

## Designing Narrow-Bandwidth Ladder Filters

Wes Hayward, w7zoi, April 7, 2019.

**Abstract:** The performance of radio equipment depends upon the filters included in the radio. Especially critical are the narrow width bandpass circuits. This note summarizes the design methods for filters using either LC or quartz crystal resonators. Series and parallel LC circuits are included, as are the crystal LSB and USB ladders. The filter designs are then illustrated with computer simulations.

**A personal aside:** I've spent a lot of time and effort in recent years with LC bandpass filters and lower sideband (LSB) ladder crystal filters. I've not really considered the upper sideband ladder. My goal was to examine this circuit, beginning with the classic paper by Dishal. Rather than merely presenting this work, it seemed to make sense to generalize and to compare and contrast the upper and lower sideband ladders. Some of the simulations illustrate the way coupling and loading might be determined.

My approach to crystal filter design has used a perturbation approach wherein the crystal is approximated by a LC resonator. The general discussion includes the design of the LC filters as well as LSB crystal filters with the perturbation scheme. Some approximations may be used in the filter designs and are always described. However, the analysis, or simulations use no approximations. They are exact, within the constraints of the simple linear models.

None of the work presented here is new. Rather, the note is merely presented as a collected illustration of some of the methods I've found useful.

### Some basic filter characteristics and terms.

What is the meaning of *narrow* in RF filters? A 1 or 2% bandwidth LC filter might be narrow. A 200 Hz wide crystal filter with .004% bandwidth might also be "narrow." Hence, a meaningful definition for *narrow* must include the resonators used to build the filters. A resonator is characterized by an unloaded Q, specified as  $Q_u$ . A bandwidth  $B_{min}$  is related to  $Q_u$ ,  $B_{min}=f/Q_u$  where  $f$  is the resonant frequency.  $B_{min}$  and  $f$  have the same units. It is impossible to build a single resonator passive filter that is narrower than  $B_{min}$ . Filters with more than one resonator are further restricted. An estimated filter minimum will be  $2B_{min}$  for a circuit with 2 resonators. Hence, LC resonators with  $Q_u$  of 250 at 5 MHz could be used to build a double tuned circuit with a bandwidth of 40 kHz. Loss will be high for these *narrowest* filters, with loss decreasing as design bandwidth increases.  $Q_u$  of 100,000 and beyond is common for crystals, so filters with lower bandwidths are possible. The same  $B_{min}$  constraints apply.

This work is based upon so-called *modern network theory* first appearing in the 1950s. Earlier, filters were designed using *image parameter* concepts. Modern network theory has the following characteristics:

1. Designs are low pass filters. These prototypes are characterized by their transfer functions and by a cutoff frequency.
2. The low pass prototype filters are usually described by well defined polynomial transfer functions. These expressions describe an output voltage for a specified input voltage.

3. The low pass prototypes can be transformed to allow arbitrary cutoff frequency and terminations.
4. The prototypes can also be transformed to design high pass, bandpass, and band stop circuits.
5. The low pass prototype filters usually consist of a table of normalized component values. The number of components in the filter is commensurate with the order of the filter, which is the order of the polynomial that is the low pass transfer function. Higher order leads to greater stop band attenuation.
6. The low pass prototype filter can be used to examine the way energy is shared with the terminations and between elements in the prototype. This analysis produces an alternative description of the filter consisting of the Q of end sections and coupling coefficients describing the interface between elements. This new parameter set are end section normalized Q values and normalized coupling coefficients. This new parameter set can be used to design a very wide variety of filter types. Calculation of the “k and q” parameter set is described in an appendix.

The purpose of this paper is to summarize the mathematics and perhaps impart some intuition. Equations are presented with example data. The numeric data will allow the reader to do the calculations as the text is read. Calculations use fundamental units of Hz, Farad, Henry, etc, although we may often use practical components in text, such as capacitors in pF. We assume that the reader will seamlessly make these transformations.

There are no advanced mathematics here. It’s just algebra, although often with imaginary (complex) elements.

After filters are designed, they are analyzed with computer programs. Some of the analysis is with programs we have written while others use SPICE. This report does not include measured results. An appendix presents more numeric detail for readers wishing to see such data. Generally, we obtain excellent correspondence between filter simulation and physical measurements.

Two filter orders (N=2 and N=4) are considered for each of the four resonator types. The reader should then be able to extend the presentation to any order. The key to this flexibility is found in the *central design rules* in section 2 below.

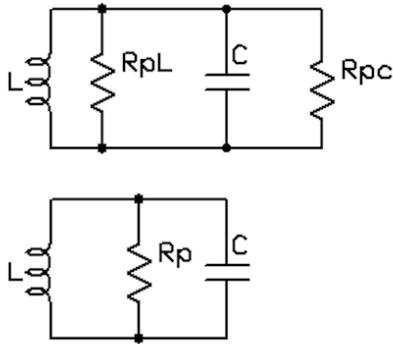
## Contents.

The paper is divided into the following sections:

1. A discussion of resonators and tuning.
2. The central design rules.
3. Terminations that load resonators. Deriving the equations.
4. A double tuned circuit with parallel LC circuits.
5. A bandpass with 4 parallel LC tuned circuits.
6. A double tuned circuit with series LC resonators.
7. A N=4 bandpass with series LC resonators.
8. A crystal filter of the LSB Ladder type with two crystals.
9. A LSB Ladder crystal filter with N=4.
10. A USB Ladder crystal filter with two crystals.
11. A USB Ladder crystal filter with N=4.
12. A Crystal Notch Filter, N=3.
13. Bibliography.
14. Appendix A, numeric examples.
15. Appendix B, converting prototype components to k and q.

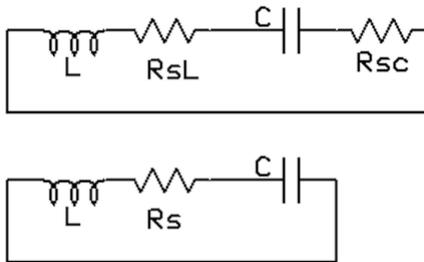
## 1. Basic Resonator Circuits.

This section presents some schematic diagrams and some single stage filters. There are no mathematics here, although the math to be presented later can be adapted to analyze these simple filters.



