

Introduction

Since designing my first 2 KW filter for 40m, I have continued to research different circuit and filter techniques. In so doing I have come across, or should I say “remembered”, some very powerful circuit transformations which are invaluable to the filter designer.

My intention, expressed over the following pages, is to lay the ground work enabling a non-engineer to champion these circuit techniques in their own designs. While I will attempt to provide sufficient background with some examples here and there, it is unlikely I can cover everything to the extent needed. Therefore, making use of some of the cited references and other information available will likely be necessary.

Before getting into the details of the remaining filter designs for use at KØZR, I will develop the necessary background in the following areas:

- Transformer winding ratios and impact on voltage, current, and impedance
- Develop background on definition and use of ABCD parameters
- Use ABCD parameters to derive the first of Norton's impedance transforms (very powerful)
- Summarize many of the transformations that are available to the designer
- Exhibit, step-by-step, the use of some of these transforms in an actual filter design for 20m
- Outline the details for my 80, 20, and 15m bandpass filters
- Provide other helpful insights in an Appendix

Transformer Relationships

Pictured to the right is a typical transformer, wherein the primary has associated with it an impedance Z_1 , and the secondary has an impedance, Z_2 . The transformer serves to make a “transformation” in impedance as will be described shortly.

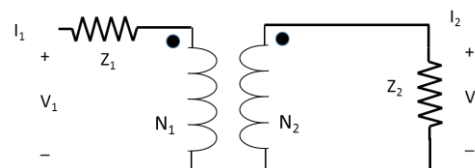


Figure 1

The voltage on the transformer secondary is governed by the number of turns on the primary, N_1 , compared to the number of turns on the secondary, N_2 . If the primary has more turns, $N_1 > N_2$, the transformer will have a lower voltage on the secondary compared to the primary. Conversely, if $N_1 < N_2$, the secondary voltage is stepped-up, or higher, than that applied to the primary.

$$N_1 > N_2 \quad \text{Voltage Step-Down at Secondary}$$

$$N_1 < N_2 \quad \text{Voltage Step-Up at Secondary}$$

There is a direct relationship between number of turns and the voltage observed. In simple arithmetic,

this leads to:

$$\frac{V_1}{V_2} = \frac{N_1}{N_2} \quad [1]$$

Turns Ratio and Transformer Currents

The turns ratio also has an impact on the currents observed in the primary and secondary. If we consider the transformer as “ideal”, the power input to the transformer must equal the power appearing at the output of the transformer. In terms of the mathematics of voltage and current,

$$v_1 i_1 = v_2 i_2 \quad [2] \quad \text{where the “1” and “2” relate to the primary or secondary voltage or current of the transformer}$$

We can divide both sides of equation [2] by i_1 , then divide both sides of that result by v_2 , and obtain a voltage-current relationship for the transformer.

$$v_1 i_1 = v_2 i_2 \quad \frac{v_1 i_1}{i_1} = \frac{v_2 i_2}{i_1} \quad \Rightarrow \quad v_1 = v_2 \frac{i_2}{i_1}$$

$$\frac{v_1}{v_2} = \frac{v_2 i_2}{v_2 i_1} \quad \Rightarrow \quad \frac{v_1}{v_2} = \frac{i_2}{i_1}$$

Principle: If the voltage is increased at the secondary, the secondary current decreases.

[3]

Turns Ratio and Impedance

To understand how the transformer impacts primary and secondary impedances, we begin with the principles just introduced. As already mentioned, transformers are often described in terms of “turns-ratio”, that is N_1 compared to N_2 . This is commonly written as $N_1 : N_2$. Generally, both sides of this relationship are divided by N_1 or both sides are divided by N_2 , giving these modifications:

$$r:1 \quad \text{where} \quad r = \frac{N_1}{N_2} \quad [4]$$

Both ideologies are found in literature, and therefore one must be prepared to work with either.

$$1:r \quad \text{where} \quad r = \frac{N_2}{N_1} \quad [5]$$

A given value of “r” will represent an impedance step-up for one definition of “r” while for the other, an impedance step-down would result.

Form 1: r:1 Ratio

$$i_1 v_1 = i_2 v_2 \quad \text{Divide both sides of this equation by } i_1^2 \text{ giving } \frac{v_1 i_1}{i_1^2} = \frac{v_2 i_2}{i_1^2}$$

Let's look at the left hand and right hand sides of this equation separately.

$$\text{Left Hand Side: } \frac{v_1 i_1}{i_1^2} = \frac{v_1}{i_1} = Z_1$$

$$\text{Right Hand Side: } \frac{v_2 i_2}{i_1^2} \quad \text{Multiply numerator and denominator by } i_2: \quad \frac{v_2 i_2}{i_1^2} \left(\frac{i_2}{i_2} \right) = \frac{v_2}{i_2} \left(\frac{i_2^2}{i_1^2} \right) = Z_2 \left(\frac{i_2}{i_1} \right)^2$$

The ratio of the currents can be rewritten in terms of the turns ratio, which in this case is:

$$\text{for } \frac{i_1}{i_2} = \frac{1}{r} \quad \frac{i_2}{i_1} = r \quad \boxed{Z_1 = Z_2 r^2 \quad Z_2 = Z_1 \left(\frac{1}{r^2} \right)} \quad [6]$$

Form 2: 1:r Ratio

For this alternate definition of transformer ratio, the voltage and current relationships are flipped. The left hand side of the equation remains the same as before, but the right hand side involving the current ratio of i_2 to i_1 is impacted.

$$\frac{v_2 i_2}{i_1^2} \left(\frac{i_2}{i_2} \right) = \frac{v_2}{i_2} \left(\frac{i_2^2}{i_1^2} \right) = Z_2 \left(\frac{i_2}{i_1} \right)^2 \quad \text{Here, for the 1:r definition, } \frac{i_2}{i_1} = \frac{1}{r} \text{ giving}$$

$$\boxed{Z_1 = Z_2 \frac{1}{r^2} \quad Z_2 = r^2 Z_1} \quad [7]$$

Summary Table

Transformer Ratio	Voltages	Currents	Impedance
$r:1$	$V_1/V_2 = r$	$I_1/I_2 = 1/r$	$Z_2 = Z_1/r^2$
$1:r$	$V_1/V_2 = 1/r$	$I_1/I_2 = r$	$Z_2 = Z_1 r^2$

Table 1

Note: in either case, r can take on the following values: $0 < r < \infty$
 { Obviously, 0 and ∞ are impractical values.... } See **Appendix**

ABCD Matrices and Transformations

Two-Port Parameters in General

Throughout electrical engineering it is common practice to use what are termed “two-port parameters” in circuit design. There are several parameter sets in use, each with an accompanying list of pros and cons. For example, z -parameters are used to combine impedances which occur in series, while y -parameters, often called admittance parameters, find use in dealing with parallel or “shunt” components. The general area of RF and microwave design relies heavily on two-port s -parameters, known as “scattering parameters”. Table 2 introduces several of these parameters. We will make use of ABCD parameters.

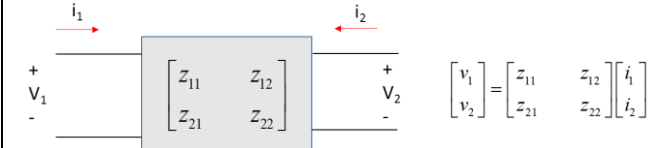
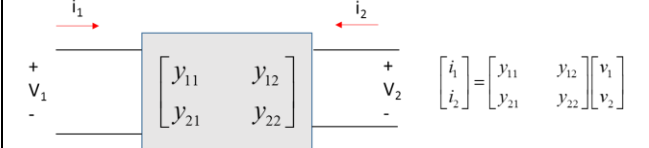
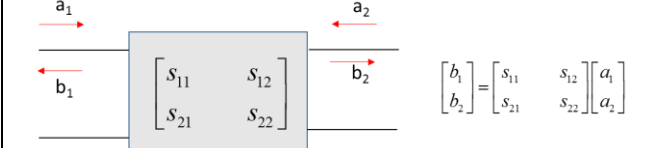
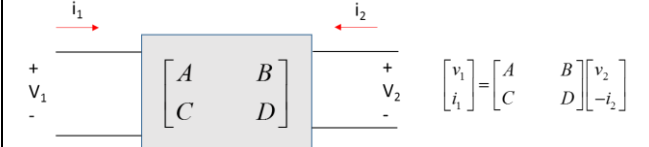
Parameter Set	When Used
	Impedance or “Z” Parameters Used at lower frequencies where impedances are more easily measured. Suited for combining R, L, and C when in series
	Admittance or “Y” Parameters Used up to several hundred MHz. In deriving each element, is easier to get good “shorts” to ground rather than “opens” as required with z -parameters
	Scattering or “S” Parameters Used from tens of MHz to greater than 50 GHz. Based on incident and reflected voltage waves
	ABCD Parameters Provides for cascading elements. One multiplies, in order, the ABCD matrix for each successive element in the chain

Table 2

As mentioned, we will make use of ABCD parameters to derive a powerful transformation known as the Norton transformation. You may have heard of Norton and Thevenin equivalents, wherein a current or voltage source with its accompanying resistor may be changed into the other format. This is different than what will be discussed here.

The Norton transforms I will be developing are used to change sections of a filter to higher or lower impedance levels, or introduce additional capacitance at nodes in a filter where stray capacitance could damage the response of the filter. In this latter case, it is better to have incorporated into a design a known, deliberate amount of capacitance rather than be at the mercy of whatever stray capacitances are

in the design due to the physical layout. This is not a significant problem at HF, but increasing frequency elevates this potential area of concern.

The ABCD Parameter Set

Why ABCD parameters? ABCD parameters allow one to directly matrix-multiply the ABCD matrix of each component that is cascaded in series, arriving at the ABCD matrix for the total cascade. This is far simpler than had a different two-port set been used.

As you will soon see in developing the Norton transform, a shunt capacitor, series capacitor, and additional shunt capacitor are all cascaded together and equated to a single series capacitor and transformer. The same analogy for inductors applies. The ABCD parameters, sometimes called “transmission parameters”, are ideally suited for this cascading.

Figure 2 below shows schematically what we wish to develop. It is desired to develop the element values which will make a series capacitor and ideal transformer electrically equivalent to a “pi” section of capacitors. In so doing, the impedance level within a circuit, in our case a filter, can be increased or decreased, or a shunt capacitor introduced to deal directly with the stray capacitance issue.

The reason one may want to increase or decrease the impedance within a filter is that in so doing, the inductor or capacitor values can be made more achievable. From a manufacturing sense, these transforms also allow one to synthesize a filter with more uniform inductor values throughout, as an example, thus lessening the unique parts count.

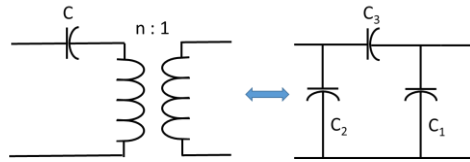


Figure 2

Our objective is to derive the mathematical relationships for these two circuits to the left, making them equivalent over all frequencies. Note the phrase “all frequencies” rather than an “approximation”.

This is where we begin. $\begin{bmatrix} v_1 \\ i_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} v_2 \\ -i_2 \end{bmatrix}$ or written out: $v_1 = A v_2 - B i_2$
 $i_1 = C v_2 - D i_2$. In developing the

values for A, B, C, and D, v_2 is set to zero or i_2 to zero and the reduced equations solved. For example, to determine the value for “A”, one open-circuits the second port making i_2 equal to zero. When this is done the first equation becomes

$$v_1 = A v_2 + B \times 0 \quad A = \frac{v_1}{v_2} \quad [8] \quad \text{for the circuit under consideration}$$

This simple procedure is applied, in turn, for each A, B, C, and D for each particular circuit element. This procedure was applied to a shunt capacitor, series capacitor, and ideal transformer resulting in the following ABCD parameters for each of these simple elements.



