

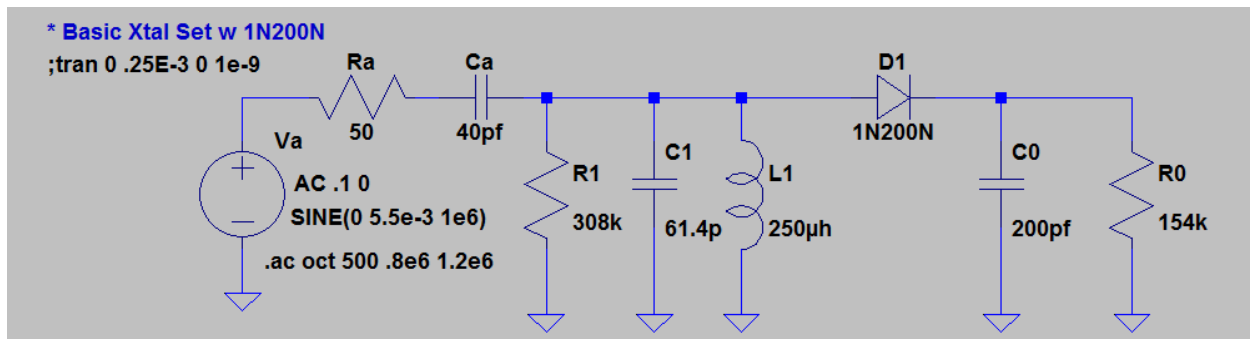
## Anatomy of Crystal Set Modeling: Equivalent Circuit Substitutions

By Phil Anderson, WØXI

In the last half-century the tools used to understand and model RLC & RF circuits, primarily linear or piece-wise linear, were simultaneous loop and node equations and “circuit network” substitutions. The substitutions we have in mind are current sources in place of voltage sources, an R and C in parallel for an R and C in series or vice versa at a specific frequency, and so on. These network tools are still taught in introductory electronics courses. With the advent of high-powered home computers, circuits may now be entered in schematic form and captured into a circuit analysis program, such as one of the many spice versions. With a single command, the “designer” can run a simulation of a circuit to see what results at the output for a given voltage or current input. While the program can save tons of work, it does not explain – in a deductive way – how the circuit works. Productivity and understanding will usually more than doubled if bench time, design experience, and the use of the “old” substitution methods of analysis are added to the designer’s toolbox. Network substitutions often have a way of reducing the complexity of a circuit and presenting the designer with another “Ah Ha!”

The analysis of the traditional crystal set is no exception. Let’s analyze a sets operation using the tools mentioned above. Let’s start with a schematic – a model if you will – of a basic set, including a capacitively coupled antenna schematic, as noted in Figure 1.

Figure 1:



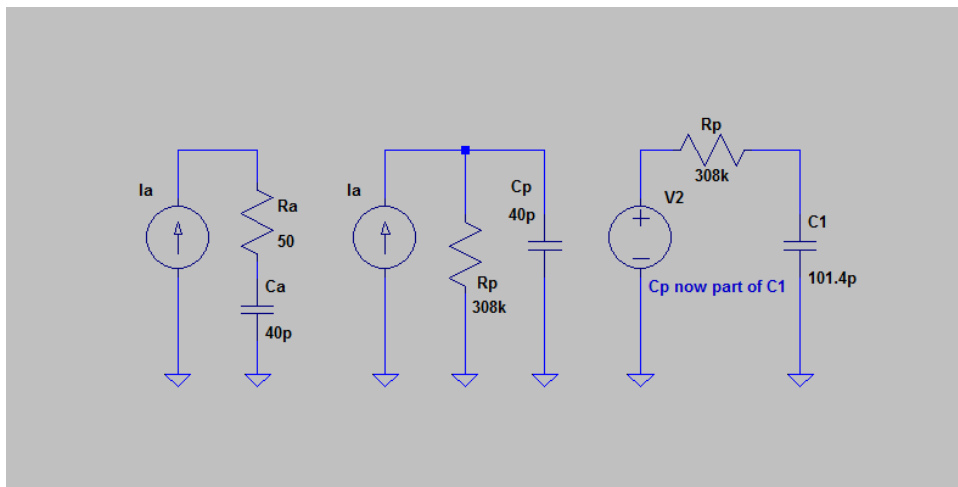
From left, the antenna model consists of a voltage source,  $V_a$ , ground and antenna resistance,  $R_a$ , and capacitance,  $C_a$  – which may represent a cap in series with a larger antenna capacitance. The traditional RLC parallel tuned circuit has  $R_1$ ,  $C_1$  and  $L_1$ . The detector and simulated load circuit includes a germanium diode,  $D_1$ , blocking capacitor  $C_0$ , and resistive load  $R_0$ . The component values shown closely model an intermediate performance set tuned to 1 MHz. The text, such as “;TRAN 0 .25e-3.0 1e-9,” are commands that the spice program will use.