**Fig 1-1.**

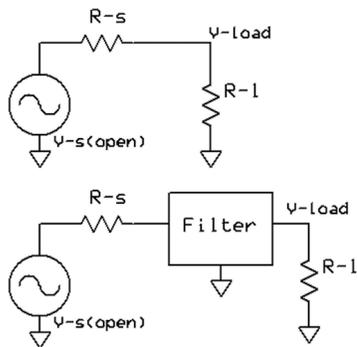
A parallel tuned circuit places an inductor in parallel with a capacitor. Both elements have their own loss, each represented here by a parallel resistance. More often than not, we combine the two to form a single resonator parallel resistance,  $R_p$ , shown in the second part of Fig 1-1.



**Fig 1-2.**

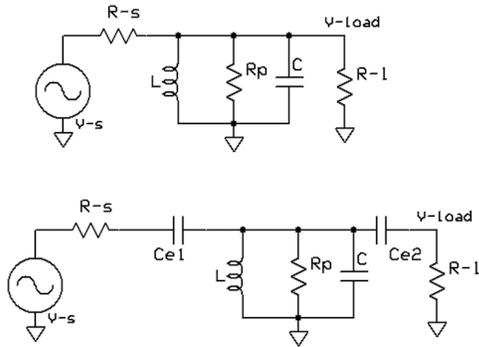
A series tuned circuit is shown in Fig 1-2. Again, the series loss resistances that model the individual elements are combined in a single resistance,  $R_s$ , that models the complete series tuned circuit shown in the lower part of Fig 1-2.

Fig 1-3 models the way a filter might be used in a system.



**Fig 1-3.**

The upper part of Fig 1-3 shows a source with internal resistance  $R-s$  followed by a load  $R-l$ . The source has an open circuit voltage of  $V-s$ . A filter circuit is inserted between the source and load, shown in the lower part of the figure.

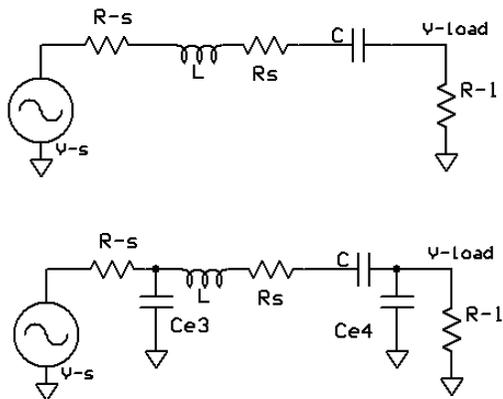


**Fig 1-4.**

Two variations are shown in Fig 1-4 for using a **parallel** tuned circuit as a filter. In the upper case, the reactive L and C must be carefully chosen with respect to the source and load. L and C must resonate at the filter center frequency. But the L (and hence, also the C) will be chosen to interact with the source and load resistances as well as the Rp which models resonator loss to establish a useful bandwidth. Consider an example. Source and load R are both set at 50 Ohms. A small inductor of 1  $\mu$ H resonates at 5 MHz with 1013.2 pF. Assume, for the moment, that loss element Rp is very high and can be ignored. The source and load are both 50 Ohms, so their parallel value is 25 Ohms. The inductive reactance of L at 5 MHz is 31.4 Ohms. The loaded Q of this circuit is then 25/31.4, which is less than 1. The heavy loading comes from the terminations. This is a very wide filter, but still useful in some situations. This filter bandwidth will be slightly more than the center frequency.

The second part of Fig 1-4 shows a variation where series end capacitors Ce1 and Ce2 are placed in series with the source and load resistances. A careful capacitor choice produces whatever loading we wish. This will then turn the filter into something with a much narrower bandwidth. But the end capacitors will detune the original resonator from the initial value, so resonator C must be altered to maintain the original frequency.

If we allow the resonator loss resistance, Rp, to take on practical values, we will see that this single parallel tuned circuit filter will have an insertion loss. This can be calculated with methods that will follow.



**Fig 1-5.**

The **series** tuned circuit form of a N=1 filter is shown in Fig 1-5. (N is the number of resonators.) The upper half shows the source and load resistances in series with the reactive elements and series loss modeling resistance R-s. In this variation, the higher the loading resistances R-s and R-l become, the lower the Q and the wider the circuit bandwidth. The lower half of Fig 1-5 shows the common modification where shunt capacitors, Ce3 and Ce4, transform the usual terminations to “look like” lower values when considering loads for the resonator. The equations for this transformation are presented later in Section 3.

## 2. The Central Design Rules for Coupled-Resonator Filters.

Modern network theory was mentioned earlier. The low pass circuits that are the basis for our coupled-resonator filters are all derived from characteristic polynomials. For example, a Butterworth polynomial describes a low pass that is maximally flat at the origin, frequency=0 for a low pass. (By maximally flat, we mean that the frequency derivative of the transfer function is zero at the origin.) Another popular polynomial is that of Chebyshev, a shape that results from an approximation that asks for the lowest possible response in the stopband, but allows an equal ripple within the passband. Filter enthusiasts refer to this as a “brick wall” shape. These details are discussed in the references. There are many other interesting and useful polynomials that are used. Once a polynomial has been chosen, it will define the shape of the filter response. It will also define a set of normalized low pass component values.

The low pass prototype component values can be used directly to design low pass, high pass, and even some bandpass filters. But most narrow band filter designs use an alternative set of filter design parameters, the so called normalized end section  $q$  and coupling coefficients. For example, a Butterworth bandpass filter with four resonators (order  $N=4$ ) would have the following parameters:

$$q_1 = 0.7654, \quad q_4 = 0.7654, \quad k_{12} = 0.841, \quad k_{23} = 0.541, \quad k_{34} = 0.841.$$

Every filter shape and order will have a corresponding set of parameters. We emphasize that these are not different filters from the normalized low pass parameters, but are just a new set of values that are derived from the low pass elements.

So, what do these parameters represent?

First, these parameters are *normalized*. This means that the most common low pass filter that is the basis for a bandpass is a circuit with a 1 radian per second cutoff and one ohm terminations. A  $N=4$  Butterworth low pass would have the normalized values of  $C_1=0.7654$  Farad,  $L_2=1.85$  Henry,  $C_3=1.85$  Farad, and  $L_4=0.7654$  Henry. These normalized component values are converted to the  $k$  and  $q$  values shown above. (See Appendix B.) Or the normalized  $L$  and  $C$  low pass values can be denormalized to form low pass or high pass filters.

Denormalizing the  $k$  and  $q$  occurs with respect to a parameter called *filter Q*, or  $Q_f$ . Assume that we wanted to build a filter with a 5 MHz center frequency and a bandwidth of 0.2 MHz.  $Q_f$  is then  $5/0.2=25$ . The other often used parameter is *nodal capacitance*,  $C_0$ . This is the capacitance value for the LC resonator forming the basis of the filter. The details of the denormalization will emerge in the later sections.