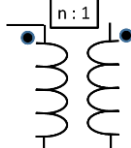
Lumped Element Component	Describing ABCD Matrix
 Shunt "C"	$\begin{bmatrix} 1 & 0 \\ j\omega C & 1 \end{bmatrix}$
 Series "C"	$\begin{bmatrix} 1 & 1/j\omega C \\ 0 & 1 \end{bmatrix}$
 Ideal Transformer	$\begin{bmatrix} n & 0 \\ 0 & 1/n \end{bmatrix}$

Table 3

The beauty of the ABCD parameter set allows us to multiply each of these matrices in succession, setting the two representations equal. Solving the resulting simultaneous equations will give us the Norton transformation for a single series capacitor and ideal transformer. The figures below show pictorially the next steps.

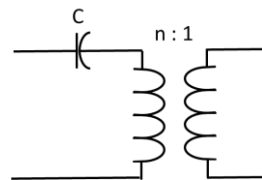
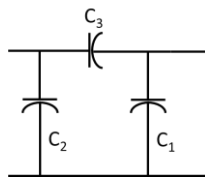
	
$\begin{bmatrix} 1 & 1/j\omega C_s \\ 0 & 1 \end{bmatrix} \begin{bmatrix} n & 0 \\ 0 & 1/n \end{bmatrix}$	$\begin{bmatrix} 1 & 0 \\ j\omega C_2 & 1 \end{bmatrix} \begin{bmatrix} 1 & 1/j\omega C_3 \\ 0 & 1 \end{bmatrix} \begin{bmatrix} 1 & 0 \\ j\omega C_1 & 1 \end{bmatrix}$

Table 4

The left column above is the matrix-multiply of the ABCD matrices for the series capacitor and ideal transformer. The right column, in similar manner, is the ABCD matrix multiplication of the three capacitors in cascade.

Multiplying the matrices for the series capacitor and ideal transformer gives:

$$\begin{bmatrix} n & \frac{1}{j\omega C n} \\ 0 & \frac{1}{n} \end{bmatrix} \quad [9]$$

Multiplying the three ABCD matrices together for the pi section of capacitors gives:

$$[10] \quad \begin{bmatrix} 1 + \frac{C_1}{C_3} & \frac{1}{j\omega C_3} \\ j\omega C_2 + j\omega C_1 \left(1 + \frac{C_2}{C_3} \right) & 1 + \frac{C_2}{C_3} \end{bmatrix}$$

In methodical order, we now equate each of the four terms in the first matrix to the second matrix.

(1,1)	$n = 1 + \frac{C_1}{C_3}$ [11]
(1,2)	$\frac{1}{j\omega C_s n} = \frac{1}{j\omega C_3} \rightarrow C_3 = n C_s$ [12]
(2,1)	$C_2 + C_1 \left(1 + \frac{C_2}{C_3}\right) = 0 \rightarrow C_2 C_3 + C_1 C_3 + C_1 C_2 = 0$ [13]
(2,2)	$\frac{1}{n} = 1 + \frac{C_2}{C_3}$ invert and sub for C_3 from (1,2): $n = \frac{n C_s}{n C_s + C_2}$ [14] $n^2 C_s + n C_2 = n C_s$ $C_2 = C_s (1 - n)$

Table 5

So far we have relationships for C_3 and C_2 . Substituting (1,2) into (1,1) gives the final needed relationship for C_1 .

$$n = 1 + \frac{C_1}{C_3} \quad n = 1 + \frac{C_1}{n C_s} \quad n = \frac{n C_s + C_1}{n C_s} \quad n^2 C_s - n C_s = C_1 \quad [15]$$

$$C_1 = n(n-1)C_s$$

Norton Transformation for a Single Series Capacitor and Ideal Transformer

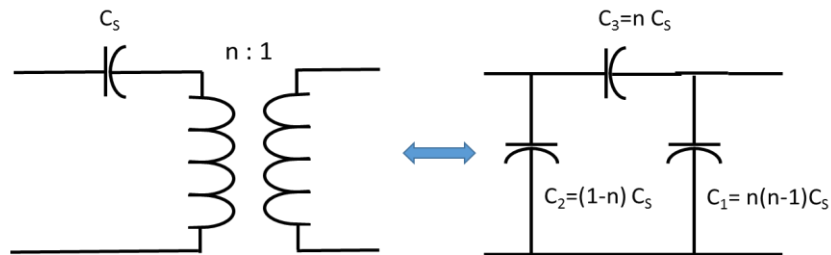


Figure 3

A set of Norton Transformations is nicely summarized below. Other useful transformations are found in the references. In the *Appendix I* address a turns ratio 1:n and its impact on the formulae.

<p>Series L or C</p>		
<p>A series C or L with transformer</p>		

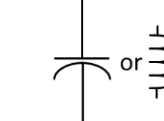
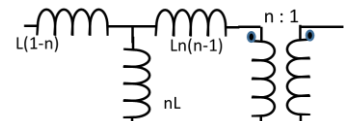
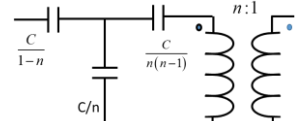
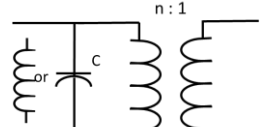
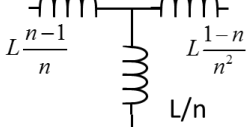
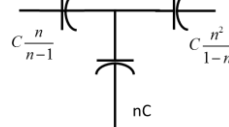
 <p>Shunt C or L</p>		
		

Table 6

The left-hand column in Table 6 depicts the transformation for L or C instances. Looking at the first row (previous page), if one has a series C you use the transformation containing the C values in the third column. If you were working with a series L, one would use the transformation containing L values in the second column.

An important point is the following: performing these transformations results in negative component values. These will occur on the high Z side of n. The negative value is combined with other Ls and Cs in the circuit (in other words, other Ls or Cs as appropriate must be present to make this combination; if they are not you cannot use the transform in this way). Obviously, one must perform the proper transformation so the resulting negative values can be absorbed with a positive component value. {This will limit you sometimes in whether you can do a step-up or step-down in impedance}

Filter Design in General

A number of parameters are available to the filter designer. The different filter families, i.e. Butterworth, Chebyshev, and Cauer (elliptic) to name a few, have their individual characteristics. Of great importance is selection of the proper passband width (for bandpass filters), the width of the stopband, and in the case of Cauer types, what ultimate attenuation is desired in the stopband. Additionally, the filter's characteristic impedance may be designed for lower or higher than 50 Ω , subsequently employing impedance matching networks at the input and output of the filter if necessary. While this increases total part count, sometimes the benefit in reduced voltages or currents makes the additional impedance transformations worthwhile. And of course, there is the application of available transforms, such as the Norton transformsⁱⁱ, albeit this is considerably more complex than aforementioned methods.

20m Elliptic Bandpass Filter Design

Objective: Attain passband insertion loss ~ 0.2 dB while placing notches as close as possible at 7 and 21 MHz. Stopband attenuation of 50 dB was selected, resulting in a 5th order requirement. A multitude of software tools are available to the filter designer, with some being completely free or "close to it". ELSIE is such a program and is used in this design to obtain the values for the filter.

Topology

☐ Capacitor-input lowpass
☐ Inductor-input lowpass

☐ Nodal capacitor-coupled bandpass
☐ Nodal inductor-coupled bandpass
☐ Shunt-input bandpass
☐ Series-input bandpass
☐ Mesh capacitor-coupled bandpass
☒ Cauer-only bandpass

☐ Capacitor-input highpass
☐ Inductor-input highpass

☐ Series-input bandstop
☐ Shunt-input bandstop

Dimensions

☐ cm
☒ in

Add title to printout

Add info to Elsie file

Entry assistance

Family

☐ Butterworth
☐ Chebyshev
☒ Cauer
☐ Bessel
☐ Gaussian
☐ Gaussian to 12
☐ Constant-K
☐ M-derived

See normalized values

☐ Manual entry

☐ Cauer even terms
☒ Best selectivity
☐ Equal terms

☐ Cauer BPF topology
☒ Normal
☐ Zig-zag

Ripple

Bandwidth (Hz) (Fc)

5M

Center frequency (Fo)

13.25M

Order (N) [21 max]

5

Input termination (Rs)

50

Passband ripple (Ap)

0.00167

VSWR: 1.04 Return: -34.152 dB

LP prototype

Stopband width (Fs)

12M

Stopband depth (dB) (As)

50

In selecting these design parameters, a considerable amount of time was spent trading f_c , stopband bandwidth, passband bandwidth, etc in order to strategically place the notches of the elliptic response. Furthermore, it is known that the narrower the passband the higher some of the currents will be in parts of the filter; one needs to minimize this.

Figure 4 ELSIE Design Window

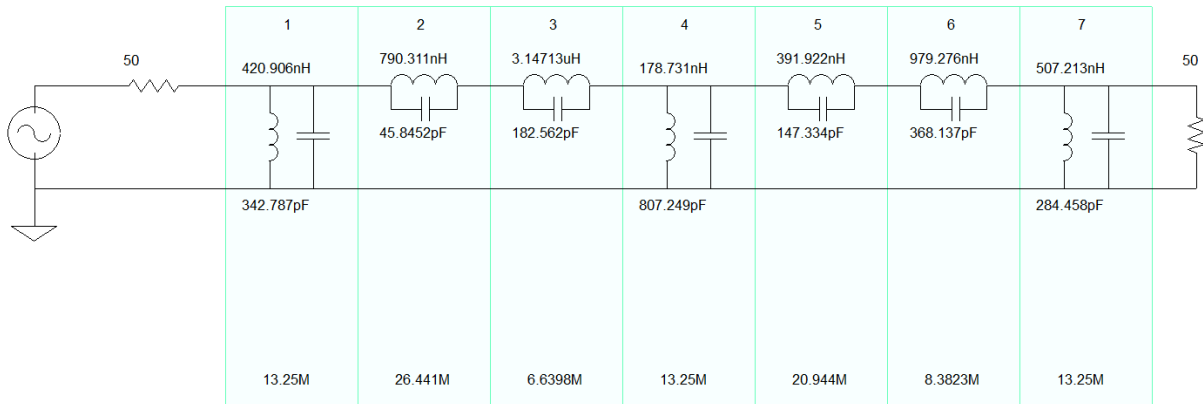


Figure 5 Output from ELSIE

The circuit for this filter was augmented with coil resistance netting Q_s of 400. A SPICE-like program, Simetrixⁱⁱⁱ, was then used to analyze the filter for voltage and current when subjected to 1500 watts at the input. For the seven different resonators above, the accompanying currents were determined as shown below in Table 7.

L ₁	11.5	L ₅	14
C ₁	9.7	C ₅	6.25
L ₂	11.1	L ₆	3.6
C ₂	3.0	C ₆	11.3
L ₃	1.9	L ₇	8.1
C ₃	9.9	C ₇	9.6
L ₄	21.2		
C ₄	25		

Table 7 RF Currents in ELSIE 20m Filter

Clearly some of these RF currents are noteworthy and require special attention if one does not want a catastrophic failure when operating at high power.

To withstand these high currents, multiple paralleled capacitors are required. More will be said on this topic later, but the CDV series of silver mica capacitors is documented as handling under continuous conditions, 5-6 amps at HF. My original designs had planned on using the thought-to-be less expensive route of MLCCs (multi-layer ceramic chip capacitors), however further research into the minimal

available data indicated a higher risk than I was willing to accept. This design will use CDV16 and CDV19 capacitors purchased from Mouser Electronics^{iv}.

