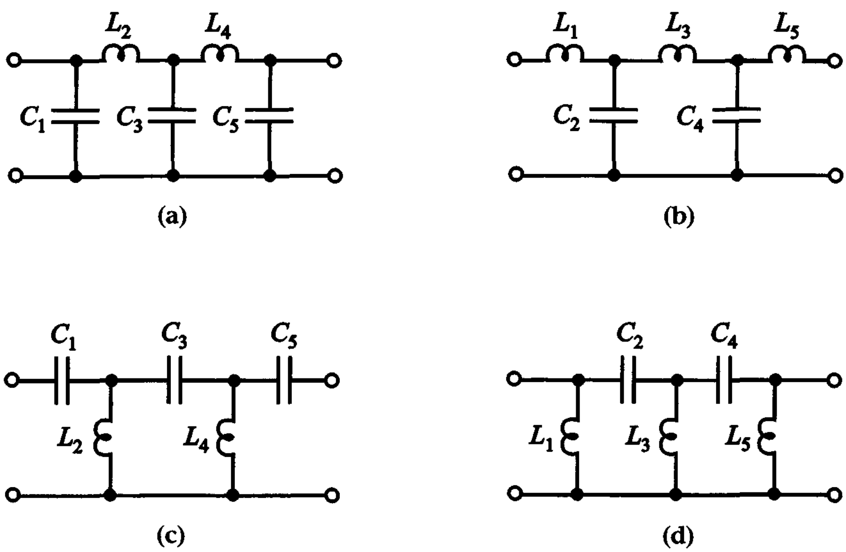


Figure 5.1. Low-pass (a, b) and high-pass (c, d) ladder filters. Often inductors are larger and more expensive than capacitors, and so filters that use fewer inductors, like (a) and (c), are more common. The number of elements is called the *order* of the filter. These are all fifth-order filters.

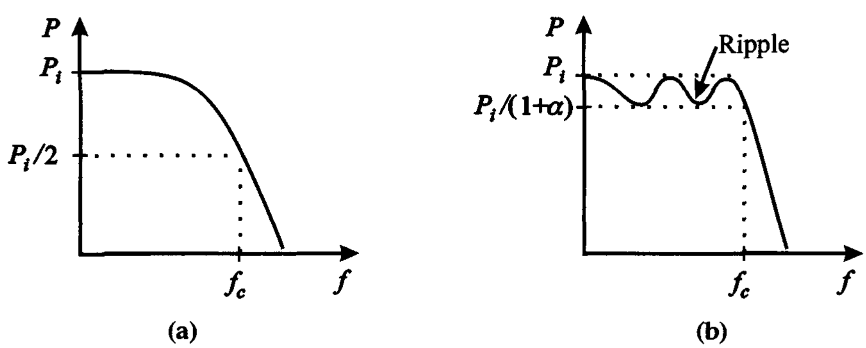


Figure 5.2. Butterworth (a) and Chebyshev (b) filter characteristics.

written approximately as

$$L \approx (f/f_c)^{2n}. \tag{5.2}$$

This means that the loss increases by $6n$ dB each time the frequency doubles. We say that the loss increases by 6 dB per octave per element.

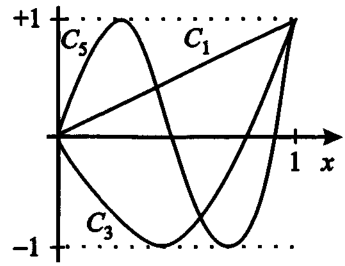
It turns out that we can get a faster roll-off if we allow a ripple in the pass band. The Chebyshev filter takes advantage of this (Figure 5.2b). Its loss is given by

$$L = 1 + \alpha C_n^2(f/f_c), \tag{5.3}$$

where α is the ripple size and $C_n(x)$ is the Chebyshev polynomial of order n. Chebyshev polynomials have the interesting property that they oscillate between -1

Order	Polynomial
0	1
1	x
2	$2x^2 - 1$
3	$4x^3 - 3x$
4	$8x^4 - 8x^2 + 1$
5	$16x^5 - 20x^3 + 5x$
6	$32x^6 - 48x^4 + 18x^2 - 1$
7	$64x^7 - 112x^5 + 56x^3 - 7x$

(a)



(b)

Figure 5.3. The Chebyshev polynomials (a), and plots of C_1, C_3, and C_5 (b).

and +1 as x varies from −1 and +1. The first two polynomials are given by

$$C_0 = 1, \tag{5.4}$$

$$C_1 = x. \tag{5.5}$$

We can calculate the rest from the following formula:

$$C_i(x) = 2xC_{i-1}(x) - C_{i-2}(x). \tag{5.6}$$

You need to calculate the polynomials in order to use this formula. Figure 5.3a gives the Chebyshev polynomials through order 7.

We will use only the odd-order polynomials, because the even-order polynomials are for filters with different source and load resistances. The odd-order polynomials are 0 when x is zero, and then oscillate between +1 and −1, finally ending up at +1 when $x = 1$ (Figure 5.3b). This is really rather surprising, considering how large the coefficients of the polynomials are. For example, the cubic coefficient in C_5 is 20. However, the different terms offset each other as long as $x \leq 1$. Now consider what this means for the loss. Each time the Chebyshev polynomial hits either +1 or −1, the loss factor becomes $1 + \alpha$. This means that in the pass band, the loss factor swings between 1 and $1 + \alpha$ repeatedly. For this reason we call this an *equal-ripple* filter. This happens once for C_1, twice for C_3, and three times for C_5. This is shown in Figure 5.2b for a fifth-order filter. The last time the loss factor is $1 + \alpha$ is when $f = f_c$, at the pass-band edge. As the frequency increases above f_c, the loss factor increases sharply, and the filter rolls off rapidly. The roll-off advantage for the Chebyshev filters comes from the large leading coefficient, given by 2^{n-1}. By comparison, this coefficient for Butterworth filters is 1. In dB terms, the loss factor of the Chebyshev filter in the stop band is $6(n-1)$ dB larger than that of a Butterworth filter with the same pass-band loss. For example, for fifth-order filters this is 24 dB, a considerable improvement.

5.2 Filter Tables

A good way to start a filter design is to consult a table that lists component values for different filters. With these values, you can simulate the filter response on a computer, and from there you can make adjustments to account for the available components. Manufacturers make only a limited range of capacitor values. Inductors also must have an integral number of turns. In practice, however, the turns can be squeezed to increase the inductance somewhat, or spread to reduce it. Also, the components themselves have loss, and this effect can be included in the computer simulations. Finally, it is a good idea to see the effect of component variation. Chebyshev filters are particularly sensitive to this. Deriving the formulas for the tables is quite difficult, and so I will just give the results. These formulas are for filters with the same source and load resistance, and we will use this resistance for normalizing. The normalized susceptances and reactances for a Butterworth filter at f_c are given by

$$a_i = 2\sin\left(\frac{(2i-1)\pi}{2n}\right), \tag{5.7}$$

where i is an index for the components and n is the order of the filter. These values are given in Table 5.1a through seventh order.