We'll not cover those here. Diode D1 has a label of 1N200N, denoting that it has a saturation current of 200 nA. This is my notation for the spice model of the diode, stored in a file with the program. It is not the part number of any commercially available diode.

A typical schematic doesn't show the distributed antenna parts, simply a symbol for the antenna and wire attaching it to the top of R1. In addition, a specific set might include a transformer or two in combination with hi-impedance phones or an ear piece in place of the simulated load resistor R0. At this point, I'm using a resistor, R0, to keep the model general.

Now let's make our first "circuit network" substitution; and, keep in mind that through a series of substitutions the set will get simpler (show fewer parts in the schematic) and perhaps reveal some of its secrets. The voltage source, Va along with Ra and Ca, in Figure 1 is replaced by a current source, Ia, as noted at the left of Figure 2. See any introductory electronics text for a proof and conditions of use of source substitutions. For now, just assume one can do this.

Figure 2:



Va is replaced by Ia by shorting the output of Ca to ground and calculating the current for Ia. The same series Ra and Ca become the impedance in parallel with the current source.

(1.1)

$$I_a = \frac{V_a}{R_a - jX_a} = \text{approximately } \frac{V_a}{X_a}, \text{ since } R_a \ll X_a.$$

Our second substitution is made by converting Ra and Ca into an equivalent parallel circuit of Rp and Cp - at just one frequency - as noted at the center of Figure 2. The formulas for these conversions are as follows:

(1.2)

$$R_p = \frac{R_a^2 + X_a^2}{R_a} = \text{approx } \frac{X_a^2}{R_a}, \text{ since } R_a \ll X_a; \text{ so note that } X_a = \sqrt{R_p R_a}.$$

(1.3)

$$X_p = \frac{R_a^2 + X_a^2}{X_a} = \text{approx } X_a, \text{ since } R_a \ll X_a.$$

Since  $R_a$  is so small compared with  $X_a$ , the reactance of  $C_a$ , at AM band frequencies, both formulas are simplified. In fact, to a good approximation, the series capacitance is roughly equal to the parallel capacitance. Here are the details for  $R_p$ :

(1.4)

$$R_p = \frac{1 / (2\pi fC)^2}{R_a} = \frac{(1 / (2\pi * 1e6 * 40 * 1e-12))^2}{50} = \frac{3920^2}{50} = 308k$$

Finally, we'll complete a third substitution and move some parts around. First, let's move the capacitor  $C_p$  over to be in parallel with the tank capacitor,  $C_1$ , as noted in the right-hand portion of Figure 2. Finally, we'll convert the current source back into a voltage source.  $I_a$  in parallel with  $R_p$  becomes  $V_1$  in series with  $R_p$ . When we make this substitution, the voltage of this new source is larger than the value of the original source,  $V_a$ . Here's how to calculate it.

(1.5)

$$V_1 = I_a R_p = \frac{V_a}{X_a} (R_p) = \frac{V_a}{X_a} \frac{X_a^2}{R_a} = \frac{V_a X_a}{R_a} = \frac{(5.5e-3)(3924)}{50} = 0.432 \text{ volts.}$$

So, the new source, through substitution and combination, is shown in Figure 3. Running the spice (simulation) program as noted in Figure 1 or 3 gives exactly the same results, i.e. the AC waveform at the top of the tank and DC voltage attained at  $R_0$ .

What insights can we now draw about our crystal set so far?

- The original source – including the 50 ohms and 40 pf in series – has been transformed to a higher voltage source in series with a much higher impedance. The series cap has

assisted in matching the resistance of the antenna to the resistance of the LC tank at resonance. Note the following:

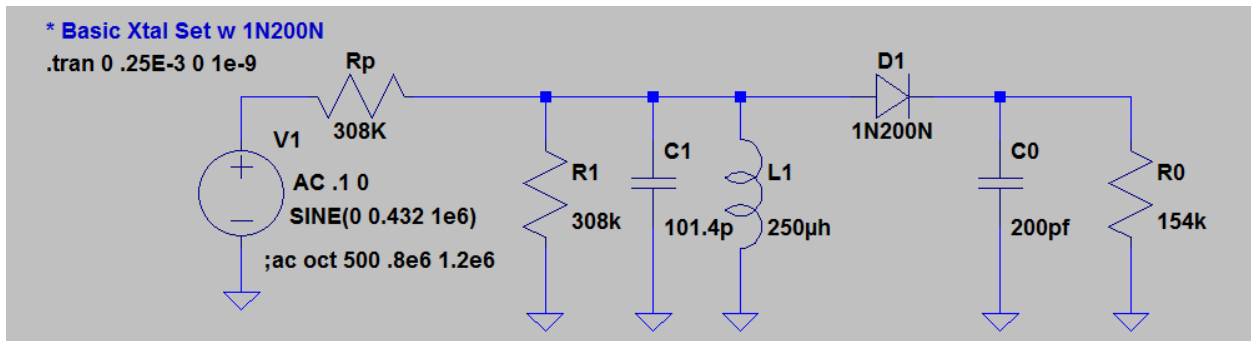
(1.6)