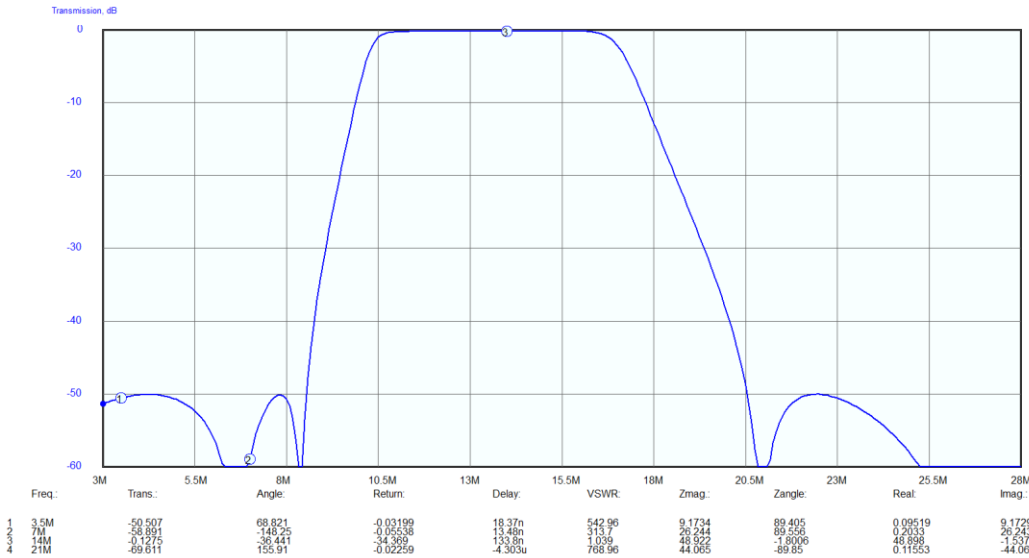


Figure 6 ELSIE Simulated Filter

Due to the low frequencies in use here, as well as the air-suspended nature of filter construction, I do not believe stray capacitance occurring at two of the floating nodes is a serious concern. Of greater concern is the high current in L₄-C₄. In an attempt to reduce this current, and simultaneously diminish any potential stray capacitance occurring at the two high impedance points

(between L₂||C₂ and L₃||C₃ as well as between L₅||C₅ and L₆||C₆ of Figure 5), Norton transformation techniques were applied to the ELSIE synthesized filter.

In use of the transform, it is the case that negative values of capacitance and inductance result. This is expected and used to our advantage. The direction of the transform (impedance step-up or step-down) must be done in such a direction that where the negative component values occur, other positive components are present to absorb them. More will be said about this shortly.

The transform is first applied to the second series parallel resonator composed of L₃ and C₃, values L=3.147u and C = 182.6p. An additional transform^v directly attributable to the work already discussed, is used as it applies directly to the case of a parallel L-C followed by a transformer. The details of the derivation follow a similar venue as used in the earlier development for the first Norton transform.

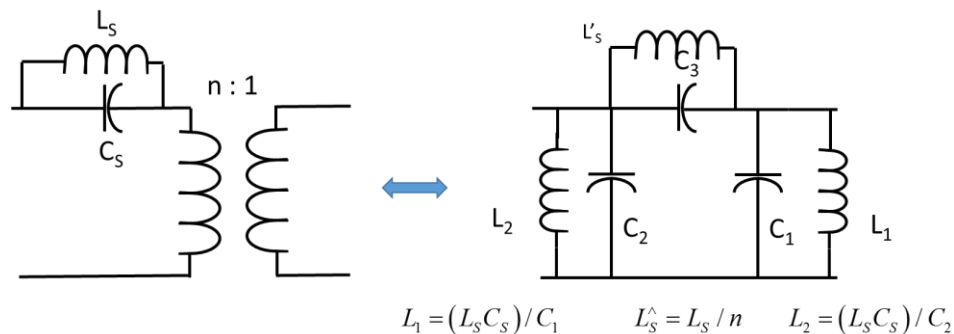


Figure 7 LC Resonator Transform

To eliminate any possibility for ambiguity, the beginning filter schematic is shown below in Figure 9.

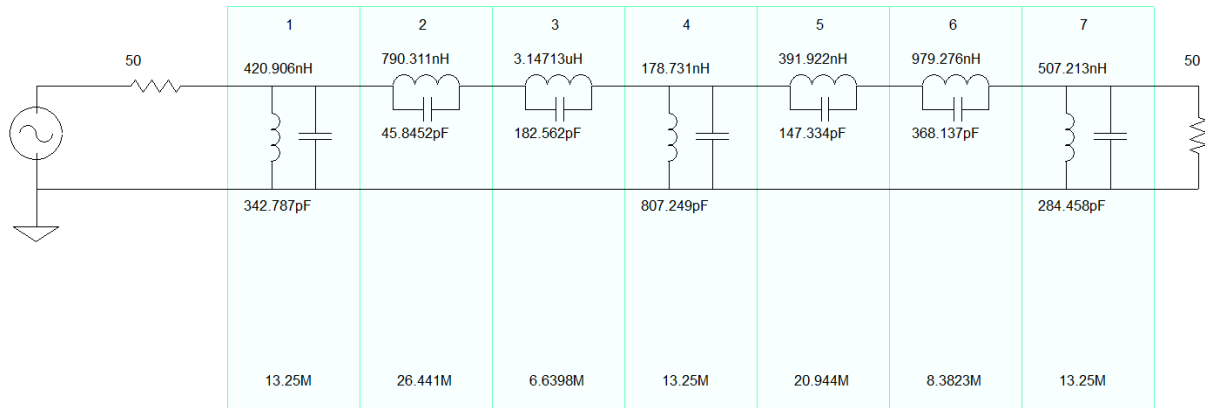


Figure 8 Beginning Filter - Same as Figure 5

The transform just presented (Figure 7) will be applied to resonator #3. In so doing the impedance at that point (and beyond to the right) is elevated to 100 ohms. Therefore, the 4th and 5th resonators must be impedance scaled by a factor of 2 as well. Resonators 6 and 7 remain unchanged because an impedance step-down transformer will be applied next at resonator 5. More discussion on this follows.

The equations presented with the transform are programmed into a simple Excel sheet. The results of those calculations are the following:

Ls	3147.00	
Cs	182.56	
n	0.71	
Cap1	-37.8094	
Cs_prime	129.0894	pF
Cap2	53.4706	
L1	-15195.0602	
Ls_prime	4450.5301	nH
L2	10744.5301	

A quick review of the equations shows that units cancel out, thus one can enter the inductance of L_s in terms of uH or nH, and similarly for the capacitors.

We see immediately that C_1 and L_1 have negative values, and as such must occur to the right side of the transform so they can be later combined with resonator 4. (This also tells one a step-down is not realizable) As a result of this first transform application, raising the impedance in this area to 100 Ω , the following schematic of Figure 10 applies. Included in this schematic is the impedance scaling of the shunt resonator #4 occurring in Figure 8.

Figure 9 Transform

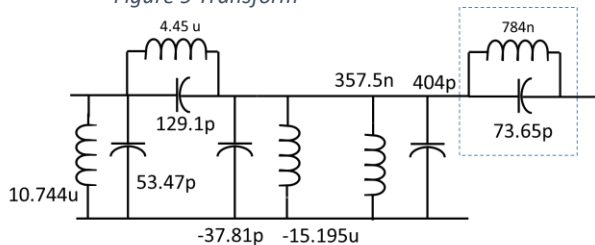


Figure 10 Application of First Transform

To the left in Figure 10 are the results of:

- Applying the transform to the 3.147 uH and 45.8 p cap
- Doubling the impedance of the original shunt 807pF and 178.7 nH inductor
- Impedance scaling by X2 the next parallel resonator, originally 391.9 nH and 147.3 pF (before the scaling), outlined in a "dotted" box in Figure 10

It is extremely easy to forget that before applying the second transform (yet to come), one must impedance scale those L_s and C_s to the right of the first transformed area. Also,

note that no combination of negative elements has been done yet because additional shunt elements will appear when the reverse transform (going from 100 Ω back to 50 Ω) is applied to the 784n||73.65p combination in Figure 10.

As you can see, there is a lot going on here. The only way you will fully understand this process is by drudging through the calculations yourself.

Now we are ready to apply the second transform. The Excel spreadsheet for this is shown below in Figure 11.

Ls	7.84E+02	
Cs	7.37E+01	
n	1.414213562	
Cap1	43.15	
Cs_prime	104.19	pF
Cap2	-30.52	
L1	1338.10	
Ls_prime	554.26	nH
L2	-1892.36	

Figure 11 Second Transform

The second transform is now applied, reverting from 100 Ω back to 50 Ω . As shown to the left, there are negative values of L and C which occur on the left side of the transformed result. These additional elements are now added to those just presented schematically in Figure 10.

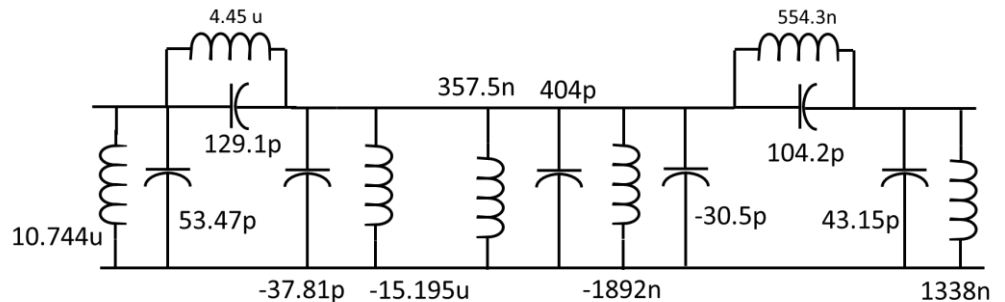


Figure 12 Second Transform Schematic

The finish line is in sight! All that is left to do is combine the inductors, yes both the positive and negative ones, and the same for the capacitors. The inductors combine as resistors in parallel and the capacitors like resistors in series.