Calculating the values for a Chebyshev filter is quite involved. We usually specify a maximum ripple loss in dB in the pass band. In practice, these specifications vary over a wide range from 0.01 dB to 1 dB. We relate this loss L_r to α by

$$1+\alpha = 10^{L_r/10}. \tag{5.8}$$

We calculate an auxiliary quantity β as

$$\beta = \sinh\left(\frac{\tanh^{-1}(1/\sqrt{1+\alpha})}{n}\right), \tag{5.9}$$

where sinh is the hyperbolic sine function and $\tanh^{-1}$ is the inverse of the hyperbolic tangent. We calculate the Chebyshev components in order, starting with

$$c_1 = a_1/\beta, \tag{5.10}$$

where a_1 is given by Equation 5.7. Then we proceed sequentially to c_2 and the rest, using the formula

$$c_i = \frac{a_i a_{i-1}}{c_{i-1}(\beta^2 + \sin^2[(i-1)\pi/n])}. \tag{5.11}$$

The Chebyshev components for 0.2-dB ripple are given in Table 5.1b.

Table 5.1. Component Values for Ladder Filters.

Order	a_1	a_2	a_3	a_4	a_5	a_6	a_7
1	2						
2	$\sqrt{2}$	$\sqrt{2}$					
3	1	2	1				
4	0.765	1.848	1.848	0.765			
5	0.618	1.618	2	1.618	0.618		
6	0.518	$\sqrt{2}$	1.932	1.932	$\sqrt{2}$	0.518	
7	0.445	1.247	1.802	2	1.802	1.247	0.445

(a)

Order	c_1	c_2	c_3	c_4	c_5	c_6	c_7
1	0.434						
3	1.228	1.153	1.228				
5	1.339	1.337	2.166	1.337	1.339		
7	1.372	1.378	2.275	1.500	2.275	1.378	1.372

(b)

Note: The values for Butterworth filters from Equation 5.7 are given in (a), and for Chebyshev filters with a ripple of 0.2 dB from Equation 5.11 in (b). These are the normalized susceptances of the shunt elements at f_c, and the normalized reactances of the series elements. People call these *immittance* values to indicate that the numbers can be used for either susceptance or reactance. Any of the filters shown in Figure 5.1 can be designed from these tables. For a low-pass filter, the series elements are inductors, and the shunt elements are capacitors. For a high-pass filter, the series elements are capacitors and the shunt elements are inductors, and the reactances and susceptances are negative. Because the values are symmetric, we can count from either end. Start with either a series element or a shunt element, and alternate throughout the filter. One warning: By tradition, f_c for a Butterworth filter means the 3-dB frequency, but for Chebyshev filters, f_c is defined by the frequency that gives the maximum ripple, 0.2 dB in this case.

5.3 Examples

Using the tables is surprisingly complicated, and it is a good idea to simulate filters on the computer to make sure that you actually get the characteristics you want. To show how the tables work, we will go through several examples in detail. Let us assume that we need a filter for a 50-Ω antenna cable with a 3-dB cut-off frequency of 10 MHz and a loss of at least 20 dB at 20 MHz. A fourth-order Butterworth filter with a cut-off frequency of 10 MHz should have a loss at 20 MHz of

$$L(20\,\text{MHz}) = 6n = 24\,\text{dB}, \tag{5.12}$$

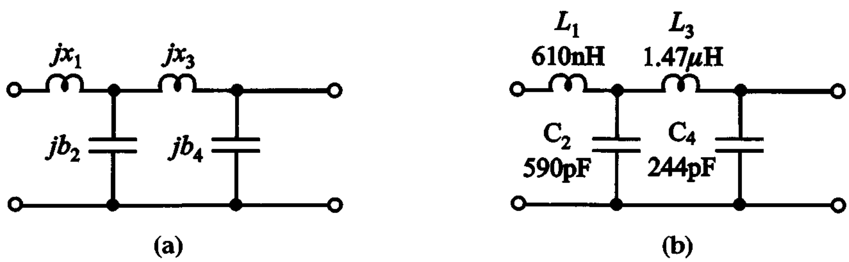


Figure 5.4. Designing a 50-Ω low-pass Butterworth filter with $f_c = 10$ MHz. Defining the normalized reactances for the series elements and the normalized susceptances for the shunt elements (a), and the component values (b).

which is sufficient. We start with the filter structure in Figure 5.4a. There are two series inductors and two shunt capacitors. From Table 5.1a, we can write the normalized reactance of the first inductor as

$$x_1 = a_1 = 0.765. \tag{5.13}$$

We can find the actual reactance X_1 at 10 MHz by multiplying by the characteristic impedance of the cable, $Z_0 = 50\,\Omega$. This gives us

$$X_1 = x_1 \cdot Z_0 = 38\,\Omega, \tag{5.14}$$

and the inductance L_1 is given by

$$L_1 = X_1/\omega_c = 610\,\text{nH}. \tag{5.15}$$

Now we proceed to the other values. From the table, the normalized susceptance of the first shunt capacitor is given by

$$b_2 = a_2 = 1.848. \tag{5.16}$$

We can find the actual susceptance B_2 at 10 MHz by dividing by Z_0. This gives us

$$B_2 = b_2/Z_0 = 37\,\text{mS}, \tag{5.17}$$

and the capacitance C_2 is given by

$$C_2 = B_2/\omega_c = 590\,\text{pF}. \tag{5.18}$$

The inductance L_3 is given by

$$L_3 = a_3 Z_0/\omega_c = 1.47\,\mu\text{H}, \tag{5.19}$$

and finally

$$C_4 = \frac{a_4}{Z_0\omega_c} = 244\,\text{pF}. \tag{5.20}$$

Figure 5.4b shows the complete filter.

Now consider a high-pass filter with a 3-dB cut-off frequency of 10 MHz and a loss at 5 MHz of at least 20 dB. Figure 5.5a shows the structure, with series

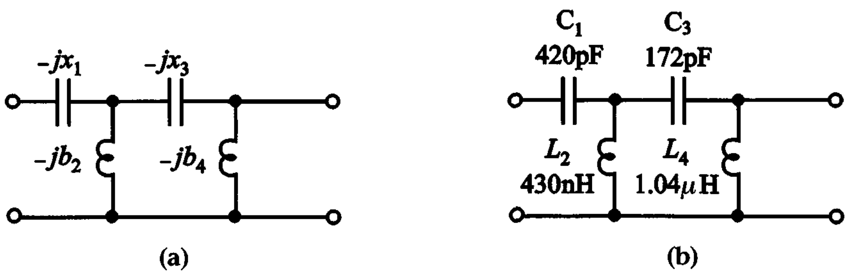


Figure 5.5. Designing a high-pass Butterworth filter. Defining the normalized reactances for the series elements and the normalized susceptances for the shunt elements (a), and the final component values (b).

capacitors and shunt inductors. We use the same table values as before. This time the reactances and susceptances are negative. We write

$$C_1 = \frac{1}{a_1 Z_0 \omega_c} = 420\,\text{pF}, \tag{5.21}$$

$$L_2 = \frac{Z_0}{a_2 \omega_c} = 430\,\text{nH}, \tag{5.22}$$

$$C_3 = \frac{1}{a_3 Z_0 \omega_c} = 172\,\text{pF}, \tag{5.23}$$

$$L_4 = \frac{Z_0}{a_4 \omega_c} = 1.04\,\mu\text{H}. \tag{5.24}$$

These values are shown in Figure 5.5b.

