Exam #3 Rework

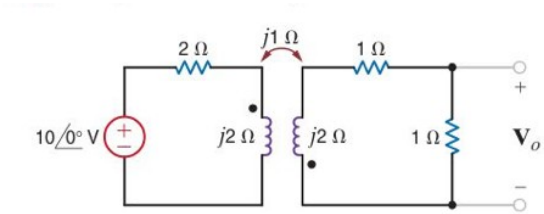
by Zack Smith

May 7th, 2013

ECE 2322: Electrical Engineering Science II

Problem #1

10.7: Find V_o in polar form (magnitude and phase).



The first step of this circuit is to write KVL equations. We already know from the picture that the mutual inductance term is j1. This creates two simple KVL equations to solve for I1 and I2.

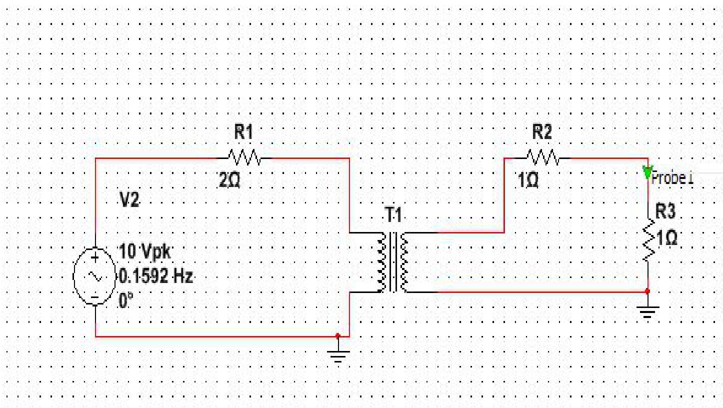
$$10=(2+j2)I_1+jI_2$$

$$0=jI_1+(2+2\mathrm{j})I_2$$

By solving these two equations for I_1 and I_2, we get I_1 = 2.7692 – 2.1538j, and I_2 = -1.2308 – 0.1538j. Since we are looking for our output voltage across the 1 ohm resistor, we can see that V_0 = IR, V_0 = I_2, therefore V_0 = -1.2308 – 0.1538j, or 1.2404 at an angle of -173.8773 degrees. We can easily verify this calculation in Matlab as shown below.

```
>> Loops = [2 + 2*j, j; 1*j, 2 + 2*j];
>> V_S = [10;0];
>> Loops \ V_S

ans =

   2.7692 - 2.1538i
  -1.2308 - 0.1538i

>> [rad mag] = cart2pol(-1.2308, -.1538)

rad =

   -3.0173


mag =

    1.2404

>> radtodeg(rad)

ans =

 -172.8773
```

In Multisim, we can use coupled inductors to properly get our mutual inductance terms. The coupled inductors allows us to write in values for each inductor, and a coupling coefficient. I know that the mutual inductance term in this circuit is equal to $M = K * \sqrt{(L_1 * L_2)}$. Since $L_1 = L_2$, and we know that at 1/2pi Hz, a 1j inductor is equivalent to a 1H inductor, the maximum mutual inductance term is the coupling coefficient (K) times L_1, which according to the circuit diagram is 2j or 2H. Therefore, if we used a coupling coefficient of 1 we would have a mutual inductance of 2j. So instead we use a coupling coefficient of .5 to get the proper mutual inductance term of 1j. The circuit diagram can be seen below.



The AC analysis of this circuit proved that our magnitude was correct, but since the coupled inductors in Multisim have no way of showing a polarity, the phase is off by -180 degrees. This is because the mutual inductance term for I_2 is backwards in Multisim. However, in my calculations it is correctly placed giving the true phase. The analysis can be seen below.

Design1

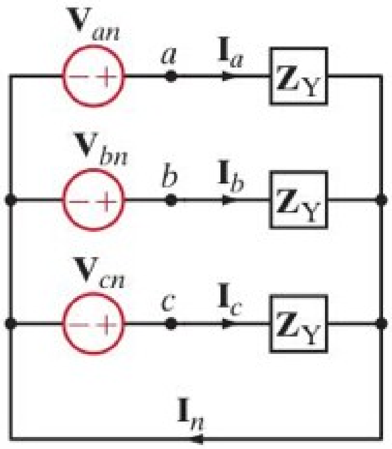
Single Frequency AC Analysis @ 0.1592 Hz

	AC Frequency Analysis	Magnitude	Phase (deg)
1	V(Probe1)	1.24039	7.11104

As you can see, the Multisim and Matlab calculations both corroborate my hand calculations.