$$\frac{1}{\frac{-1}{15.195} + \frac{1}{0.3575} - \frac{1}{1.892}} = 454n$$

-15.195u 357.5n -1892n

Figure 13

$$-37.81p + 404p - 30.5p = 335.7p$$

-37.81p 404p -30.5p

Figure 14

Transformed 20m Filter

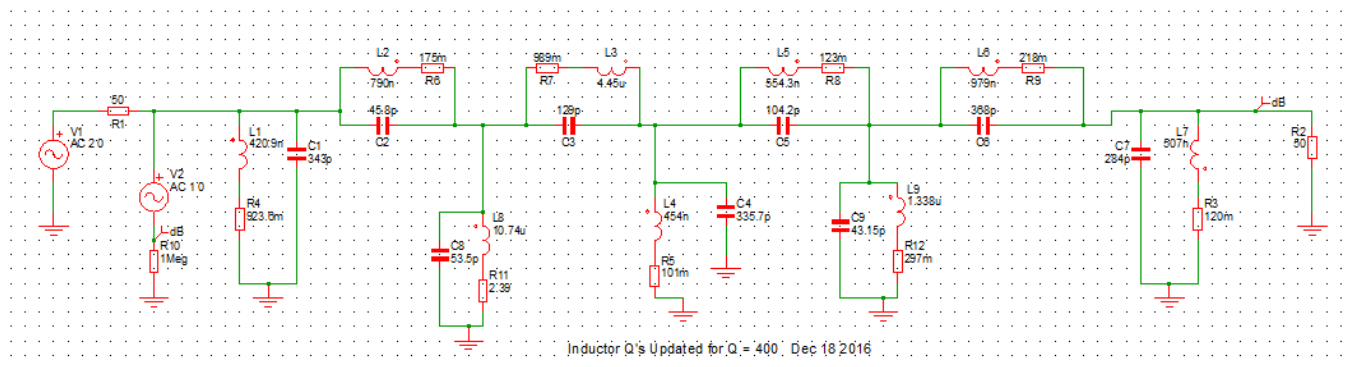


Figure 15 Results of Double-Transformed 20m Filter

Transformed 20m Filter Response

Over 65 dB of loss at 21 MHz; 58 dB at 7 MHz Insertion Loss ~ 0.25 dB

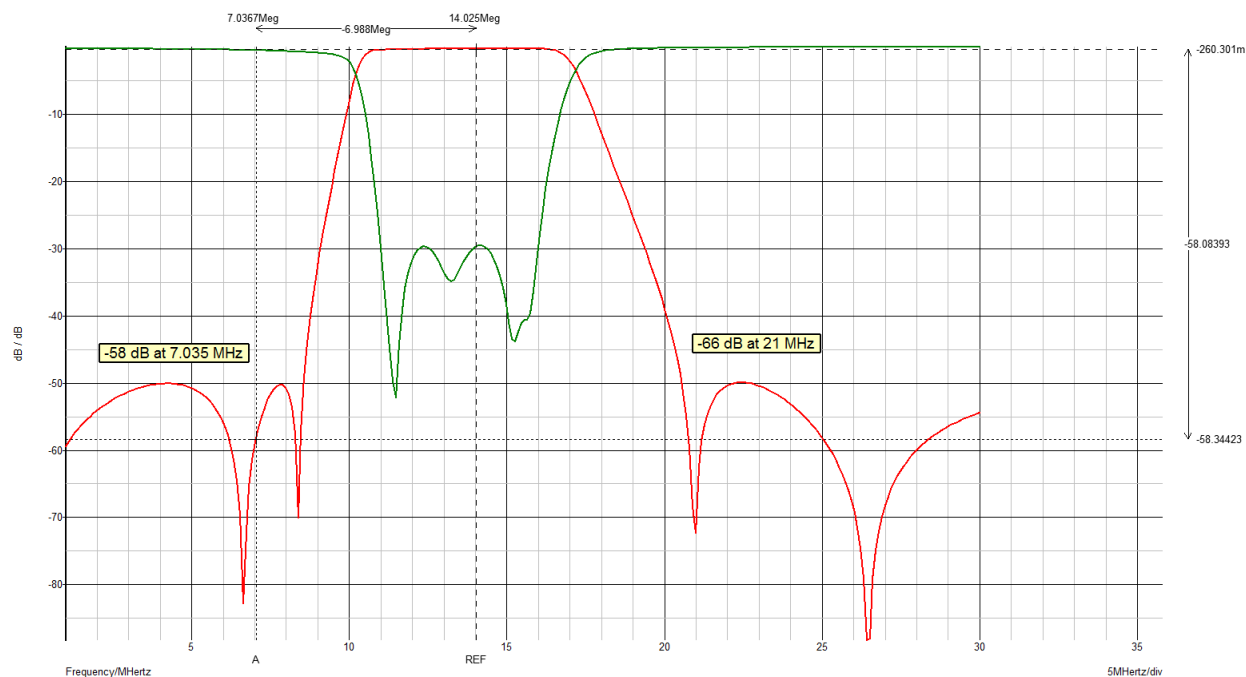


Figure 16 Simetrix Simulation With Finite Inductor Qs

L ₁	11.5 (10.0)	L ₅	14 (11.5)
C ₁	9.7 (11.5)	C ₅	6.25 (5.4)
L ₂	11.1 (11.5)	L ₆	3.6 (4.3)
C ₂	3.0 (3.4)	C ₆	11.3 (11.9)
L ₃	1.9	L ₇	8.1
C ₃	9.9	C ₇	9.6
L ₄	21.2 (13.5)		
C ₄	25 (16.4)		

Table 8 Comparison of Currents in Two Designs

While there is not a one-to-one correspondence in part labeling between the original and transformed filters, some of the “like parts” currents are shown to the left in parenthesis. Of particular note is the troublesome 25 amps in the former design. It is now ~ 16 amps after the transformation – a huge improvement. Those current values within parentheses in Table 8 represent the improved design values.

Don’t Plug in the Soldering Iron Yet

While the filter just designed has improvements, primarily in the high currents within the middle parallel resonator, all is not well when one considers the component values required. For example, the second shunt resonator requires almost 11 μ H and one of the series inductors requires over 4 μ H. While one might get away with this, the physical size may become unacceptable, or possibly the interwinding capacitance may lead to an unacceptably low self-resonance for the 11 μ H inductor; a real possibility.

Therefore another design attempt was made on the filter beginning with the schematic of Figure 15. The Norton transform is now performed on the #2 and #8 resonators, first transforming down to 25 ohms then back up to 50 ohms. The inductor sizes are hence reduced and the parts spread is diminished. No inductors are larger than 1.6 μ H, giving added confidence that inductor self-resonances will not be problem. However, with these positive attributes comes a huge negative of some 30 amps of current circulating in new resonator #4, the middle resonator shunted to ground. This occurs because the impedances were reduced by 50% in doing the transformation, resulting in a current doubling.

No combinations of transforms worked to remove this problem that I could identify. I considered quite a number of options far beyond what is discussed herein, and invariably negative valued components would end up in the wrong place where they could not be absorbed by other components in the vicinity. I decided to apply another solution.

One of the virtues obtained by the transformation between resonators #2 and #6 is, I believe, a diminished voltage level across the circuit nodes compared to the original ELSIE design. This allows one to consider a powdered-iron transformer where the center shunt resonator is located. SPICE evaluates this node as being 216 V rms for a 1500 watt input. HOWEVER, this 216 V rms is across each winding, so the voltage appearing across C₄ in Figure 17 is closer to 1.2 KV. The 216 V rms is used in the formulas for toroid B_{MAX} available from Amidon (and repeated in the index for completeness) and found to be very compatible with our design goals. The lower V_{RMS} per winding, higher frequency, and use of a quadrifilar winding is the ticket and solves the high current issue.

The use of the quadrifilar winding elevates the impedance 16 times (to 400 ohms), thus the current through the 100 pF capacitor is now only some ~ 7.4 amps rms; a fantastic improvement of 4X over the original design of Figure 5 that had C₄ current max of 25 amps.

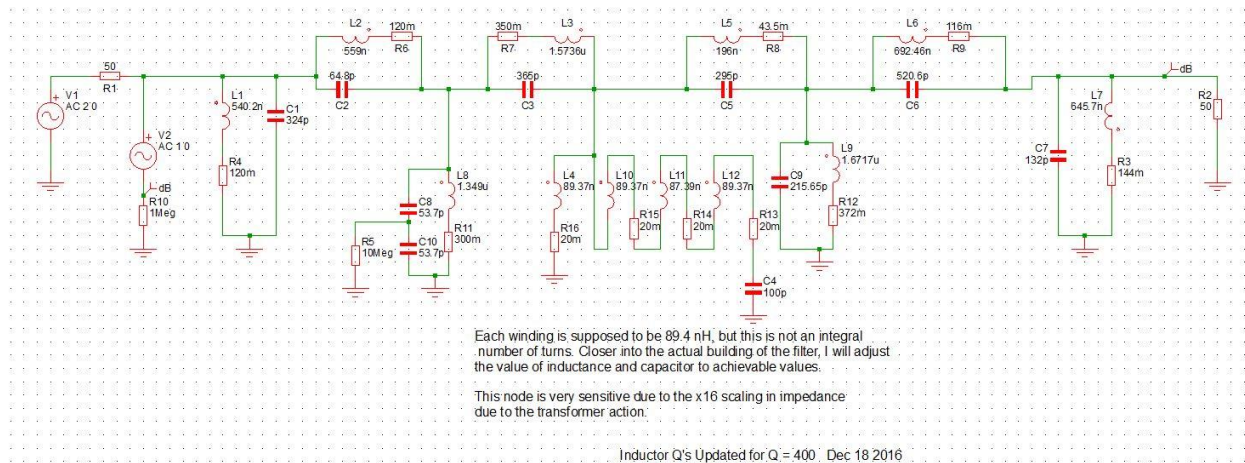


Figure 17 Second 20m BPF Filter Attempt

In SPICE, every node must have a DC path to ground, even if through a 10 megohm resistor. This, in fact, occurs between the two series capacitors above in Figure 17 and is a requirement of the SPICE algorithms.

For completeness and further documentation of this second design attempt, the input and output transformations as calculated in a simple Excel spreadsheet are included below.

Input Transform	n:1 definition								
	Ls	790.31				64.83462077			
	Cs	45.85				Cs_prime			
	n	1.41421							
						558.8343		Zo =	0.5
						Ls_prime			
	Cap1	26.8554							
	Cs_prime	64.8346	pF			-1907.9795	-18.9896	26.85538	1349.1453
	Cap2	-18.9896				L2	C2	C1	L1
	L1	1349.1453							
	Ls_prime	558.8343	nH						
	L2	-1907.9795							

Figure 18 First Transform

Output Transform

Ls	489.64								
Cs	736.27								
n	0.707107								
			Zo =	0.5					
Cap1	-152.49								
Cs_prime	520.62	pF							
Cap2	215.65								
L1	-2364.19								
Ls_prime	692.46	nH							
L2	1671.74								

Figure 19 Second Transform

The filter response is shown below in Figure 20. It meets most of the design parameters established at the beginning of this process.