Figure 5.6 shows a simulation of the loss of these two filters with the computer program *Puff*. The response of these two filters is complementary, and they cross at the 3-dB level at 10 MHz. One interesting fact is that the filters show *reciprocity*. This means that it does not matter which end we use for the input. This is not obvious, because the filters are not symmetric end to end, but it is easily checked on the computer by swapping the input and output. We will see another example of reciprocity when we study antennas in Chapter 15.

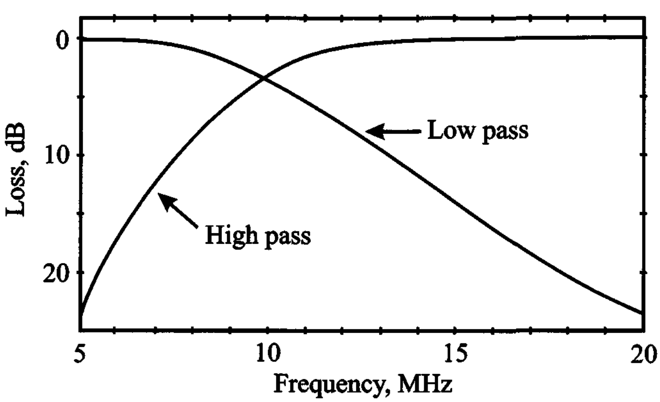


Figure 5.6. *Puff* simulation for the low-pass Butterworth filter in Figure 5.4b and the high-pass Butterworth filter in Figure 5.5b.

5.4 Band-Pass Filters

The ladder structure can also be used for band-pass and band-stop filters. For band-pass filters, the series elements are series resonant circuits, and the parallel elements are parallel resonant circuits (Figure 5.7a). Each of the elements is resonant at the center frequency f_0, so that the signal passes through unaffected. For band-stop filters, it is the other way around (Figure 5.7b), and the resonant circuits are arranged to block the signal at f_0. We can make Butterworth and Chebyshev filters with the same tables as before. For the band-pass filter, we find the series inductors and shunt capacitors as we did in the low-pass filter, except we use the filter bandwidth $\Delta\omega$ instead of ω_c in the reactance and susceptance calculations. For the band-stop filter, the values for the series capacitances and shunt inductors are calculated like the ones in the high-pass filters, but with $\Delta\omega$ instead of ω_c.

As an example, let us design a second-order band-pass Butterworth filter for 7 MHz, by adding a parallel resonant element to the series resonant circuit that we tested in Problem 8. The series resonant circuit had a 15-μH inductor and a variable capacitor that was adjusted for resonance at 7 MHz. This means that we can write

$$L_1 = 15\,\mu\text{H} \tag{5.25}$$

and

$$C_1 = \frac{1}{\omega_0^2 L_1} = 34.5\,\text{pF}. \tag{5.26}$$

For the first element, Table 5.1a gives us the value

$$a_1 = \sqrt{2}. \tag{5.27}$$

This is the normalized reactance of L_1 evaluated at $\Delta\omega$, and therefore we write

$$\Delta\omega L_1 = a_1 Z_0 \tag{5.28}$$

so that

$$\Delta\omega = a_1 Z_0 / L_1 = 4.71 \times 10^6\ \text{radians/s}. \tag{5.29}$$

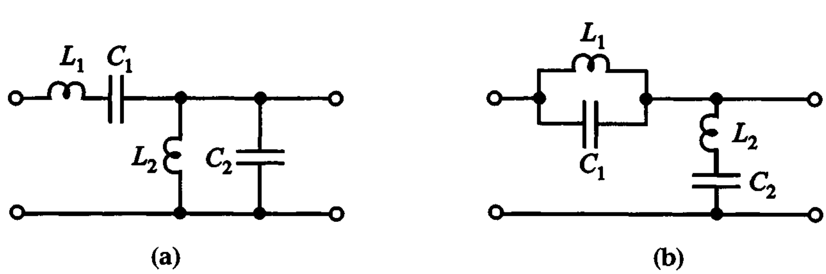


Figure 5.7. Band-pass filter (a), and band-stop filter (b). These are second-order filters.

In hertz, this is

$$\Delta f = \Delta\omega/(2\pi) = 750\,\text{kHz}. \tag{5.30}$$

Thus the 3-dB bandwidth of the filter is 750 kHz. Now we find C_2. We write

$$C_2 = \frac{a_2}{Z_0 \Delta\omega} \tag{5.31}$$

and substitute for $\Delta\omega$ from Equation 5.29 to get

$$C_2 = L_1/Z_0^2 = 6.0\,\text{nF}, \tag{5.32}$$

where we have used the fact that $a_1 = a_2$. The inductance L_2 is given by

$$L_2 = \frac{1}{\omega_0^2 C_2} = 86\,\text{nH}. \tag{5.33}$$

These component values are shown in Figure 5.8a.

We plot the response of the filter for two different loss scales. Figure 5.9 shows the response from 0 to 3 dB. For comparison, I have plotted the loss for the

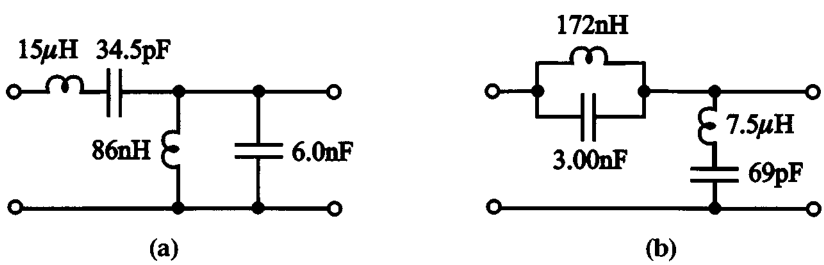


Figure 5.8. Component values for second-order, 7-MHz Butterworth band-pass (a) and band-stop (b) filters. The 3-dB bandwidth for each filter is $f_c = 750$ kHz.

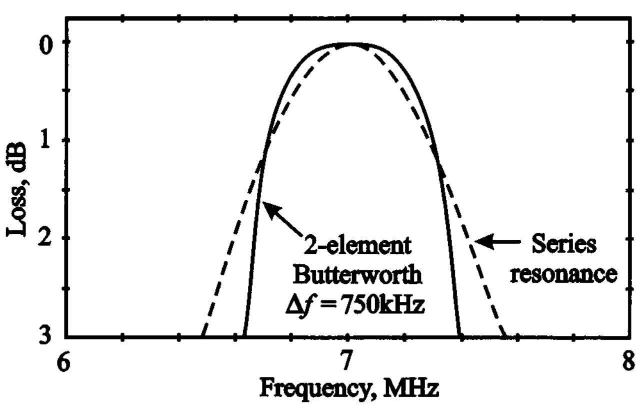


Figure 5.9. *Puff* simulation for the 2-element band-pass Butterworth filter in Figure 5.8a showing the range from 0 to 3 dB. The 3-dB bandwidth is 750 kHz, as predicted by Equation 5.30. Also shown is a plot for the series resonant element alone for comparison.