$$\frac{V_1}{V_a} = \frac{0.432}{5.5e-3} = 78.5 = n = \sqrt{\frac{R_p}{R}} = \sqrt{\frac{308k}{50}}$$

In effect, the series to parallel combination acts like a transformer!

- The capacitance of the antenna circuit has been added in parallel with the tank capacitance. That capacitance is part of the total capacitance that sets the resonant frequency of the set.

Figure 3:



We take one last step, converting V1, Rp and R1 into another equivalent source, resulting in Figure 4. The value of V2 and Rp in parallel R1– denoted as R2 - are calculated as follows.

(1.7)

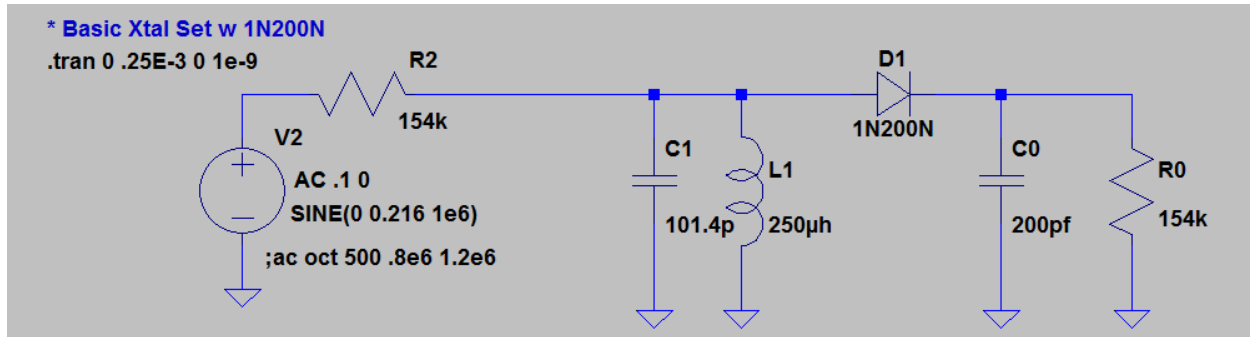
$$V_2 = V_1 \frac{R_1}{R_1 + R_p} = (0.432) \frac{308K}{308k + 308k} = 0.216, \text{ and } R_2 = R_p \parallel R_1 = 154k.$$

While Rp and R1 are the same in this example, they don't need to be in real life. A tank circuit can have one resistance at resonance while the transformed antenna resistance might have another.

Figure 4 is the spice circuit often shown for a crystal set. Clearly, this schematic hides some physical aspects of a real crystal set, shown more fully in Figure 1. Yet, both circuits produce the

same results given equal assumptions made. Figure 4 is handy in investigating, for example, what happens when the source resistance shown – which is actually a combination of transformed antenna resistance and tank resistance – is varied while holding the antenna source and antenna resistance constant.

Figure 4:



Let's look at a specific example. Assume the diode has a saturation current of 200 nA and an  $m$  of 1.2, pretty typical but a bit better than a 1N34A. Its cross-over resistance will be

(1.8)

$$R_x = \frac{0.0257m}{I_s} = \frac{0.0257(1.2)}{200e-9} = 154K.$$

Also assume that the circuit is always tuned to resonance at 1 MHz; that the voltage source in the antenna circuit along with its resistance are held constant at  $v=5.5e-3$  volts and 50 ohms respectively; and that the transformed antenna resistance is calculated to match the tank resistance. We'll show through simulation that the maximum power delivered to the simulated load resistor, R0, also equal to 154K, occurs when the source resistance (as defined by Figure 2) matches  $R_x$ . We'll vary the total source resistance, labeled R2 in Figure 4, from 75K to 185K in steps and measure tank voltage, output DC voltage at  $V_o$ , power input to the detector and power to the load, R0.

In order to use the schematic of Figure 4 accurately, we'll need to calculate the value for V2 used for each value of source resistance, R2, used. We cannot use V2 as shown for the optimal condition in Figure 4 for all input resistances, since that value would not reflect changes in the transformed antenna resistance. We can leave C1 at 101.4 pf, however, since it represents the adjusted value of  $C_a$  placed in parallel with the adjustable C1 – to resonate with L1 at 1 MHz. The following is a spread sheet of values calculated for V2 given total R2, using the various equations already covered.