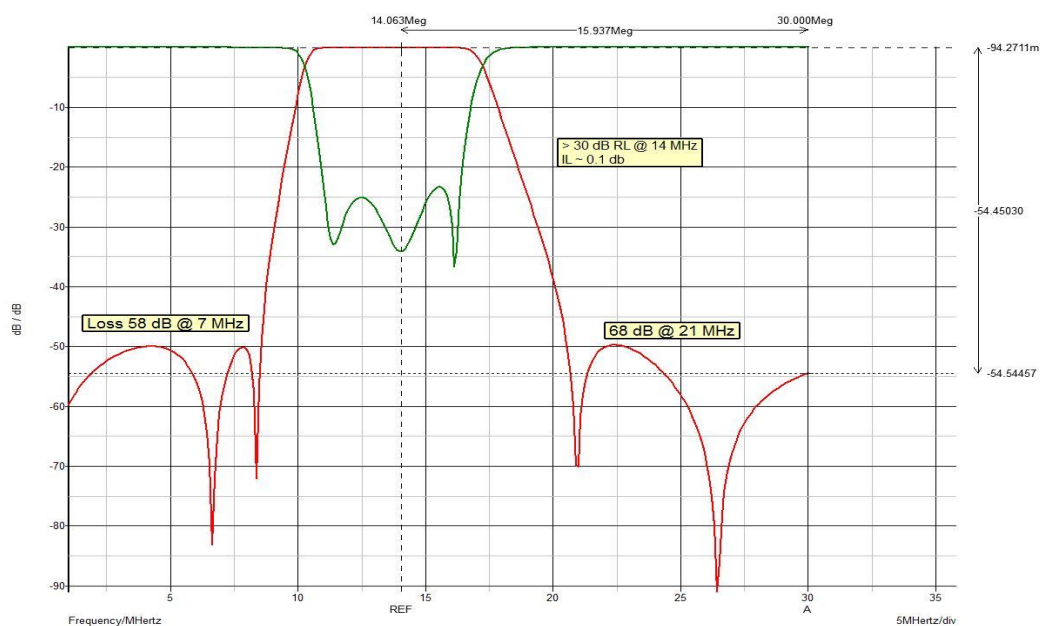


Figure 20 Second 20m BPF Design Simulated in Simetrix

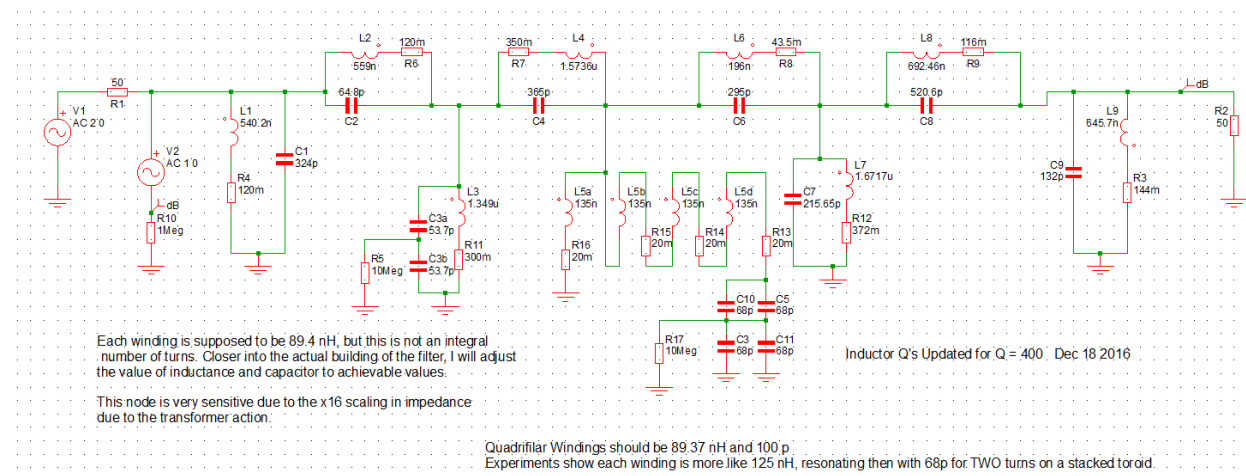


Figure 21 Finalized Schematic of 20m BPF

Figure 21 is essentially a repeat of Figure 17 with two differences. First, the toroid and its accompanying capacitor C₅ have been adjusted to what I believe will be closer to the actual implementation in terms of amount of inductance. Secondly, all the L and C designators have been relabeled, starting from the left, to improve readability. The component peak currents and individual L-C resonant frequencies are shown in Tables 9 and 11. The “Res #” is for the L-C pair, beginning from the left and proceeding to the far right. Critical node voltages as a function of load impedance are shown in Table 10.

L ₁	8.4	L ₆	20.6
C ₁	11.8	C ₆	9.6
L ₂	11.8	L ₇	2.7
C ₂	3.4	C ₇	7.8
L ₃	4.6	L ₈	4.5
C ₃	1.3	C ₈	12.5
L ₄	3.0	L ₉	6.7
C ₄	14	C ₉	4.5
L ₅	6.7		
C ₅	6.7		

Table 9 Peak RF Currents by Component

Load Z	L ₃	L ₅	C ₅	L ₇
25	510	220	850	390
50	485	270	1.1 K	400
75	520	320	1.3 K	430
100	560	360	1.43 K	460
100 @ 2 KW	645		1.6 K	

Table 10 Critical Node Voltages as Loads Change

$$I_{Pow Lev} = I_{1500W} \sqrt{\frac{Power}{1500}} \quad [16]$$

Res #	Res Freq
1**	12.03
2	26.44 ++
3	26.44 ++
4	6.641 ++
5**	13.309
6	20.93 ++
7	8.382 ++
8	8.382 ++
9**	17.239

Table 11 Resonator and Resonant Frequency

In tuning the filter, the transmission zeros, designed with a double “++” sign Table 11, should be adjusted first. They control the deep notches in the stopband of Figure 20. To the extent possible, further tuning of the filter should attempt to leave these untouched once set. Those resonators with a double asterisk ** in Table 11 are adjusted last and will have a strong impact on the passband return loss.

Construction Considerations

The 40m bpf constructed a few weeks ago was done wholly with MLCCs (multi-layer ceramic chip capacitors). MLCCs are more generally available, are considerably less in cost, but do come with some challenges. I have listed what I see as the major ones below, which include:

- 1) ESR (equivalent series resistance) and RF current capacity details are difficult to find, especially at RF
- 2) Difficult to really know how much margin one has in current capacity due to lack of information
- 3) Current capacity is generally < 1 amp per MLCC, thus requiring many capacitors in parallel, driving up capacitor costs
- 4) Most MLCCs, except the most expensive ones (from American Technical Ceramics), stipulate more exotic soldering methods, such as solder reflow, with very tailored and highly controlled, short duration temperature exposures, etc. In lieu of this type of manufacturing, i.e. just using a soldering iron, one subjects unknown thermal stresses on the MLCCs, perhaps leading to premature catastrophic failures. Some things can be done to ameliorate this concern, such as heating the PCB before mounting, putting the MLCCs on a hotplate prior to mounting to lessen the thermal shock, etc. In my mind today, the MLCC route comes with too many process control issues which I do not care to solve, at least right now.
- 5) Another process related concern. The MLCCs should be mounted with solder paste. This has a shelf life of approximately six months (you might find a product with longer shelf life). The most problematic to me is the concern that the resistivity of the solder paste may be higher than that of a soldered connection. If this were true, in the presence of some of the high RF current, elevated heating at the junction between the MLCC and PWB could occur, adding further thermal stresses to the MLCC during actual operation. Some PMP (processes, manufacturing, parts) expert could better address this. In recent weeks I have found that CDV16 and CDV19 silver mica capacitors have HF RF current capacities around 5-6 amps, and while more expensive than most MLCCs, in the long run are likely better than

the MLCC route, and less expensive, in my mind. So, for this design, CDV16 capacitors are used almost exclusively.

Construction Details and Measured Results

Below in Figure 22 is the 20m BPF populated PCB. It is not yet in a chassis; that will be the final step. In the center you are able to see a stacked pair of T-130 Mix 17 powdered iron toroids. Two turns was “too much” but that was about the minimum I could reasonably do. Had I had some more suitable toroids available (without ordering more), I could have likely gotten closer to the design value of $1.43\ \mu\text{H}$. Unfortunately a higher inductance closer to $1.8\ \mu\text{H}$ or $1.9\ \mu\text{H}$ was realized (resonant frequency measurements with a known capacitance). Placing this higher inductance value in the Simetrix model decreased the stopband attenuation some (47 dB instead of the as-designed 50 dB) but I elected to take this “hit”. Recall that had the original design value of 89.37 nH been used, some 30+ amps would have been circulated in this parallel resonator rather than ~ 7 amps.

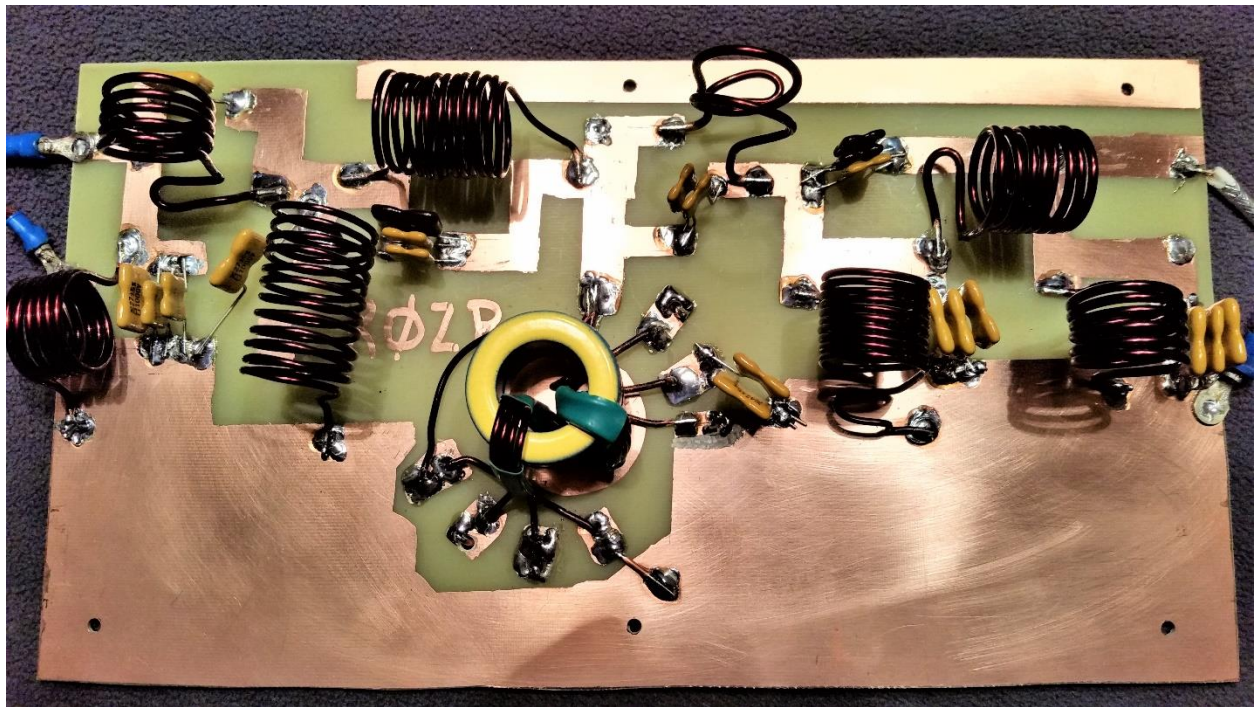


Figure 22 Populated 20m PCB

The inductors in Figure 22 which carry the higher currents are formed from AWG 12 permaleze wire while the others use AWG 14. The “loose arrangement” of the toroid and its own self resonance is believed responsible for eroded stopband performance above 30 MHz; more on that shortly. My greatest concern is performance on 40m and 15m since in SO2R operation, the neighboring bands will likely find the greatest utilization. Such a caveat is not applicable in a multi-multi station, in which case all stopbands are important.

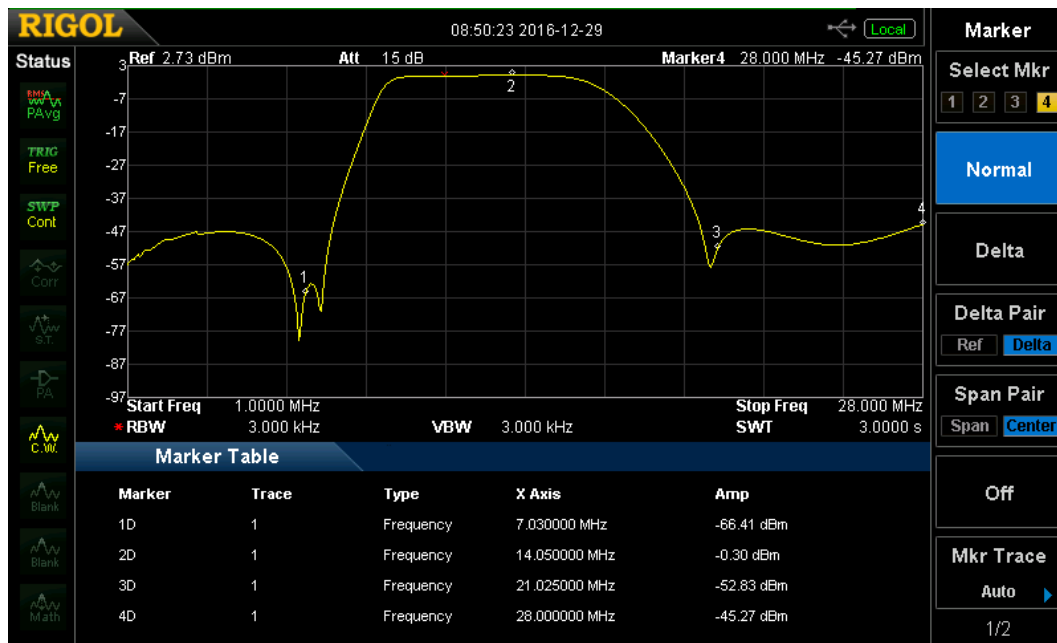


Figure 23 MHz to 28 MHz Sweep With Marker Table – Unmounted PCB

In Figure 23 it is noticeable that the stopband is edging up higher at 28 MHz. That is shown further in Figure 24 below. Take note, however, that this is without the PCB mounted to its aluminum plate.