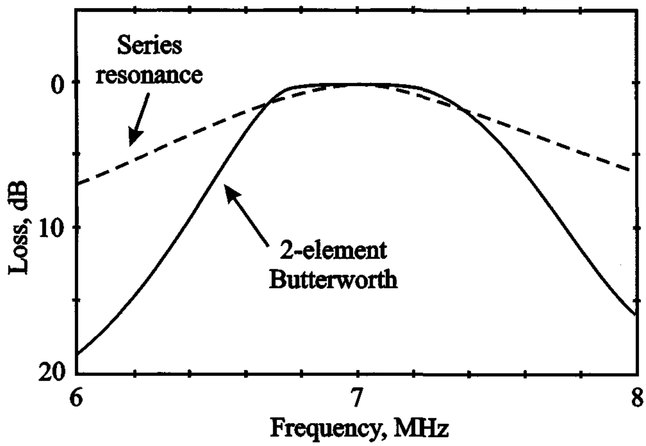


Figure 5.10. The same *Puff* simulation as in Figure 5.9, but with the range extended to 20 dB to show how the 2-element Butterworth filter rolls off much faster than the single resonant circuit.

series resonant circuit alone. This plot shows that near the center frequency, the response of the Butterworth filter is flatter than the series resonant circuit. Figure 5.10 is the same simulation with the loss scale extended to 20 dB. This shows that the Butterworth filter rolls off much more quickly than the series resonant circuit.

As a final example, we design a 2-element band-stop Butterworth filter with a 3-dB bandwidth of $\Delta f = 750$ kHz centered on 7 MHz. We use the circuit shown in Figure 5.7b. We write

$$C_1 = \frac{1}{a_1 Z_0 \Delta\omega} = 3.00\,\text{nF}, \tag{5.34}$$

$$L_1 = \frac{1}{\omega_0^2 C_1} = 172\,\text{nH}, \tag{5.35}$$

$$L_2 = \frac{Z_0}{a_2 \Delta\omega} = 7.5\,\mu\text{H}, \tag{5.36}$$

$$C_2 = \frac{1}{\omega_0^2 L_2} = 69\,\text{pF}. \tag{5.37}$$

These component values are shown in Figure 5.8b. The response of this filter is shown in Figure 5.11, together with the band-pass filter plot for comparison.

One thing to notice about the 2-element Butterworth filters is that the immittance values a_1 and a_2 are equal. This means that the normalized reactance of the series components is the same as the normalized susceptance of the shunt components. This gives us a way to recognize this filter in a circuit. The RF Filter in the NorCal 40A is a second-order Butterworth band-pass filter. We will study it in Problem 16. In addition, we can think of the IF Filter as a cascade of a pair of these band-pass filters. The IF Filter is a critical circuit in the receiver.

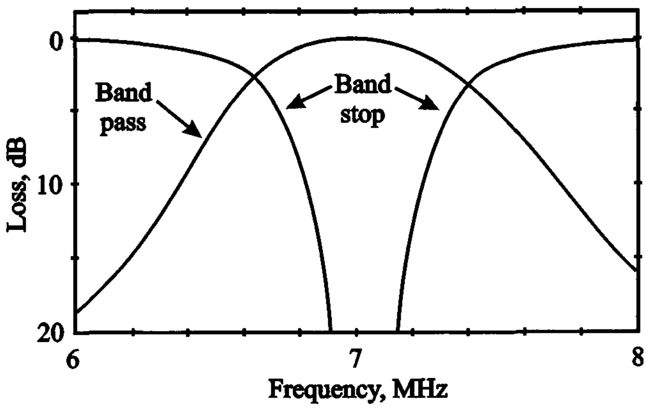


Figure 5.11. *Puff* simulation for the 2-element band-stop Butterworth filter in Figure 5.8b with the band-pass filter in Figure 5.8a shown for comparison.

It uses high-Q quartz-crystal resonators to achieve an extremely narrow bandwidth.

5.5 Crystals

Crystal quartz is an important material in electronics. Quartz crystals allow watches to keep precise time, and they control the master oscillators in microprocessor systems. In radios, they set oscillation frequencies and act as extremely narrow-band filters. This crucial role may seem surprising, because quartz is an insulator. However, quartz has several interesting properties. It is *piezoelectric*, which means that when we apply a voltage across it, it moves. The piezoelectric effect allows us to couple electrical signals to mechanical vibrations. This works both ways. Voltages cause motion, and motion causes voltages. For example, gas stoves and water heaters often use piezoelectric starters. In a piezoelectric starter, a force deforms the crystal, causing a large voltage across the contacts to make a spark.

Quartz mechanical resonators have very high Qs in the range of 50,000 to 100,000. The main loss is not within the quartz itself, but to the air and the supports for the crystal. These Qs are much higher than those of LC and transmission-line resonators, which are limited by metal resistance and are usually 100 or less. The resonant frequencies are in the range from 1 kHz to 100 MHz. Chemically, quartz is silicon dioxide. The raw material for quartz is sand, and this means that quartz crystals can be manufactured inexpensively. In addition, quartz crystals can be cut precisely so that the resonant frequencies change only slightly with temperature. The orientation in our crystals is called an AT cut, and this gives a temperature coefficient as low as 1 part in a million per degree Celsius. In a clock,

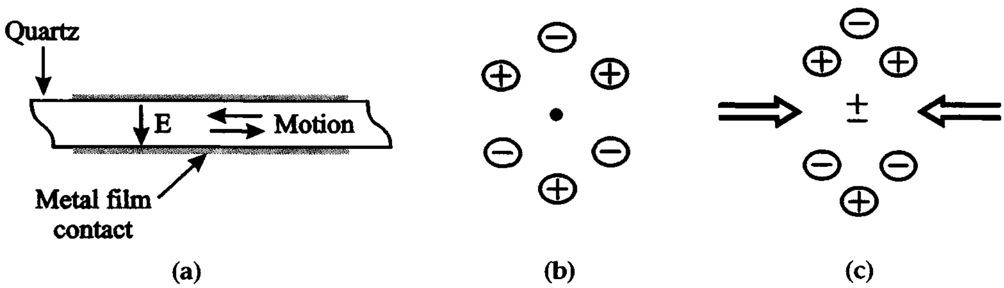


Figure 5.12. (a) Quartz crystal with metal contacts. The thickness needed for AT-cut quartz in the lowest shear mode is approximately 1.67 mm/f, where f is the frequency in MHz. (b) Structure of a material with a piezoelectric effect. The center of balance of the positive and negative charge is the same. (c) The charge movement when a force is applied. The charge movement causes the center of balance for the positive and negative charges to separate. The + indicates the center of positive charge, and the – the center of negative charge.

this corresponds to an error of one second per day for a 10°C change in temperature. This stability is also important for transceivers, because the frequency should not shift when the temperature changes. Figure 5.12a shows the structure. A thin wafer of quartz has evaporated metal film contacts on each side. A voltage between the metal contacts creates a vertical electric field E between the plates, which causes a horizontal *shear* movement in the crystal.