The normalized  $q$  is a value related to an input or output resonator. Once the parameter is denormalized, a loaded  $Q$  for the resonator is established. This is the  $Q$  of that resonator when loaded by the termination at the related end. Recall Fig 1-3 where an arbitrary filter was terminated at both input and output by source and load resistances. The end section  $Q$  for the first resonator is the result of loading by  $R_s$ , the source.

The normalized coupling coefficients have a similar role. The parameter  $k_{12}$ , when denormalized, will specify the element that couples energy from the first resonator to the second.

The central rule for filter design is then: **A filter shape is established when the end section Q values and the couplings between resonators are established. Passband shape does not depend upon how we realize the coupling and loading. Many different circuits may be used, but once the denormalized K and Q values are established, the filter shape close to the center frequency is determined.** This rule will be illustrated in the sections that follow.

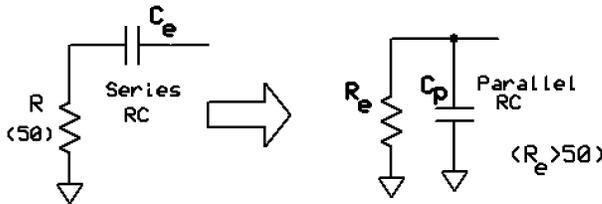
## 3. Terminations that Load Resonators, Series to Parallel and Back...

The central rules stipulate that we must establish a specific end section  $Q$  to realize a desired filter shape. This section will derive the important equations that are used in later designs to obtain desired loading

values. The equations presented are essentially transformations between parallel and series circuits. 5 MHz examples are used. Some capacitor values will be specified, but these are merely used for the numerical calculations. The important *outputs* of this section are the formulas. The initial parameters for examples are:

$$f = 5 \cdot 10^6, \omega = 2\pi f, C = 10^{-10}, R = 50, c = 500 \cdot 10^{-12}$$

Two capacitor values, C and c, are specified. We will use C to deal with a *series* circuit of R and C. Later, a *parallel* R and c will be analyzed. We begin with the impedance of the series RC of Fig 3-1.



**Fig 3-1.**  $C_e=100$  pF at 5 MHz and  $R=50$ .

$$Z = R - \frac{i}{\omega C}$$

$$Z = 50 - 318.31i \quad \text{Eq. 3-1.}$$

Lower case i is the square root of minus 1. This equation expands to yield

$$Z = \frac{(R\omega C - i)}{\omega C}, Y = \frac{1}{Z}, Y = \frac{\omega C}{(R\omega C - i)} \quad \text{Eq. 3-2.}$$

We have converted the impedance Z to an admittance, Y. Continuing, Y is resolved into real and imaginary parts, G and B,

$$Y = \frac{\omega C (R\omega C + i)}{(R\omega C - i)(R\omega C + i)} = \frac{R\omega^2 C^2 + i\omega C}{(R\omega C)^2 + 1} = G + iB \quad \text{Eq. 3-3.}$$

G is a conductance while B is a susceptance. Further expansion yields

$$G = \frac{(R\omega^2 C^2)}{(R\omega C)^2 + 1}, B = \frac{\omega C}{(R\omega C)^2 + 1} \quad \text{Eq. 3-4.}$$

G may be inverted to obtain an equivalent parallel resistance. If we were to write the susceptance of a parallel capacitor, it would be  $B = \omega C_p$  where  $C_p$  is the parallel capacitance and  $\omega$  is the angular frequency. Hence,  $C_p = B / \omega$ . The resulting equations are

$$R_p = \frac{(R\omega C)^2 + 1}{(R\omega^2 C^2)}, C_p = \frac{C}{(R\omega C)^2 + 1} \quad \text{Eq. 3-5.}$$

Recall that  $R=50$  and  $C$  was 100 pF, resulting in 5 MHz calculated results

$$R_p = 2076.42$$

$$C_p = 9.7592e-11$$

Eq. 3-6.

The series capacitor has readily created a resistance much higher than the original 50 Ohms. The equivalent parallel capacitance  $C_p$  is close to, but still less than the original 100 pF.

We rewrite Eq 3-5 by defining the parallel resistance to be  $R_e$ . The subscript e emphasizes that this is an **end** resistance.

$$R_e = \frac{(R\omega C)^2 + 1}{R\omega^2 C^2}, X = \frac{1}{\omega C}$$

Eq. 3-7.

Note that Eq. 3-7 includes a definition for the reactance of the series capacitor. This becomes useful, leading to

$$RR_e = \frac{\frac{R^2}{X^2} + 1}{\frac{1}{X^2}}$$

$$RR_e = R^2 + X^2$$

Eq. 3-8.

This heads to the simple form.

$$X = \sqrt{RR_e - R^2}$$

Eq. 3-9.

Extracting the value of the capacitor produces

$$C_{end} = \frac{1}{\omega \sqrt{RR_{end} - R^2}}$$

Eq. 3-10.

For example, say that we needed a resistance of 3000 Ohms across the resonator to achieve the desired end section Q, but we had  $R=50$  at the other end. The required capacitor from Eq. 3-10 is then

$$R_{end} = 3000$$

$$C_{end} = 8.28808e-11$$

Eq. 3-11.

We also need to know the equivalent parallel capacitance across the newly generated 3000 Ohms. This is given by Eq. 3-5, repeated here

$$C_{par} = \frac{C_{end}}{(R\omega C_{end})^2 + 1}$$

$$C_{par} = 8.14995e-11$$

Eq. 3-12.

The shunt 81.5 pF is close to, but not quite equal to the series end capacitance, 82.9 pF.

In the analysis above we were able to create a large resistance load with a series capacitor. Next we consider the case of adding a parallel capacitor to a load to generate the equivalent of a lower resistance. This is shown in the next figure. Lower case c represents capacitors merely to distinguish from the earlier case.

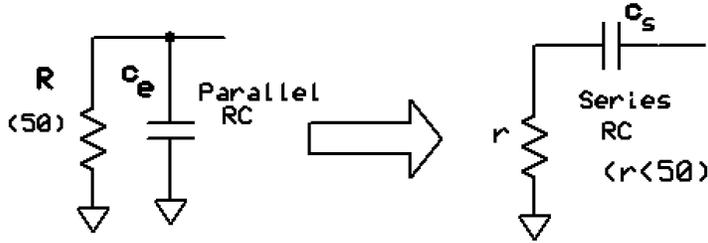


Fig 3-2.  $R=50$  and  $c_e=c=500$  pF for the numeric example.