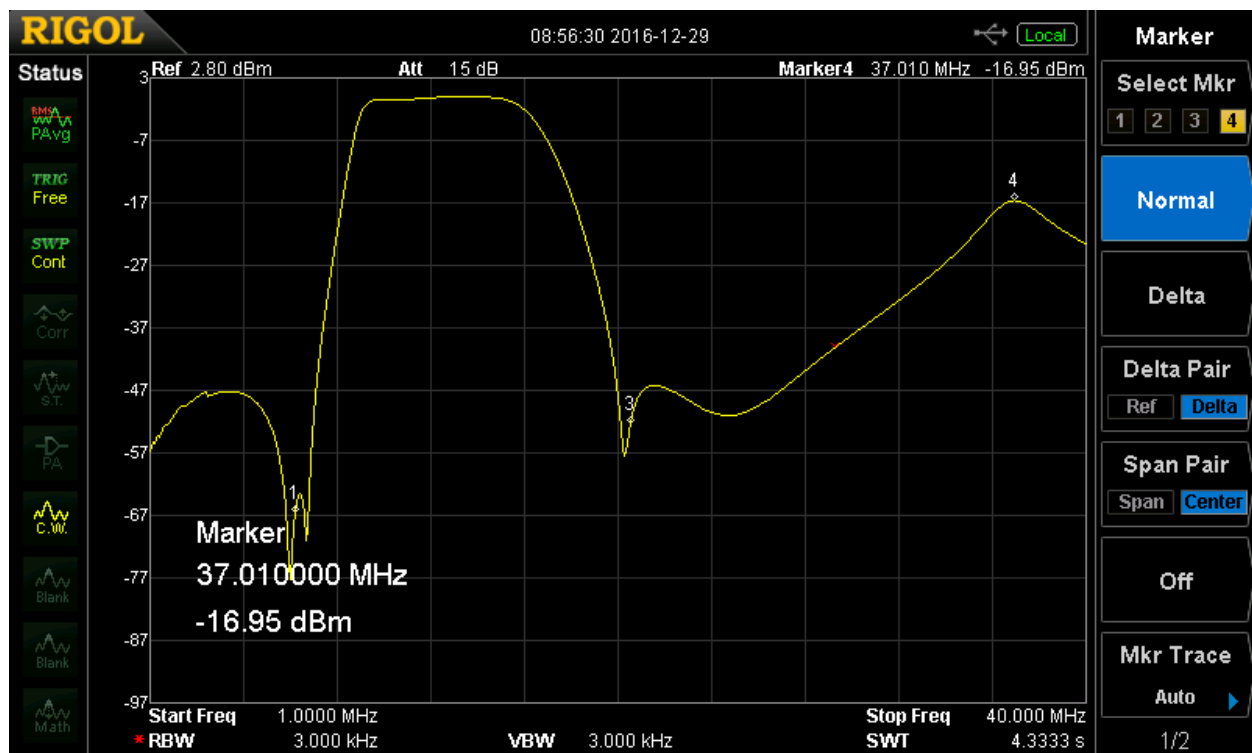


Figure 24 Full Bandwidth Span – Unmounted PCB

While I will be the first to admit I do not like this “second return” around 37 MHz, in actuality it is far from the frequencies of operation and I want to preserve the 4X reduction in current afforded by use of the toroidal transformer.

During adjustments on the filter, I rather “double dipped” in that the pole normally occurring at 8.38 MHz was moved down in frequency to add to the attenuation normally afforded to the 40m band. Figure 24 shows > 65 dB attenuation on 40 m!

Insertion loss has been somewhat difficult to measure, accurately, because it is so low. Several attempts have garnered measurements between 0.1 and 0.15 dB loss. The return loss over the 20m band is also very good, as one would expect for such low insertion loss; return loss exceeds 25 dB across the band.

Final Tweaks Required

Somewhat surprising to me, given my experience on my first 40m BPF (which had no toroids, however), was the fact that when mounting the PCB on an aluminum plate caused some shifts in VSWR and passband. The plate will eventually be enclosed by an aluminum box. The inductance of the toroid increased still further (likely due to “reflected inductance” in close proximity to the aluminum ground plane and interconnecting wires), thus requiring another change in the capacitor to ground off the fourth winding of the cores.

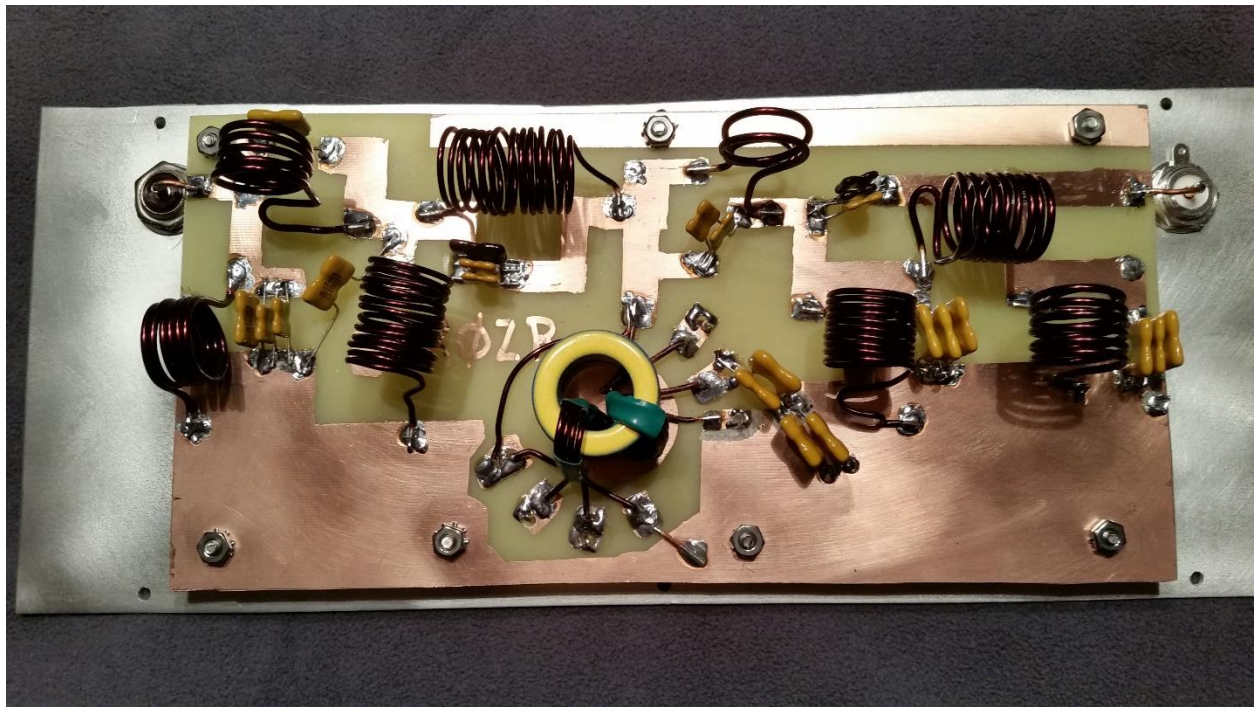


Figure 25 20m BPF Mounted on Aluminum Plate

The overall results of the filter once placed on the aluminum plate are more favorable than presented earlier in Figures 23 and 24; see Figures 26-28. The aluminum plate brings ground closer to the entire PCB rather than just through the copper ground plane of the board itself.

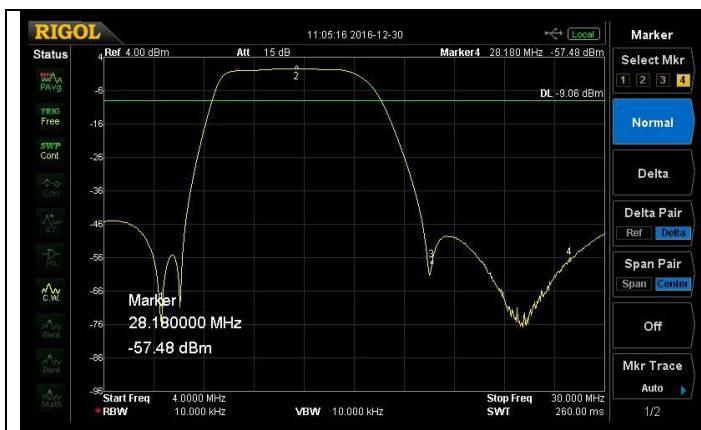


Figure 26 4 MHz to 30 MHz Response

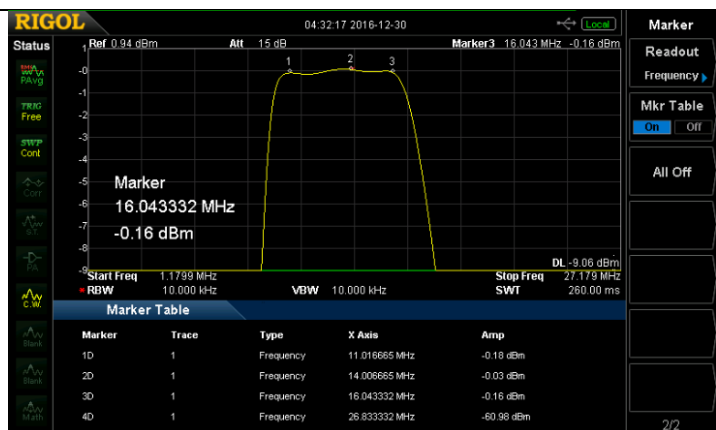


Figure 27 Passband Response on 1-dB Scale

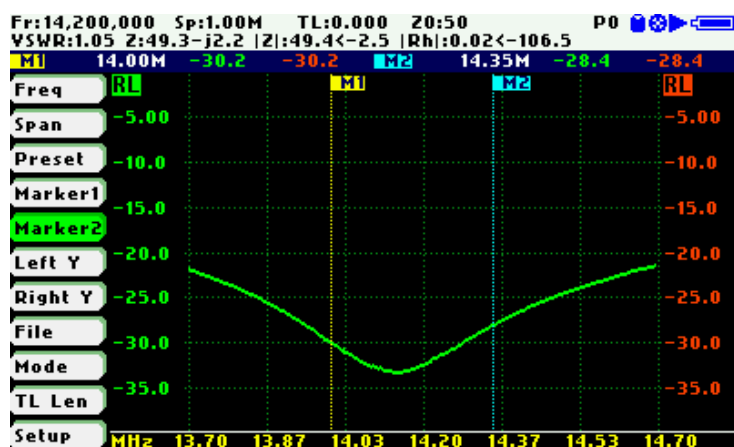


Figure 28 Passband Return Loss

Frequency	Insertion Loss, dB
3.5	~ 50
7.0	65
14.1	0.15
21	58
28	57
Return Loss	
14.0	30 dB
14.35	28.4 dB

Table 12 Electrical Parameters

Other Construction Details and the "Smoke Test"

As one would expect, I was unable to get the "exact" capacitor values, thus opting for as close as possible with the CDV capacitors I had on hand. An updated schematic is shown in Figure 29. In preparing for the powered test, I discovered an oversight in the design of the PCB layout. While I have been vigilant regarding each components' voltage and current requirements, I overlooked the fact that the capacitor on the fourth winding of the toroid actually receives 4X the voltage I was expecting, so instead of the anticipated 216 V predicted by Simetrix, the voltage appearing across the capacitors Figure 21, L5d, is over 1.2 KV. Therefore, in Figure 25 one can see four capacitors to ground off one of the toroidal windings.