The details of the piezoelectric effect are complicated, but we can say qualitatively why it happens. Inside a solid, different atoms carry different charges. For example, in quartz the oxygen atoms have a net negative charge, and the silicon atoms have a net positive charge. In Figure 5.12b, we show the charged atoms in a triangular arrangement. The center of balance of the positive charges and the center of balance of the negative charges are in the same position, indicated by the dot. When a force is applied, the charge centers separate (Figure 5.12c). This causes a voltage across the material.

In circuit terms, a crystal has both a series and a parallel resonance. Figure 5.13a shows the schematic symbol for a crystal, and Figure 5.13b shows an equivalent circuit. It includes a series RLC circuit. However, this does not represent an electrical effect, but a mechanical one. L and C are called *motional inductance* and

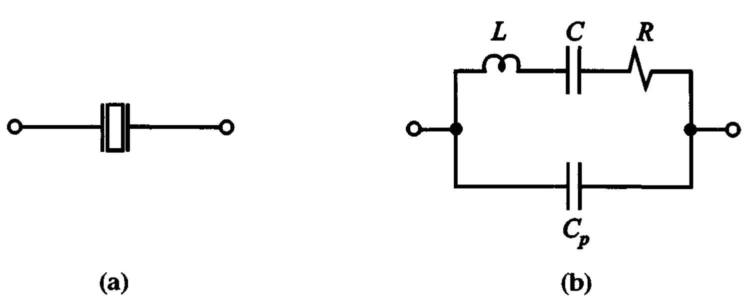


Figure 5.13. Schematic symbol for a crystal (a), and the equivalent circuit (b).

motional capacitance to make this clear. Here L accounts for the crystal density, C the stiffness, and R the loss during mechanical vibration. The parallel capacitance C_p, in contrast, is purely electrical. It arises from the capacitance between the two metal contacts, and it is usually a few picofarads. C_p gives the crystal a parallel resonance a few kilohertz above the series resonance. This affects filters that have wide pass bands that extend clear to the parallel resonance. You can ignore it in the filter that you make, which has a pass band of only a few hundred hertz.

5.6 Impedance Inverters

The band-pass filter that we designed requires both series and parallel resonant circuits. This is a problem if we try to make a narrow-band filter, because we need high-Q circuits. We can make excellent high-Q series resonant circuits with quartz crystals, but we do not have equivalent high-Q parallel resonant circuits. However, there are circuits that act as impedance inverters that effectively turn a series resonance into a parallel resonance. This allows us to make a band-pass filter with impedance inverters and series resonant quartz crystals. We have already studied one impedance-inverter circuit. We saw in the last chapter that a quarter-wave transmission line acts as an inverter. However, at our frequencies, a quarter-wave cable would be quite long. Fortunately, we can also make an impedance inverter with inductors and capacitors. This circuit is shown in Figure 5.14.

To see how this circuit works, write the input impedance Z_i as

$$Z_i = jX + \frac{1}{1/(-jX) + 1/(jX + Z_l)}. \tag{5.38}$$

After some arithmetic, this simplifies to

$$Z_i = \frac{X^2}{Z_l}. \tag{5.39}$$

If we define impedances normalized to the inverter reactance X, we can rewrite this as

$$z_i = 1/z_l. \tag{5.40}$$

The normalized input impedance is the inverse of the normalized load impedance. We could also say that the normalized input admittance is equal to the normalized

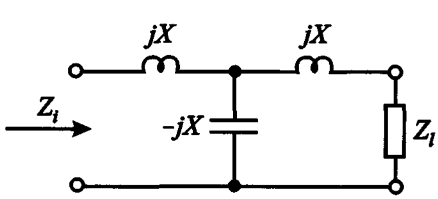


Figure 5.14. Impedance inverter with a load Z_l. The reactances of the inductors and the capacitor are the same.

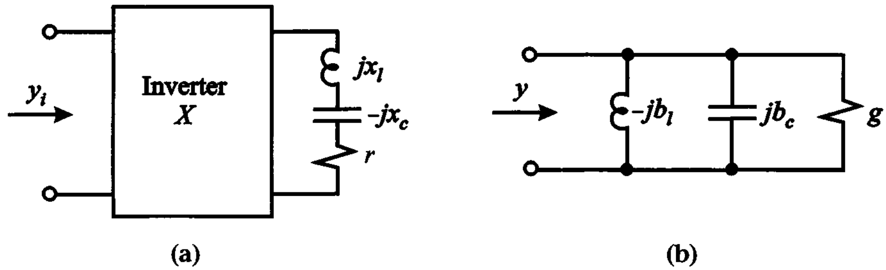


Figure 5.15. Inverter in front of a series resonant circuit (a), and the equivalent parallel resonant circuit (b). All quantities are normalized to the inverter reactance X.

load impedance:

$$y_i = z_l. \tag{5.41}$$

Notice that this kind of formula only makes sense for normalized quantities, because impedance and admittance ordinarily have different units. Also, for the inverter to work, we must be near the frequency where the reactances of the inductors and capacitors are equal. This is a reasonable assumption in crystal filters because the bandwidths are so narrow.

Now consider what happens when we put an inverter in front of a series resonant circuit (Figure 5.15a). We write the input admittance y_i with Equation 5.41 as

$$y_i = jx_l - jx_c + r. \tag{5.42}$$

Now let us compare this with the admittance of the parallel resonant circuit shown in Figure 5.15b. We can write this as

$$y = jb_c - jb_l + g. \tag{5.43}$$

If we compare these formulas, the two circuits are equivalent if

$$b_c = x_l, \tag{5.44}$$

$$b_l = x_c, \tag{5.45}$$

$$g = r. \tag{5.46}$$

This means that the combination of the inverter and the series resonant circuit behaves as a parallel resonant circuit.