We begin by writing the admittance for the parallel RC.

$$y = \frac{1}{R} + i\omega c$$

$$y = 0.02 + 0.015708i \quad \text{Eq. 3-13.}$$

But impedance is just  $Z=1/Y$ ,

$$z = \frac{R}{1 + i\omega Rc} = \frac{R(1 - i\omega Rc)}{1 + (\omega Rc)^2} \quad \text{Eq. 3-14.}$$

The real part of  $z$  is just the end resistance that will load the end resonator,

$$r = \frac{R}{1 + (\omega Rc)^2} \quad \text{Eq. 3-15.}$$

But this is not the form we want. Rather, we wish to specify a value for  $r < R$  and then calculate the value of  $c$  that will yield that result. We solve Eq. 3-15 for  $c$  with the following sequence,

$$r + r(\omega Rc)^2 = R \quad \text{Eq. 3-16.}$$

$$\omega Rc = \sqrt{\frac{R-r}{r}} \quad \text{Eq. 3-17.}$$

$$c = \frac{1}{\omega R} \sqrt{\frac{R-r}{r}} \quad \text{Eq. 3-18.}$$

Cleaning things a little yields a “standard form,”

$$c_e = \frac{1}{\omega R_0} \sqrt{\frac{R_0 - r}{r}} \quad \text{Eq. 3-19.}$$

This value of parallel end capacitor gives us the required end resistance. We seek the equivalent series capacitance that will contribute to resonator tuning. We focus on the imaginary part of Eq. 3-14,

$$X = \frac{\omega R^2 c}{1 + (\omega R c)^2} = \frac{1}{\omega c_s} \quad \text{Eq. 3-20.}$$

$$\frac{\omega^2 R^2 c}{1 + (\omega R c)^2} = \frac{1}{c_s} \quad \text{Eq. 3-21.}$$

Solving for  $c_s$  while inserting an “e” subscript for later clarity yields

$$c_s = \frac{1 + (\omega R c_e)^2}{\omega^2 R^2 c_e} \quad \text{Eq. 3-22.}$$

Finally, consider an example where we wish to generate a 5 ohm load with a capacitor and 50 ohm termination.

$$r = 5, R_0 = 50$$

$$c_e = 1.90986e-09$$

$$c_s = 2.12207e-09$$

$$\text{Eq. 3-23.}$$

These transformations will be applied in the filter designs that follow.

#### 4. A Double Tuned Circuit with Parallel Tuned LC Resonators.

Now that some of the preliminary details are out of the way, it’s time to design a filter. Our first choice is a 200 kHz wide double tuned circuit at 5 MHz using parallel resonators. The schematic follows.

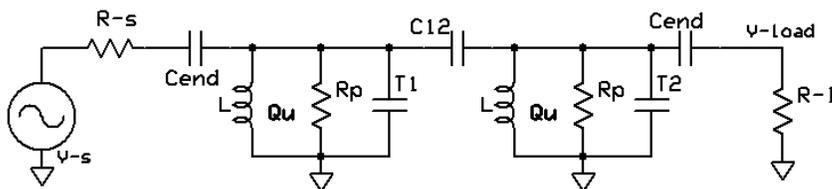


Fig 4-1.

The parameters for our filter are

$$k_{12} = \frac{1}{\sqrt{2}}, q = \sqrt{2}, f_c = 5 \cdot 10^6, Q_u = 200, L = 3 \cdot 10^{-6}, R_0 = 50, bw = 200 \cdot 10^3$$

The Butterworth parameters are the  $k_{12}$  and  $q$  values shown while the center frequency is 5 MHz. We pick inductors of 3  $\mu$ H with an unloaded  $Q$  of 200. The bandwidth was specified to be 200 kHz and the filter source and load will be 50 Ohms. The angular frequency is

$$\omega = 2\pi f_c \quad \text{Eq. 4-1.}$$

The nodal capacitance  $C_0$  is the capacitance that resonates with the inductor that we have picked.

$$C_0 = \frac{1}{\omega^2 L} \quad \text{Eq. 4-2}$$

The filter Q is given by

$$Q_F = \frac{f_c}{\text{bw}} \quad \text{Eq. 4-3.}$$

This filter Q is relatively low compared with the unloaded resonator Q, so this filter will be easily realized. A useful parameter is one we term the “normalized Q,” signified by  $q_0$  and defined as

$$q_0 = \frac{Q_u}{Q_F} \quad \text{Eq. 4-4.}$$

If  $q_0$  is more than 2, a double tuned circuit is usually practical. Later we will see that  $q_0$  can be used to calculate insertion loss.

The end Q is

$$Q_{end} = qQ_F \quad \text{Eq 4-5}$$

Note that the normalized filter q is denormalized by multiplying by filter Q,  $Q_F$ . If the bandwidth was higher,  $Q_F$  would be lower and  $Q_{end}$  would be lower. Equation 4-5 is a first approximation and it does not account for the intrinsic loss in the resonator specified by  $Q_u$ . A more complete expression is

$$Q_e = \frac{1}{\left(\frac{1}{qQ_F} - \frac{1}{Q_u}\right)} \quad \text{Eq. 4-6.}$$

A coupling coefficient for a practical filter element is the normalized value divided by filter Q. The coupling capacitor is then the nodal capacitance multiplied by the coupling coefficient. Hence, the coupling capacitor is calculated with

$$C_{12} = C_0 \frac{k_{12}}{Q_F} \quad \text{Eq. 4-7.}$$

We have already calculated the denormalized end section  $Q = Q_e$ . This allows us to find the end resistance needed to properly load the resonator.