The filter was placed immediately after a Tentec 1.2 KW amplifier, operating into a 2 KW dummy load. Operation at several hundred watts revealed no problems. I gradually elevated the power to ~ 1.2 KW and transmitted CW as though I were in "contest mode" for some 5 minutes. I detected only the smallest heating in coils L₁ and L₂. The stacked toroids were as cool as if no power was going through the filter. I will probably install a low-volume fan on the filter just to be on the safe side for the big contests.

In the layout of the filter I was conscious about field interactions between the multiple inductors. Some is unavoidable in a design such as this, but you must attempt to minimize this impact as much as possible. A nice reference on this aspect of the design is that provided by W0QE^{vi}, "Coupling Between Coils or Coefficient of Coupling".

The final test was performed with the complete the enclosure for the filter, operating approximately an hour at 1000 watts. Operation with my Alpha 8410 amplifier at the 1500 watt level has revealed no problems whatsoever.

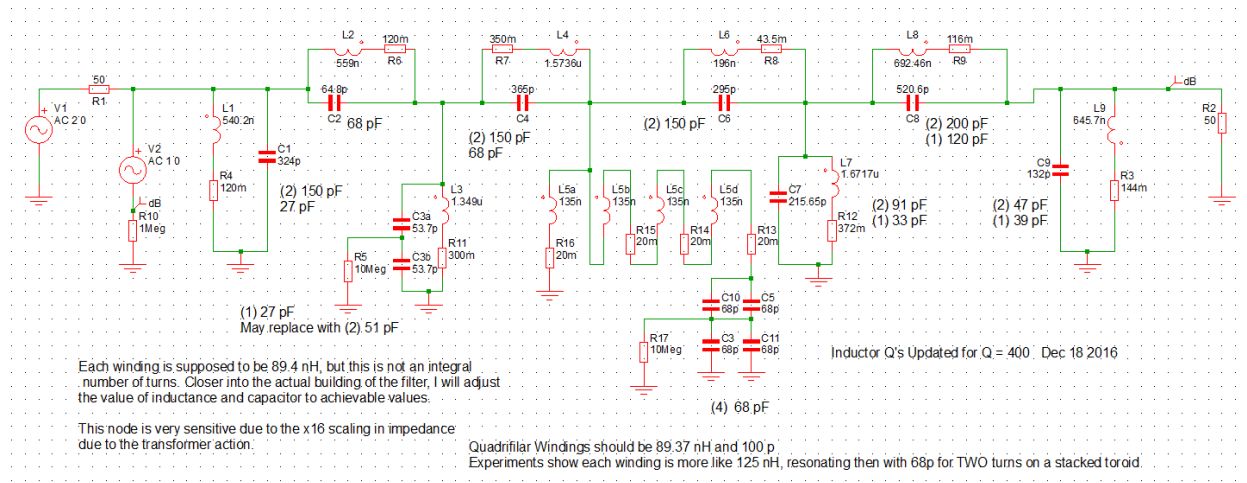


Figure 29 As Constructed Schematic

All capacitors are CDV16 or CDV19 available through Mouser Electronics. The PCB is 1 ounce copper.

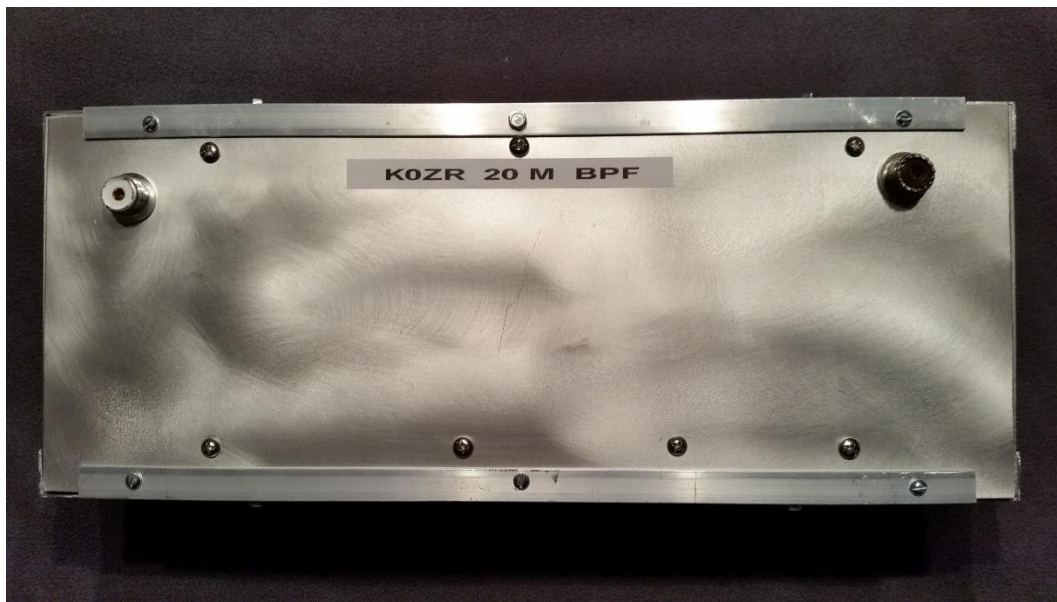


Figure 30 BPF in Homebrew Aluminum Housing

When placing the filter in the housing shown in Figure 30, there was no detectable change in passband insertion loss or return loss, nor in the stopbands. The enclosure measures (LWH) 15"x6"x4".

Other Filters to Design

At this juncture, filters for 160, 80, 15, and 10 remain to be designed and constructed. I am in the midst of building the 80m filter. This filter and likely that for 160m, will be unique from the others in that I believe more toroids will be used, with possibly no toroids in the other filter designs. Tuning is greatly facilitated if one can eliminate toroids from use as they are quite inflexible, physically and electrically, once wound. While I could possibly build the 80m with only one toroid (the resonator to ground in the filter's center), I plan to use a bifilar T-200 Mix 2 toroid to realize one of the inductors that was rather large; ~ 11 uH. I am striving to keep all filters in the same (LWH) 15"x6"x4" footprint.

I am also planning to make the 160 and 80m filters "brute force capable" of 2 KW or more. It is likely that VSWR may enter in more during a contest where one is stretching the limits, and as we know, amplifiers put out more power lower in frequency and I want to ensure sufficient headroom is available in general.

In Closing

If you encounter mistakes, have helpful suggestions, etc, I would like to hear about it so that I may update this document and hopefully not be a source of confusion for future readers. Currently my email address is kzerozr@gmail.com, but in all likelihood the email associated with QRZ.com will always be current.

Appendix

Transformer Ratios

r:1 or 1:r? Do you get confused; so do I unless I have been working with the definitions for a time. Otherwise I have to rederive what these mean. Perhaps the following words will help.

When $r > 1$, the impedance associated with that side of the transformer is highest. In other words, considering r:1, if $r > 1$ the primary impedance is higher than the secondary. Conversely if $r < 1$ in r:1, the primary impedance is lower than the secondary.

Similarly for the ratio 1:r, if $r > 1$, the impedance on the secondary side is higher than the primary. Conversely, if $r < 1$ the impedance on the secondary side is lower than that of the primary.

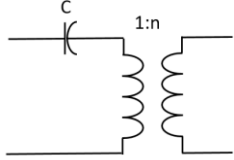
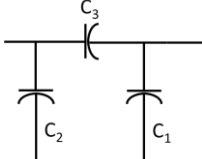
Follow which side of the “:” sign r is on and whether or not r is greater than or less than 1.

Negative Values After Norton Transformation

The side of the pi network where negative values appear is entirely governed by the impedance on that side of the transformation. The capacitor on the higher impedance side of the transformation is always negative. You can verify this by programming in the simple relationships for the first Norton transformation proven herein.

Impact on Norton Transformation if Transformer Ratio is 1:n Rather Than n:1

Rather out of my own curiosity I derived the Norton transformation for a series capacitor where the transformer turns ratio is 1:n rather than n:1. The procedure is the same as provided in the main text, only a different ABCD matrix for the ideal transformer.

	
$\begin{bmatrix} 1 & 1/j\omega C_s \\ 0 & 1 \end{bmatrix} \begin{bmatrix} 1/n & 0 \\ 0 & n \end{bmatrix}$	$\begin{bmatrix} 1 & 0 \\ j\omega C_2 & 1 \end{bmatrix} \begin{bmatrix} 1 & 1/j\omega C_3 \\ 0 & 1 \end{bmatrix} \begin{bmatrix} 1 & 0 \\ j\omega C_1 & 1 \end{bmatrix}$

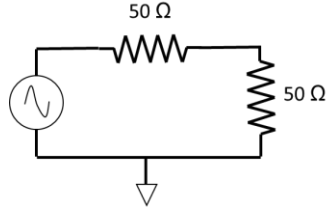
We now equate these two sets of multiplied matrices as:

$$\begin{bmatrix} 1 & 1/j\omega C \\ 0 & 1 \end{bmatrix} \begin{bmatrix} 1/n & 0 \\ 0 & n \end{bmatrix} = \begin{bmatrix} 1 + \frac{C_1}{C_3} & \frac{1}{j\omega C_3} \\ j\omega C_2 + j\omega C_1 \left(1 + \frac{C_2}{C_3} \right) & 1 + \frac{C_2}{C_3} \end{bmatrix}$$

The net result is the following: $C_1 = \frac{C_s(1-n)}{n^2}$ $C_2 = \frac{C_s(n-1)}{n}$ $C_3 = C_s / n$

Modeling in SPICE

A point regarding modeling in SPICE may be confusing, as most amateurs are likely unfamiliar with the program in general. When assembling models in SPICE, every “signal source” has its own characteristic impedance, which must be accounted for in the simulation. Consider the simple circuit below in which there are two 50 Ω resistors in series. In most cases, the “RF Source” is said to have a characteristic impedance of 50 Ω, thus the reason for the first resistor. The “Load” has an impedance, also, of 50 Ω, thus the second resistor. If the voltage source had an ideal voltage of 1 V, ½ V would be dropped across each resistor. To simply things we generally prefer to normalize the circuit to a 1 V input, so a 2 V source is required in order to drop 1 V across the load. The next section gets into more details and the reader will see again that the source is set to 2 V in order to drop 1 V across the theoretical “perfect” 50Ω load.



How to Set-Up SPICE to Produce S_{21} and S_{11} Responses

from Robert Hickey of TriQuint Semiconductor

11/12/97

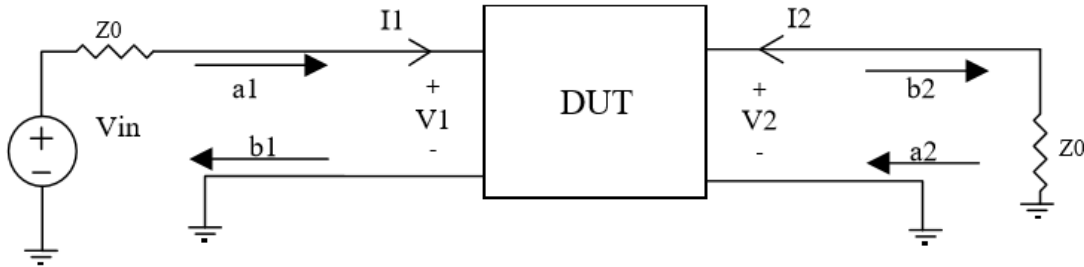
Scattering, or S parameters are easily generated by an AC analysis in SPICE. S parameters are defined with respect to incident (a_1 and a_2) and reflected (b_1 and b_2) voltage waves defined by the diagram below and by:

$$a_1 = (V_1 + Z_o I_1) / (2 \sqrt{Z_o}) \quad [1]$$

$$a_2 = (V_2 + Z_o I_2) / (2 \sqrt{Z_o}) \quad [3]$$

$$b_1 = (V_1 - Z_o I_1) / (2 \sqrt{Z_o}) \quad [2]$$

$$b_2 = (V_2 - Z_o I_2) / (2 \sqrt{Z_o}) \quad [4]$$



The scattering matrix relates the incident and reflected waves.