Table 1:

<u>R2</u>	<u>Rp</u>	<u>Xa</u>	<u>C req'd</u>	<u>V2</u>	<u>Vp</u>	<u>Vo</u>	<u>Pi</u>	<u>Po</u>	<u>Po/Pi</u>
25000	50,000	1,581	101	0.087	62.9	14.300	12.900	1.330	10.3%
55,000	110,000	2,345	68	0.129	65.5	28.400	28.800	5.200	18.1%
75,000	150,000	2,739	58	0.151	69.3	31.400	32.500	6.400	19.7%
115,000	230,000	3,391	47	0.187	71.8	33.300	35.000	7.200	20.6%
135,000	270,000	3,674	43	0.202	72.2	33.600	35.500	7.300	20.6%
154,000	308,000	3,924	41	0.216	72.2	33.600	35.500	7.300	20.6%
175,000	350,000	4,183	38	0.230	71.7	33.300	35.000	7.200	20.6%
185,000	370,000	4,301	37	0.237	71.5	33.000	34.700	7.100	20.5%

As expected, maximum power to R0 is delivered when R2 equals Rx. Note that the match is very broad –nearly flat near the max - for this example. Diode efficiency falls off much faster when R2 is much less than Rx. Tabulation of a resistor voltage divider driven by a battery shows the same broad roll off in efficiency.

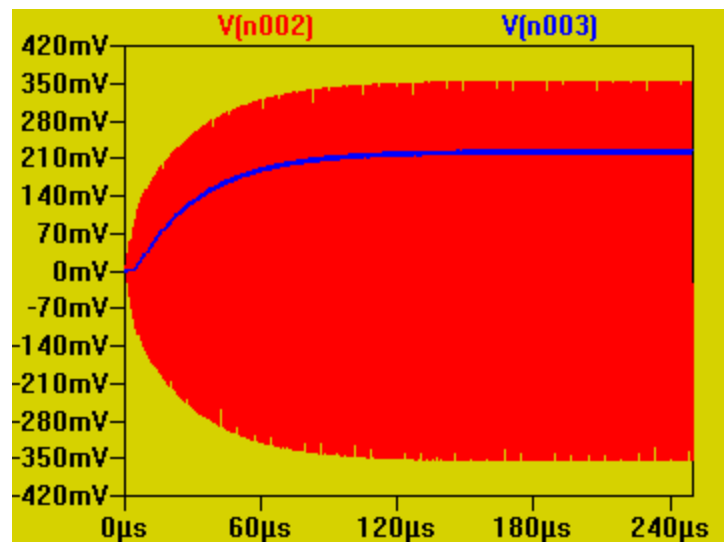
Efficiency increases, as expected when the antenna voltage is higher; here the diode does not have to work as hard. Simulations were run keeping R2 optimal but increasing the antenna voltage, Va, as noted in Table 2. At twice the input voltage, efficiency nearly doubles. After that it continues to increase but asymptotically. A graph of the voltages at the tank and output with an input of 22e-3 volts at the antenna is shown in Figure 5. That would be a very loud signal!

Table 2:

Va	<u>R2</u>	<u>Xa</u>	<u>C req'd</u>	<u>Vp</u>	<u>Vo</u>	<u>Pi</u>	<u>Po</u>	<u>Po/Pi</u>
5.5e-3	308,000	3,924	41	72.2	33.600	35.500	7.300	20.6%
11.0e-3	308	3,924	41	190	92	139	54.8	39.4%
16.5e-3	308	3,924	41	273	155	311	311	50.0%
22e-3	308	3,924	41	351	220	548	315	57.5%

We've shown that using circuit network substitutions in conjunction with a spice program helps us explain the operation of the basic crystal set. Did you have any Ah-Has? These tools can be used to dig further too, such as comparing set performance using different diodes. We'll leave that exercise for another time.

Figure 5:



General References:

- (1) Linear Technology offers a free download version of their spice program, LTspice/SwCADIII, or LTspice for short. While focused on use for simulating their ICs for product development, it can be used as a general application and works well for analyzing crystal sets! The help manual downloaded with the program is decent.  
<http://www.linear.com/designtools/software/switchercad.jsp>
- (2) Electrical Engineering Circuits, "Network Theorems," Chapter 11, 1957, J.W Wiley.
- (3) Electronic Circuits: Discrete and Integrated, "Diode-Circuit Analysis," Chapter 2, Don Schilling, 1968, McGraw Hill Book Company.