Now we can understand the IF Filter in the NorCal 40A. The circuit is shown in Figure 5.16a. It is called a Cohn filter, after the American engineer, Seymour Cohn, who invented it. In the figure, all five capacitors are identical. The shunt capacitors act as impedance inverters. To make a proper impedance inverter we need series inductors to go with the shunt capacitors. We could include the inductors, but we can get an equivalent effect by adding capacitors at each end of the filter. To see how this works, consider that if we add a series combination of an

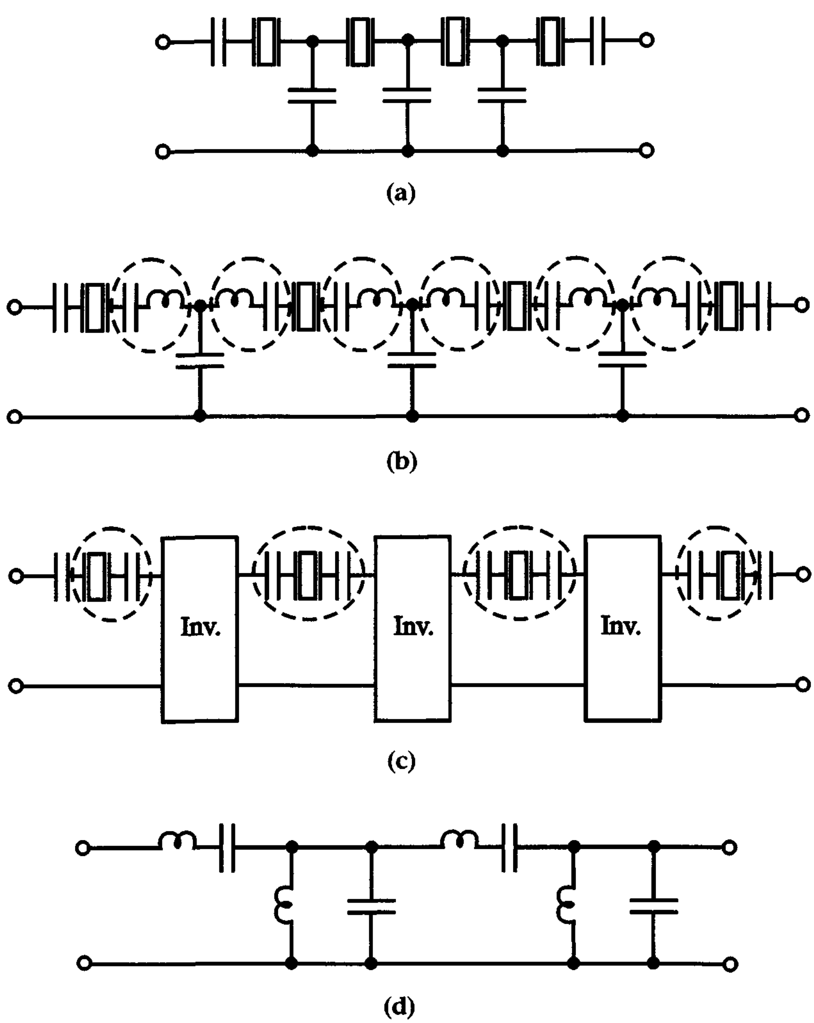


Figure 5.16. Fourth-order Cohn filter used as the IF Filter in the NorCal 40A (a). Adding series LC pairs on each side of the inverter capacitors (b). Redrawing the circuit to show the inverters (c). The equivalent band-pass filter circuit (d).

inductor and a capacitor with the same reactance, there will be no change in the circuit behavior. In Figure 5.16b, we add these series LC pairs on each side of the inverter capacitors. I have enclosed them in dotted lines to indicate that no real components are added and that the behavior of the circuit does not change. Now we associate the inductors with the shunt capacitors to form inverters, and redraw the circuit in Figure 5.16c. This effectively leaves the two inside crystals with two extra series capacitors, and the two outside crystals with one extra capacitor. This is indicated with the dotted lines. Series capacitors increase the resonant frequency, and to make sure that all the crystals have the same resonant frequency, we add a capacitor at each end. Finally we go through the circuit, removing the inverters and swapping series and parallel circuits, one by one. A crystal inverted once becomes a parallel resonant circuit, a crystal inverted twice is a series circuit, and three times, a parallel circuit. We get the equivalent band-pass filter shown in Figure 5.16d.

FURTHER READING

The *ARRL Handbook*, published annually by the American Radio Relay League, has extensive filter tables. This book also has an excellent discussion of quartz crystals. Wes Hayward's book *Radio Frequency Design*, also published by the American Radio Relay League, gives the formulas for the Butterworth and Chebyshev filter components. Useful information on the hyperbolic functions is given in *Tables of Integrals and Other Mathematical Data*, by Herbert Dwight, published by McMillan. For more depth on the mathematics of filter and inverter design, see *Foundations for Microwave Engineering*, by Robert Collin, published by McGraw-Hill.

PROBLEM 13 – HARMONIC FILTER

The Power Amplifier in the NorCal 40A produces a 7-MHz carrier with 2 watts of power. In addition, the amplifier produces a small amount of power at the harmonic frequencies 14, 21, and 28 MHz. Signals at the wrong frequencies are called *spurious emissions*, or spurs for short. Spurs are bad, because they interfere with other radio services. The FCC sets limits on spurs. For HF transmitters with an output of less than 5 W, each spur must be at least 30 dB below the carrier.

The NorCal 40A has a low-pass ladder filter to reduce the harmonics, consisting of the toroidal inductors L7 and L8 and the disk capacitors C45, C46, and C47 (Figure 5.17). The inductors use the same T37-2 cores as the Transmit Filter. These are the cores with red paint. However, they have only 18 turns, and this lets us use thicker wire (#26 instead of #28) to accommodate the large transmitter currents. Start with a 30-cm piece of wire for each core. Solder in the filter components, leaving the C45 leads partly exposed so that you can attach test hooks. Also solder on the BNC Antenna jack J1. The two small pins are the electrical connections, and the two large pins are the mechanical connections. Solder all four pins to the board.

Attach the function generator across C45 with test hooks, making sure the ground clip is connected to the ground lead of the capacitor. Connect the oscilloscope with a

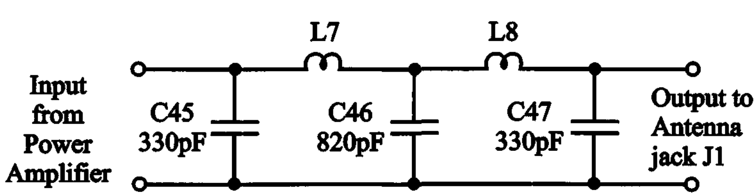


Figure 5.17. NorCal 40A Harmonic Filter.

coaxial cable to the Antenna jack J1. You should use a parallel 50-Ω termination on the scope.

A. Set the function-generator amplitude to 10 Vpp. We do not have a direct measurement of the incident voltage V_{+1}, but it is reasonable to use the amplitude setting on the function generator, 10 Vpp. This makes it convenient to calculate the loss L in dB by the formula

$$L = 20\log(10/V)\,\mathrm{dB}, \tag{5.47}$$

where V is the peak-to-peak output voltage. Measure the output voltage at 7 MHz and 14 MHz, and express L in dB at these frequencies.

B. From the manufacturer's inductance constant, $A_l = 4.0$ nH/turn^2, calculate the inductance of L7 and L8.

C. Now use *Puff* to simulate the filter response from 0 to 28 MHz (the fourth harmonic). Instructions for installing and running the program are given in Appendix C. The design frequency `fd` should be 7 MHz. In the `F2` Plot Window, set up an s_{21} plot to see the loss. 101 points is sufficient. You should choose the y axis carefully so that the curve does not drop off the bottom. Find the loss in dB at 7 and 14 MHz. Make a screen dump.