$$R_{pe} = Q_e \omega L \quad \text{Eq. 4-8.}$$

A series capacitor  $C_e$  is placed in series with the 50 ohm loads at each end of the filter to transform the low resistance to look like  $R_{pe}$  at the resonator. This calculation uses Eq. 3-10 derived above,

$$C_e = \frac{1}{\omega \sqrt{R_{pe} R_0 - R_0^2}} \quad \text{Eq. 4-9.}$$

The final detail is resonator tuning. We started with a nodal  $C_0=338$  pF, that has been diminished because of the ~10 pF capacitor coupling between resonators and by the 71 pF loading capacitors. We tune the resonators with

$$T_1 = C_0 - C_{12} - C_e \quad \text{Eq. 4-10.}$$

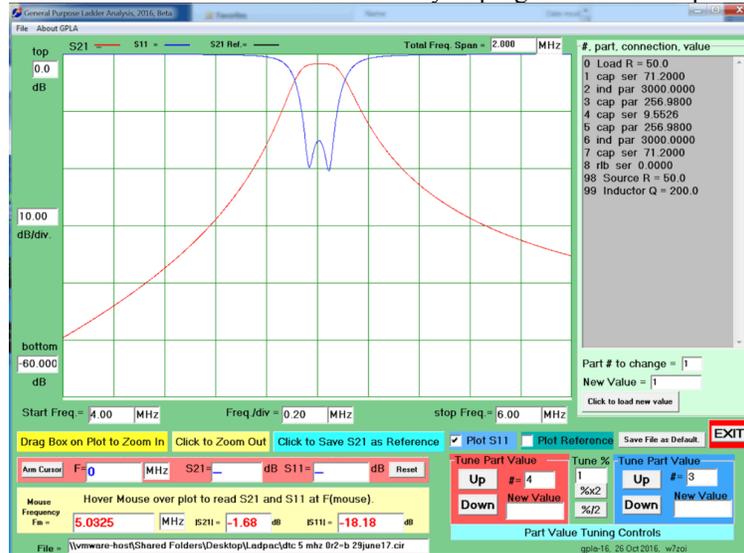
Note that this is still an approximation. A more refined equation was derived above (Eq. 3-12) if greater accuracy is needed. It would not make much difference in this case. Moreover, we would probably use trimmer capacitors as part of the resonators in a practical circuit, so a small approximation is allowed.

Finally, we calculate the filter insertion loss.

$$IL = 20 \log \left( \frac{q_o}{q_o - q} \right) \quad \text{Eq. 4-11.}$$

The loss depends upon the normalized Q,  $q_o$ , as well as the q value, here the end Butterworth q. The insertion loss is in dB.

A computer simulation of this filter is shown below. This is an analysis using the *ladder* method, outlined in our IRFD text book. This is an analysis program that is independent of the design above.



**Fig 4-2.** An analysis of the 5 MHz double tuned circuit with parallel resonators. Note that the insertion loss corresponds to the value from Eq. 4-11.

## 5. N=4 Bandpass Filter with Parallel LC Tuned Circuits.

The next filter is just like the earlier double tuned circuit, but now contains four parallel resonators. The same methods are used for end loading and for coupling as we used with the simpler double tuned circuit of section 4. The approximation used for tuning with the previous double tuned circuit is replaced with a better method. Again, we use Butterworth k and q parameters. The set-up data and equations are presented below. We picked a 6  $\mu$ H inductor for this 5 MHz filter, but that is open to the designer.

$$f_c = 5, bw = 0.2, \omega = 2\pi f_c 10^6, Q_u = 200, R_0 = 50, L = 6 \cdot 10^{-6}$$

$$k_{12} = 0.841, k_{23} = 0.541, k_{34} = k_{12}, q = 0.7654$$

Next, we calculate the nodal capacitance and the filter Q:

$$C_0 = \frac{1}{\omega^2 L} \quad \text{Eq. 5-1.} \quad Q_F = \frac{f_c}{bw} \quad \text{Eq. 5-2.}$$

The end Q is calculated from the filter Q,

$$Q_{end} = \frac{1}{\left( \frac{1}{qQ_F} - \frac{1}{Q_u} \right)} \quad \text{Eq. 5-3.}$$

This calculation includes both the normalized Butterworth end  $q$  and the unloaded resonator  $Q_u$ .

The next calculations are for the coupling capacitors.

$$C_{12} = C_0 \frac{k_{12}}{Q_F} \quad \text{Eq. 5-4.} \quad C_{23} = C_0 \frac{k_{23}}{Q_F} \quad \text{Eq. 5-5.} \quad C_{34} = C_0 \frac{k_{34}}{Q_F} \quad \text{Eq. 5-6.}$$

Note the symmetry with  $k_{12}=k_{34}$ , leading to  $C_{12}=C_{34}$ . This is a common occurrence with Butterworth and Chebyshev filters.

Having calculated  $Q_{end}$ , we can now continue with the end loading determinations.

$$R_{pe} = Q_{end} \omega L \quad \text{Eq. 5-7.} \quad C_e = \frac{1}{\omega \sqrt{R_{pe} R_0 - R_0^2}} \quad \text{Eq. 5-8.}$$

This gives us the capacitance value needed at the filter end to load resonators 1 and 4 with 50 Ohms. But the parallel capacitance that is presented to these resonators must be determined before tuning. The equation for this was derived in section 3. The parallel capacitance is given by

$$C_p = \frac{C_e}{1 + (R_0 \omega C_e)^2} \quad \text{Eq. 5-9.}$$

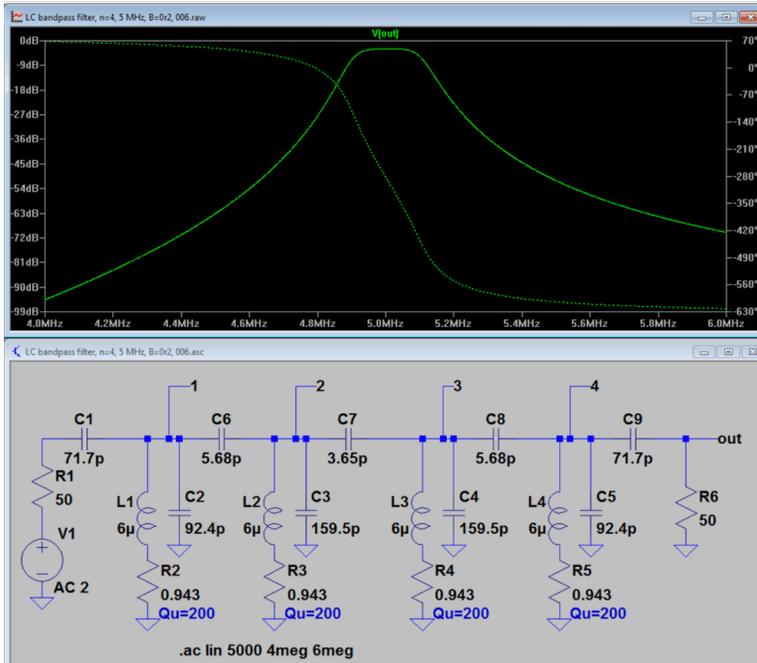
This capacitance, 70.8 pF, is close to the end  $C$  value of 71.7 pF. This is the usual situation when  $C_e$  generates a large impedance transformation. In this case we go from 50 to almost 4000 Ohms. The capacitors are not as close to each other with a smaller transformation, which we might have with a wider bandwidth filter. We can now tune the filter.