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} \quad [5]$$

$S_{11} = \frac{b_1}{a_1} \Big|_{a_2=0}$ a_2 is the wave reflected from the load toward the output port and is zero when the load is the impedance used to define the S parameters (in other words, everything is matched). Hence,

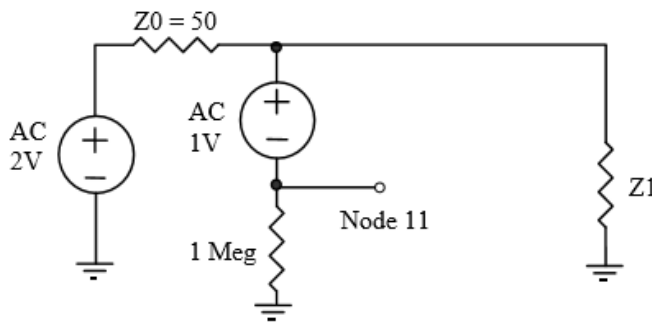
$$S_{11} = \frac{V_1 - Z_o I_1}{V_1 + Z_o I_1} = \frac{Z_1 - Z_o}{Z_1 + Z_o} \quad \text{since } Z_1 = Z_{in} = \frac{V_1}{I_1} \quad [6,7]$$

Let the Voltage Source be 2 Volts. This value simplifies calculations without compromise, for an AC run in SPICE is a small signal analysis. The voltage at the network input is then found from a voltage divider as:

$$V_1 = \frac{2Z_1}{Z_1 + Z_o} \quad [8] \quad \text{Solving for } Z_1 \text{ in terms of } V_1 \text{ results in: } Z_1 = \frac{V_1 Z_o}{2 - V_1} \quad [9]$$

Substituting this in Eq. 7 yields the simple result: $S_{11} = V_1 - 1$ [10]

This is more intuitive than many S parameter results. If a 2 volt source is used at the input, and if the input is perfectly matched, V_1 will be 1 Volt. Subtracting 1 from that leaves S_{11} of zero, a “proper” result for a perfect impedance match. In SPICE, 1 is easily subtracted from V_1 by attachment of a 1 Volt AC generator to the V_1 node. The voltage at node 11 below is S_{11} in both magnitude and phase.



S_{21} , the forward scattering parameter, is defined as $S_{21} = b_2 / a_1$. Using equations 1 and 4, this becomes

$$S_{21} = \frac{V_2 - I_2 Z_o}{V_1 + I_1 Z_o} \quad [11]$$

S_{21} is defined by Eq. 11 for the case when there is no reflection from the load, setting $a_2 = 0$. This produces $I_2 = -V_2/Z_o$, and Eq. 11 reduces to

$$S_{21} = \frac{2V_2}{V_1 + I_1 Z_o} \quad [12]$$

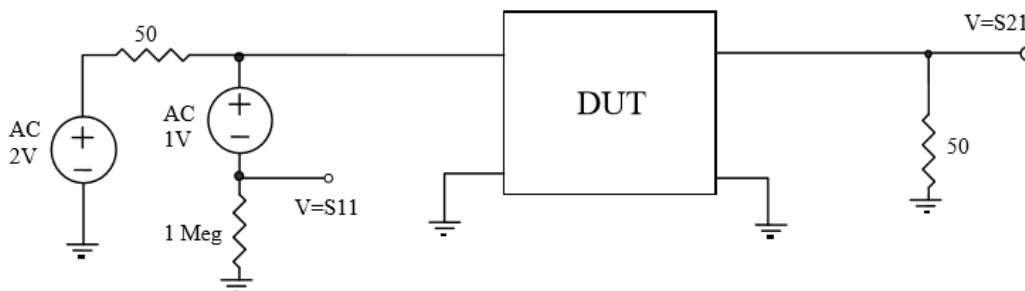
Consider our special case for a 2 volt driving generator. I_1 , the input current, is related to the input voltage, $I_1 = V_1 / Z_1$. Substitution into Eq. 12 yields

$$S_{21} = \frac{2V_2 / V_1}{1 + Z_o / Z_1} \quad [13]$$

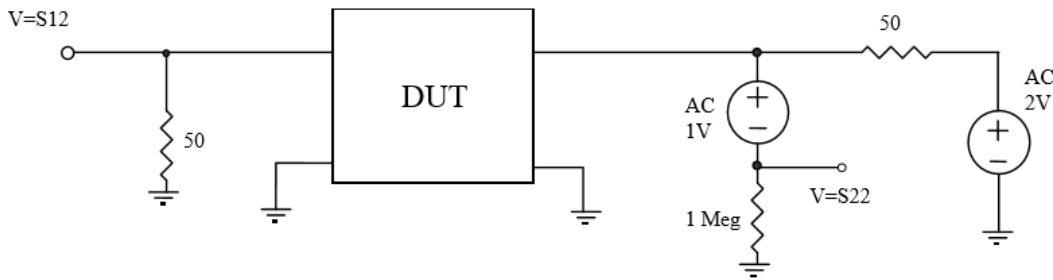
Equation 9 relates Z_1 to V_1 ; substitution simplifies to the almost trivial result that

$$S_{21} = V_2 \quad [14]$$

Hence, S_{21} and S_{11} are calculated in SPICE from



And similarly for S_{12} and S_{22} :



Power Considerations for Stacked Toroids

I had several Amidon cores around 1.25" in diameter and wanted to use them if possible in this design. The cores used here are Amidon T-130, MIX 17. For one core the A_L is 40, but for two stacked this becomes 80. The relationship to determine the maximum flux density in the cores is the following:

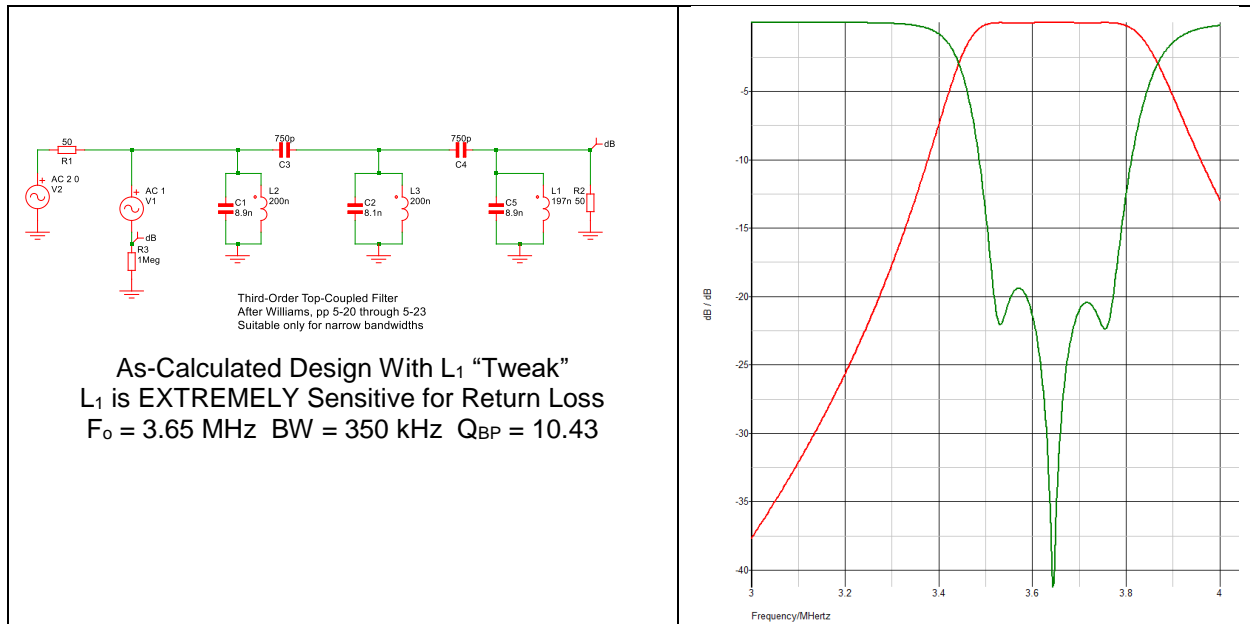
$$B_{MAX} = \frac{E \times 100}{4.44 \times A_e \times N \times F} \quad \text{where} \quad \begin{matrix} E = V_{rms} \\ N = \# \text{ turns} \end{matrix} \quad \begin{matrix} A_e = \text{cross-sectional area, cm}^2 \\ F = \text{frequency, MHz} \end{matrix}$$

In this design, two turns of four wires are used, thus $N = 8$ in the formula above. At 14 MHz Amidon recommends a maximum B_{MAX} of 42 gauss and my calculations show about 17 gauss will be developed at the full power level.

The Q-K Filter Design Method

This is an interesting filter design method which appeared very attractive from the standpoint of physical implementation. The literature cautions that this is relatively narrowband, on the order of 10% bandwidth at the maximum, but I wanted to give it a try regardless. If one restricted themselves to the CW-only portion of 80m, this filter could be very attractive. Rhea recommends correction factors of Cohn and Matthaei^{vii} for filters exceeding 5 % bandwidth. Williams^{viii} provides all the design equations on page 5-22 of his book.

The design below had the last resonator "L" value tweaked only 3 nH and it made a huge difference in the return loss across the band. The sensitivity of the first and last inductors is very high, thus making one very cautious about this implementation. Rather than let the reader find this out for themselves, I analyzed the filter below for a 1500 watt input. Mind you, the inductor and capacitor currents ranged from 80 to 140 AMPS ! Clearly some serious work remains on this design, and it is questionable if after serious transformations are applied, whether current levels can be returned to reasonable levels. (This is why I did not go to the trouble of documenting the design equations here)



Some additional useful references, although not cited directly in the main body of this writeup, include:

G30TK, Richard Harris, Bandpass Filter Design, parts 1 and 2

Wikipedia, Equivalent Impedance Transforms

Rhea, Randall, "Transforms Aid the design of Practical Filters", Applied Microwaves & Wireless, available on the Agilent Technologies website: www.agilent.com/find/eesof

ⁱ Rhea, Randall, *HF Filter Design and Computer Simulation*, McGraw Hill, Inc., 1995, Pp 169, 172, 174

ⁱⁱ Helszajn, Joseph, *Synthesis of Lumped Element, Distributed and Planar Filters*, McGraw Hill, 1990

ⁱⁱⁱ Simetrix, available from the U.K. A free version capable of ~ 100 components is available

^{iv} Mouser Electronics, www.mouser.com

^v Klappenberger, Albert, "Narrow Bandwidth Elliptic Bandpass Filters with Low Parts Value Spread," *RF Design*, December 1992

^{vi} W0QE, Larry Benko, http://www.w0qe.com/Technical_Topics/coupling_between_coils.html,

^{vii} Matthaei, Young, Jones, *Microwave Filters, Impedance-Matching Networks, and Coupling Structures*, Artech House Books, 1980

^{viii} Williams, Arthur B., *Electronic Filter Design Handbook*, McGraw Hill, 1981