In addition to reducing the harmonics, the filter sets the load impedance for the Power Amplifier. The output power of amplifiers often varies inversely with the impedance, so that halving the impedance can double the output power. In addition, having a small inductive component often improves the efficiency, by helping the amplifier approach a Class-E operating condition, where little power is lost in switching the transistor on and off.

D. *Puff* allows you to measure the input impedance of the filter conveniently. Plot the reflection coefficient s_{11}, and move the cursor to the s_{11} line in the `F2` Plot Window. Then type =, and the impedance will appear in the Message Box. Find the input impedance of the filter.

E. Assume that we would like to double the output power. You should adjust the components in the filter so that the impedance is cut in half. There are many components that you could change, but to make the problem specific, try varying only L7 and C46. For the capacitor, you should stick to values in the standard 5% series, where the first two digits of the capacitance come from this list: 10, 11, 12, 13, 15, 16, 18, 20, 22, 24, 27, 30, 33, 36, 39, 43, 47, 51, 56, 62, 68, 75, 82, 91. Otherwise you would not be able to buy the capacitors. For the inductor, use only values that you can get by adding or subtracting turns from your cores. What values of L7 and C46 give an impedance closest to half the original impedance?

F. We can improve the harmonic rejection by allowing more ripple. Using the filter table, design a 5th-order, 0.2-dB ripple Chebyshev filter with f_c = 8 MHz. Specify the closest 5% capacitor values and the closest number of turns that you can get with T37-2 cores. Simulate your design with *Puff* and make a plot of $|s_{21}|$. What is the loss in dB at 14 MHz?

PROBLEM 14 - IF FILTER

The IF Filter in the NorCal 40A is a 4-element Cohn filter (Figure 5.18). Study the endpaper to see how this filter is connected in the receiver. The filter uses crystals for microprocessor clocks. These are quite inexpensive, costing only about a dollar, but unfortunately, as they come from the dealer, the resonant frequencies are not nearly close enough together to make a good filter. Wilderness Radio sorts crystals for the NorCal 40A so that they match within 20 Hz. You need six matched crystals in all, four for the IF Filter now, and two for mixer oscillators later.

A. First we measure the resonant frequency of one of the crystals with the setup in Figure 5.19. The function generator should be set to a 4,913,500-Hz sine wave with an amplitude setting of 0.5 Vpp. You should set up the function generator so that you can change the frequency in intervals of 1 Hz. Because the crystals have a series

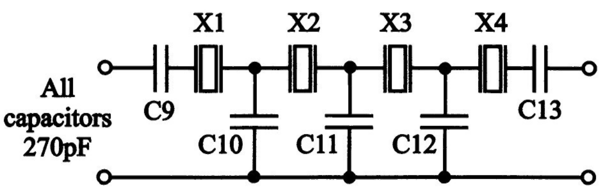


Figure 5.18. The IF Filter in the NorCal 40A.

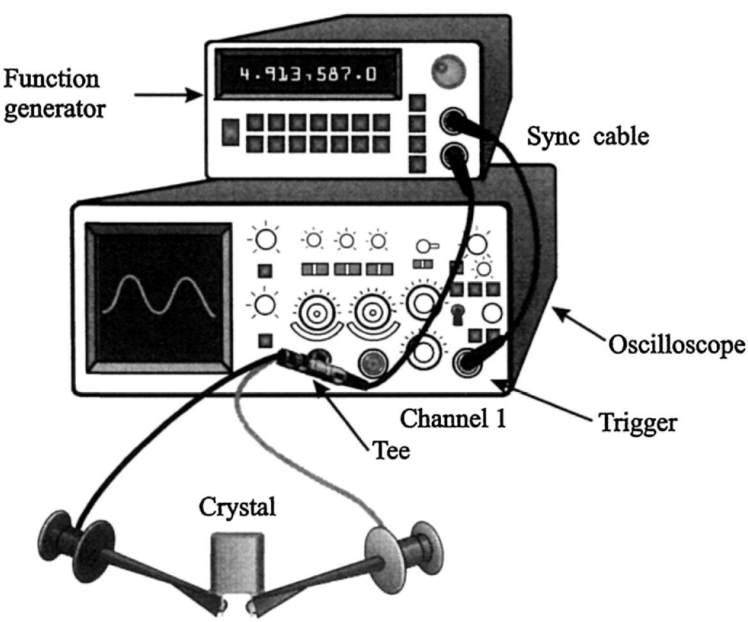


Figure 5.19. Setup for testing crystals.

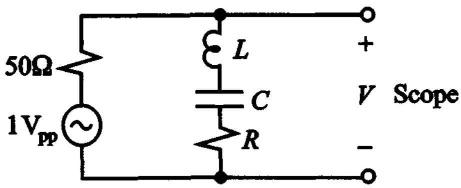


Figure 5.20. Equivalent circuit for the crystal and generator.

resonance, we can recognize the resonant frequency by a dip in the oscilloscope voltage as we vary the frequency. Find the frequency to the nearest hertz that gives the minimum voltage on the scope.

B. Next we will find the components of an equivalent circuit for a crystal, starting with the resistance. Use the equivalent circuit shown in Figure 5.20. Record the output voltage V at resonance and use it to calculate the crystal resistance R.

C. When we shift the frequency off resonance, the scope voltage will increase. Calculate the scope voltage V_x that we would expect when the crystal reactance is equal to R. Notice that this is not simply $\sqrt{2}$ times the minimum voltage, because the crystal resistance is comparable to the resistance of the function generator. Now measure the upper and lower frequencies f_u and f_l that give a scope voltage equal to V_x. Calculate the Q of the crystal from the bandwidth $\Delta f = f_u - f_l$ and the resonant frequency f_0. You need to be careful about the Q here. The crystal Q only includes the resistance of the crystal. It is different from the circuit Q, which also includes the resistance of the generator and is lower because of it. Often people call the crystal Q the *unloaded* Q, and the circuit Q the *loaded* Q.

D. Now calculate the equivalent inductance and capacitance of the crystal. One thing that you need to be careful about here is that we do not have a precise measurement of either L or C individually, but we know their product extremely precisely through the resonant frequency. For one of the components you should use only the number of significant digits that makes sense from your scope measurement, but for the other you will need to use six significant digits, so that the product will give the correct resonant frequency. Check with a calculator that the product of your L and C values gives the resonant frequency correctly to six digits. Otherwise the filter pass band will shift clear off the screen in the *Puff* simulation.

E. Make a model of the Cohn filter with *Puff*, using the equivalent circuit model for the crystal that you have developed and 270-pF capacitors. You should use a range of 2.5 kHz for frequency and 0 to 60 dB for $|s_{21}|$. The design impedance zd should be 200 Ω. Make a plot of $|s_{21}|$.

F. Investigate the effect of changing the port impedance zd to 50 Ω. Make a plot of $|s_{21}|$ and describe the behavior qualitatively.

G. Return the port impedance to 200 Ω, and investigate the effect of changing the capacitors to 200 pF. Make a screen dump and describe the behavior qualitatively.