$$T_1 = C_0 - C_{12} - C_p \quad \text{Eq. 5-10} \quad T_2 = C_0 - C_{12} - C_{23} \quad \text{Eq. 5-11}$$

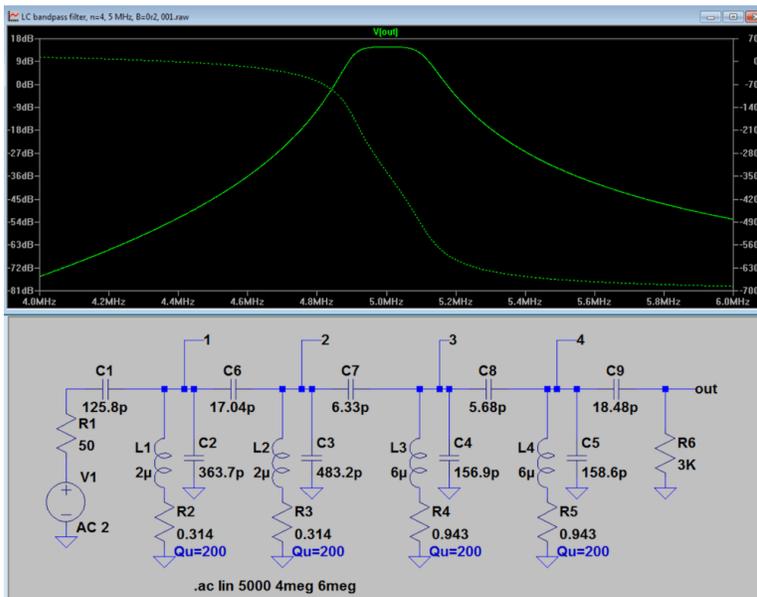
$$T_3 = T_2 \quad \text{Eq. 5-12} \quad T_4 = T_1 \quad \text{Eq. 4-13}$$

We again take advantage of the symmetry that comes from the equally terminated Butterworth filter. Even without this symmetry, the formulas for  $T_3$  and  $T_4$  should be obvious.

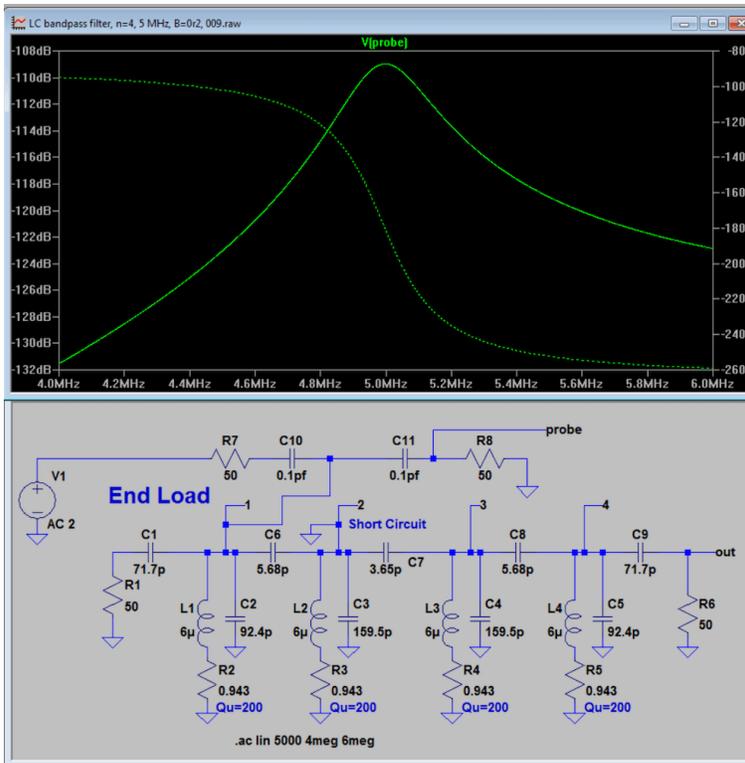
Figure 5-1 below contains the transfer function for the filter we just designed. The schematic is also included. This is an LT-SPICE analysis rather than one using our "homebrew" software. Filter analysis is, after all, divorced from the filter design.



**Fig 5-1.** The classic Butterworth shape is evident. A simulation must be used to evaluate the insertion loss, for we lack a simple formula like we had with the double tuned circuit. Series resistors have been included with the inductors in the simulation to represent the unloaded Q of 200. Alternatively, parallel resistors could be used.



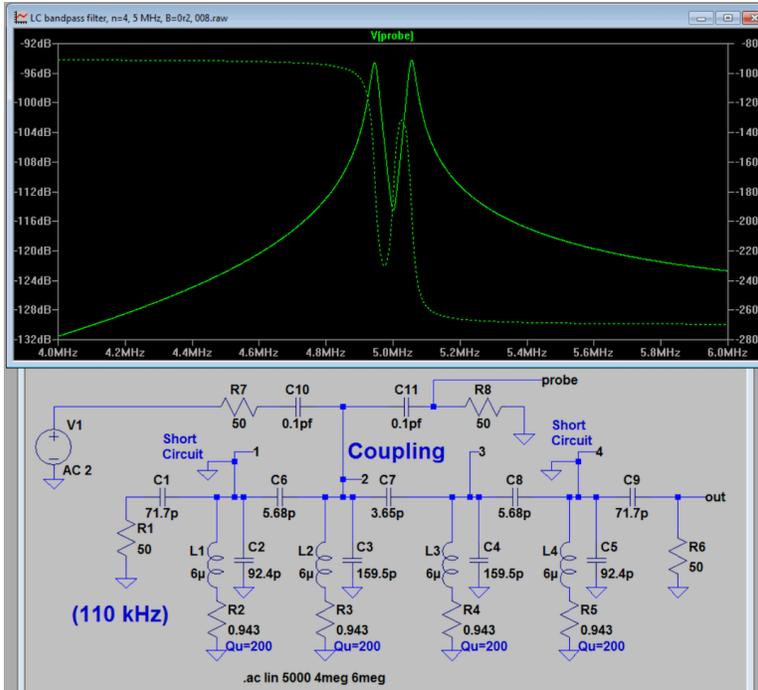
**Fig 5-2.** This filter is a special case, although this is not evident without careful inspection. There are two special features. First, the output resonator is terminated in 3K rather than 50 Ohms. Note that the filter performance is essentially unchanged, for the end section Q has been maintained. The second feature is the use of unequal inductors. Resonators 1 and 2 use 2  $\mu$ H while #3 and #4 use 6  $\mu$ H. The end calculations preserve end section Q. The coupling between resonators 2 and 3 is the unusual case owing to unequal inductors. The calculation is still straight forward. The nodal C values are calculated. The geometric mean of the two  $C_0$  values is evaluated and used to calculate the coupling capacitor, C7 in the above figure.



**Fig 5-3.** This circuit and plot shows how the same simulation, with just a modification or two, can be used to evaluate the end Q of a loaded resonator. Note that resonator 2 is short circuited. Hence, the coupling capacitor C6 in the above circuit continues to be part of the overall C to tune resonator 1, but all details related to resonators 2, 3, and 4 disappear. The end loading cap is attached to a termination, but the resonator is not excited with  $C_{end}$ . Two “probes” are attached only to the first resonator through VERY small capacitors, 0.1 pF. One probe comes from the RF source while the other goes to a detector. The sweep serves to measure the Q of the resonator. If we did not have the 50 ohm load and  $C_{end}$ , we would see a very narrow peak. This could be used to actually measure unloaded resonator Q. But the resonator is loaded to  $Q_{end}$  which is the value we would obtain from the swept response shown.

This computer simulation used a 50 ohm load and another 50 ohm detector, attached via very small capacitors, suggestive of the scheme we might use in a physical measurement. For a pure computer study, a current source could drive the resonator while the voltage is merely noted and plotted. Neither element will load the circuit. My personal preference is to emulate the physical measurement, but either method will provide the needed information.

The scheme is one that is often used for physical measurements. The very small capacitors in the simulation can be approximated with any manner of small “real world” capacitor. The scheme used depends upon the frequency. The key is to use identical “caps” at both probes and to guarantee that there is an insertion loss of at least 30 dB through the measurement setup. If these criterion are met, it should be a good measurement. Note that this “rule” can be tested with computer simulation. (This is NOT lore.)



**Fig 5-4.** This simulation shows how we might measure coupling between resonators. The setup is just like that used above to measure the loaded Q of an end section. A resonator is excited with a probe and another is used to measure the voltage across the same tuned circuit. In this example, the two probes are attached to resonator #2. We have short circuited resonators #1 and #4, so the tuning of resonators 2 and 3 is preserved. Sweeping the system of two probes attached to #2 shows the coupling from 2 to 3, indicated by a double hump response. If coupling capacitor C7 is increased, the separation between peaks will increase. This behavior is specified by

$$\delta f_{23} = k_{23} \text{bw} \quad \text{Eq. 5-14}$$

The frequency spacing  $\delta f_{23}$  is related to the filter bandwidth and the normalized coupling coefficient,  $k_{23}$ . When we zoomed in to do a careful look at the peaks, we confirmed this calculated result. Equation 5-14 can be used to measure a coupling coefficient.

The methods presented here are very powerful, for they show how computer simulation can be used to study circuit behavior and fundamental concepts. But this is not a *measurement*. An actual physical measurement will often take less time, but provide more information. The physical examination includes the effects of stray L and C as well as loss. When dealing with filters, there are three levels to the problem: design or synthesis, analysis or simulation, and physical measurement. The best filter circuits should have good correspondence between all three. Don't skip any of the steps. Especially avoid the trap of assuming that a computer simulation can erase the need for physical measurement.

## 6. Double Tuned Circuit using Series Resonators

The next design example is for a DTC using series tuned circuits. The schematic is shown below.

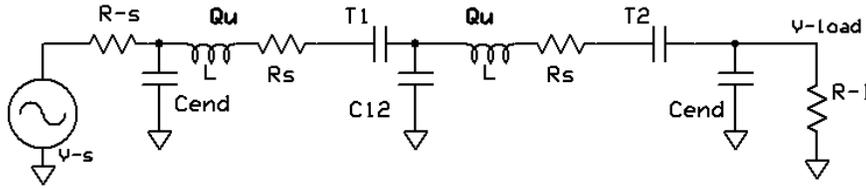


Fig 6-1.

The set up details are given with

$$f_c = 5, BW = 0.2, \omega = 2\pi f_c 10^6, Q_u = 200, R_0 = 50, L = 7 \cdot 10^{-6}, k_{12} = \frac{1}{\sqrt{2}}, q = \sqrt{2}$$

We continue to design filters with a 200 kHz bandwidth at 5 MHz with unloaded resonator Q of 200. The same Butterworth coefficients are used. This example uses 7  $\mu$ H inductors. The design begins with calculation of the nodal capacitance,

$$C_0 = \frac{1}{\omega^2 L} \quad \text{Eq. 6-1.}$$

Next, we calculate the coupling capacitor and the end section Q. The coupling cap is now a large valued shunt element.

$$C_{12} = \frac{C_0}{k_{12} \frac{BW}{f_c}} \quad \text{Eq. 6-2.} \quad Q_e = \frac{1}{\left(\frac{BW}{q f_c} - \frac{1}{Q_u}\right)} \quad \text{Eq. 6-3.} \quad R_s = \omega \frac{L}{Q_e} \quad \text{Eq. 6-4.}$$

This low value series R is realized from a 50 ohm termination with a shunt end capacitor,

$$C_{end} = \sqrt{\frac{(R_0 - R_s)}{R_s \omega^2 R_0^2}} \quad \text{Eq. 6-5.}$$

While this value generates the transformation, this is not the series capacitor that results from a parallel to series conversion. The equivalent series value is

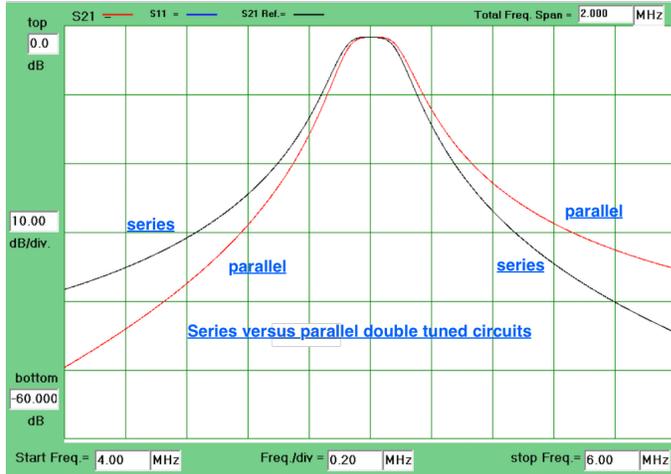
$$C_{es} = \frac{C_{end}^2 \omega^2 R_0^2 + 1}{C_{end} \omega^2 R_0^2} \quad \text{Eq. 6-6.} \quad \text{These equations were derived in Section 3.}$$

This  $C_{es}$  value allows tuning of the first resonator with T1.

$$T_1 = \frac{1}{\left(\frac{1}{C_0} - \frac{1}{C_{es}} - \frac{1}{C_{12}}\right)} \quad \text{Eq. 6-7.}$$

The second resonator is identical to the first one.