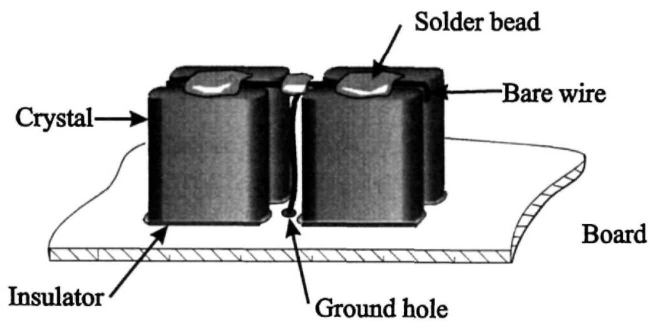


Figure 5.21. Crystal metal cases and the ground connections.

H. In your simulation, return the capacitance to 270 pF. What is the minimum loss in dB in the pass band?

I. One important job of the IF Filter is to reject interference at the upper-sideband frequency, 1,240 Hz above the signal frequency. We hear the upper-sideband frequency as a tone of the same pitch as the signal, and so our ears cannot distinguish the interference from the signal. This is called a *spurious response*. The upper-sideband frequency is a difficult spur to reject, because it is so close to the signal. In the *Puff* simulation, what is the upper-sideband rejection?

Now build the filter. Solder in the 270-pF disk capacitors (C9 through C13). Slide a plastic crystal spacer onto the leads of each of the four crystals, all the way up against the metal case. Now install the filter crystals (X1 through X4) close to the board. The metal cases of the crystals are not connected to the leads, or to any other part of the circuit yet. We say that the cases are *floating*. It is a bad idea to leave large pieces of metal in a circuit floating, because signals can couple capacitively through the metal pieces between different parts of the circuit and end up where you do not want them. To avoid this coupling, we connect each can to ground. There is a small ground hole in the board between the crystals to make this easy. Use bare #22 wire to connect the crystal cans. Figure 5.21 shows how you can do this. Connect the cans with a wire running along the top. It may help to gently bend the cans toward each other until the space between them is small. You should use large solder beads, and make sure that the top of the cans get quite hot so that the solder beads stick well to the cans. If the cases are not hot enough, the wire and solder will pop off the cans. Then solder a wire to the ground hole, hook the other end to the top wire, and solder them together.

The filter is designed for a 200-Ω generator and load. We will add resistors to give the function generator and scope this resistance (Figure 5.22). For the load, solder a 200-Ω resistor from the left L4 hole (connecting to C13 at the filter output) to the left C14 hole, which is a ground connection. Connect the scope across the 200-Ω resistor. The scope connection should be as short as possible, or else the capacitance of the cable will affect the shape of the filter response. The best thing to do is to use a BNC barrel adapter to connect directly to channel 1, and to let the board dangle off the front of the scope. For the function-generator connection, solder one end of a 150-Ω resistor to

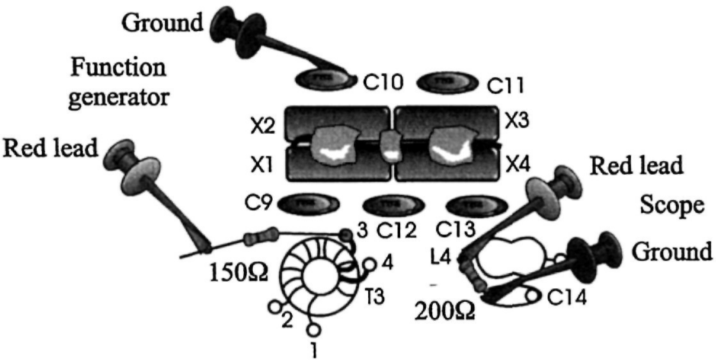


Figure 5.22. Resistor connections to the crystal filter.

the number-3 hole of T3. Attach the function-generator red lead to the other end of the 150-Ω resistor (Figure 5.22). The ground lead can be attached to C10 on the ground side.

J. With an amplitude setting of 0.5 Vpp, measure the minimum loss in dB of the filter to compare with the *Puff* simulation.

K. Next we make a plot of the loss in dB versus frequency. Because we will need to measure very small signals, it is a good idea to switch in a 10-MHz low-pass filter on the oscilloscope if one is available. Much of the noise that blurs the scope trace is at frequencies greater than 10 MHz, and so this will make the trace sharper at low voltage levels. It does, however, reduce the reading somewhat even at 4.9 MHz; thus our plot will be a relative plot. Increase the function-generator amplitude setting to 2.0 V to get a bigger signal. Even though this increases power, it is safe because the power is no longer going into a single crystal, but rather divides between the four crystals and the resistors. Measure the output voltage V over a 2,500-Hz bandwidth centered on the pass band. You should plot the loss L relative to the maximum voltage V_m in dB, by the formula

$$L = 20\log(V_m/V)\,\mathrm{dB}. \tag{5.48}$$

Use a 60-dB scale, with 0 dB at the top. Use judgment in choosing the frequency intervals. Often 50 Hz is a good spacing in the pass band, and 100 Hz is a good spacing in the stop band. You may need to increase the bandwidth beyond 2,500 Hz if the pass band is not centered in your plot. What is the upper-sideband rejection that you measure? When you have finished the plot, remove and discard the two resistors and remove the solder from the holes with solder wick.

After the signal passes through the IF Filter, it goes to the Product Detector, which converts the signal to a 620-Hz audio signal. The product detector is based on an integrated circuit, or IC, made by Philips, the SA602AN. We will have much more to say about the SA602AN later, because it is the most important IC in the transceiver. We use three of them: the Product Detector and the RF Mixer in the receiver and the Transmit Mixer in the transmitter. The SA602AN has a large input impedance, listed in the data sheets as

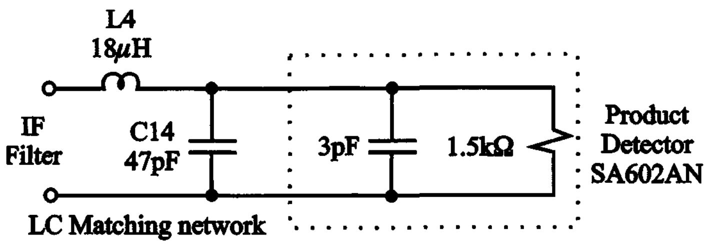


Figure 5.23. LC matching network for connecting the IF Filter to the SA602AN.

1.5 kΩ shunted by 3 pF. This is a bad load impedance for the crystal filter, which should see about 200 Ω. The NorCal 40A has an LC circuit (L4 and C14) that transforms the input of the SA602AN to near 200 Ω (Figure 5.23).

L. Calculate the resistance R and the reactance X that the matching network and the SA602AN present to the IF Filter. Notice that the result is not precisely 200 Ω. Our choice of components is limited to the values that a manufacturer makes. If you could specify any value for L4 and C14, what values would you use to transform the input impedance of the SA602AN to 200 Ω? Solder L4 and C14 into the circuit.