The response of the double tuned circuit with a series LC is almost identical to the one using parallel circuits. The one with the series tuned circuits will drop off a little faster on the high frequency side. The two are compared below.



**Fig 6-2.** Comparison between DTCs using series versus parallel tuned circuits. Both filters have a 200 kHz bandwidth centered at 5 MHz with inductors having an unloaded Q of 200. The insertion loss is the same for both cases.

## 7. N=4 Bandpass Filter with Series LC Tuned Circuits.

Having designed a DTC with series resonators, we now consider a higher order circuit with four resonators. The “set-up” details follow. This example uses 12  $\mu\text{H}$  inductors and retains the Butterworth shape.

$$f = 5, \text{BW} = 0.2, \omega = 2\pi f 10^6, Q_u = 200, R_0 = 50, L = 12 \cdot 10^{-6}$$

$$k_{12} = 0.841, k_{23} = 0.541, k_{34} = k_{12}, q = 0.7654$$

Design begins by calculating the nodal capacitance and then the three coupling values.

$$C_0 = \frac{1}{\omega^2 L} \quad \text{Eq. 7-1}$$

$$C_{12} = \frac{C_0}{k_{12} \frac{\text{BW}}{f}} \quad \text{Eq. 7-2}$$

$$C_{23} = \frac{C_0}{k_{23} \frac{\text{BW}}{f}} \quad \text{Eq. 7-3}$$

$$C_{34} = \frac{C_0}{k_{34} \frac{\text{BW}}{f}} \quad \text{Eq. 7-4}$$

Next we calculate the end Q and the related series resistance.

$$Q_e = \frac{1}{\left(\frac{\text{BW}}{qf} - \frac{1}{Q_u}\right)} \quad \text{Eq. 7-5}$$

$$R_s = \omega \frac{L}{Q_e} \quad \text{Eq. 7-6}$$

The end resistance is generated from a 50 ohm termination with a shunt end capacitance, with the related series capacitance needed to calculate tuning.

$$C_{end} = \sqrt{\frac{(R_0 - R_s)}{R_s \omega^2 R_0^2}} \quad \text{Eq. 7-7}$$

$$C_{es} = \frac{C_{end}^2 \omega^2 R_0^2 + 1}{C_{end} \omega^2 R_0^2} \quad \text{Eq. 7-8}$$

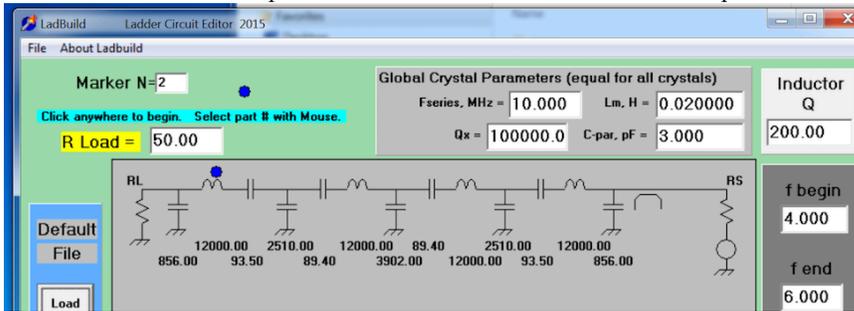
We can now tune the filter.

$$T_1 = \frac{1}{\frac{1}{C_0} - \frac{1}{C_{es}} - \frac{1}{C_{12}}} \quad \text{Eq. 7-9.}$$

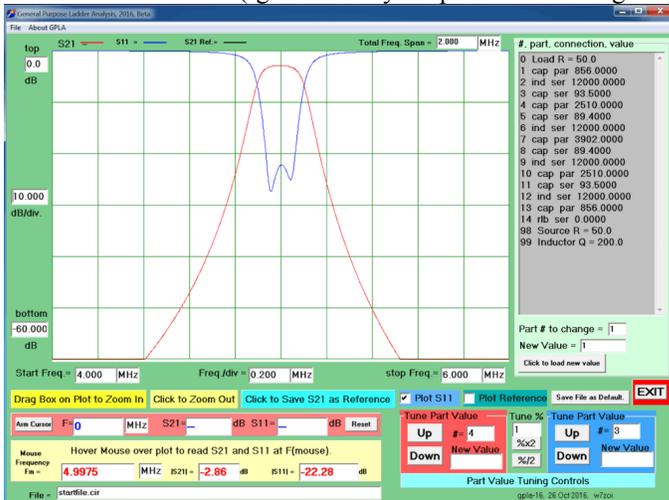
$$T_2 = \frac{1}{\frac{1}{C_0} - \frac{1}{C_{23}} - \frac{1}{C_{12}}} \quad \text{Eq. 7-10.}$$

$$T_3 = \frac{1}{\frac{1}{C_0} - \frac{1}{C_{23}} - \frac{1}{C_{34}}} \quad \text{Eq. 7-11.}$$

$$T_4 = \frac{1}{\frac{1}{C_0} - \frac{1}{C_{es}} - \frac{1}{C_{34}}} \quad \text{Eq. 7-12.}$$



**Fig 7-1.** Schematic diagram for the finished design. This is a partial screen of a schematic editing computer program used to generate the file for the plot that follows. The simulated response for this filter is shown below. (Ignore the crystal parameters in Fig 7-1, for this filter uses no crystals.)



**Fig 7-2.**

## 8. Crystals, Crystal Filters, a N=2 Lower Sideband Ladder

We continue the quest to design filters, but now move to circuits using the quartz crystal. Fig 8-1 shows the familiar model for a quartz crystal. The crystal model has a very high motional inductance,  $L_m$ , that is tuned to a series resonant frequency,  $f_s$ , with a motional capacitance,  $C_m$ . The loss is modeled with an *equivalent series resistance*, ESR. The series resonator is bridged by a parallel capacitance,  $C_0$ .