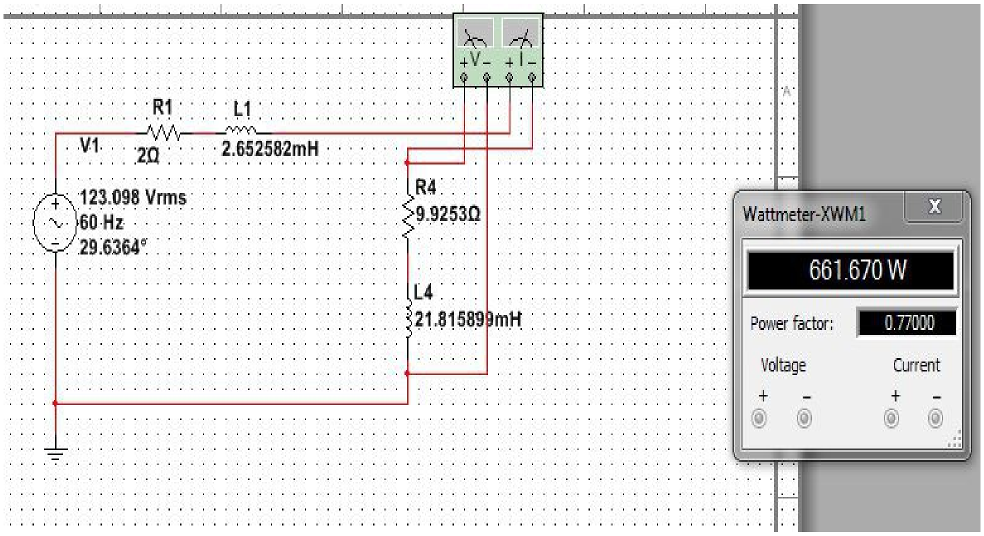
Problem #2

11.29: In a balanced three-phase wye–wye system, the total power loss in the lines is 400W. V_{AN}=105.28/31.56° V rms (the magnitude is 105.28; the angle is 31.56°) and the power factor of the load is 0.77 lagging. If the line impedance is 2 + j1 Ω, determine the load impedance. (A balanced wye-wye connection is shown to the right. The line impedance appears in series between the three-phase sources, V_{an}, V_{bn} and V_{cn}, and each of the loads.)



The key to this problem was the power lost in the lines. It's also important to realize that the power lost in a single line is $\frac{400}{3} Watts$. Using our power equation, we know I^2R = 133.333 Watts. Since R is two ohms, we can easily say $I_L = \sqrt{(\frac{200}{3})}$. From this, we can find $Z_{load} = \frac{V_{an}}{I_L}$ which finally gives us Z_{load} = 12.89. We can find the angle from the power factor, cos(.77) = 39.65 degrees.

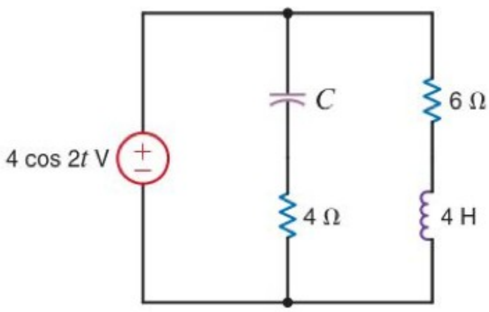
We can also use Multisim to check our answer.



Using a Multimeter for my calculated load, you can clearly see there's a power factor of .77, which matches the problem specifications. The resistor and inductor of the load reflect the Cartesian values of the previous polar representation of the load. The phase of the source is irrelevant to find the power factor of the load. At 60 Hz these inductor values represent the equivalent complex values that the problem specified. We also get a 133 Watt power dissipation across R1 and L1, which is equal to 400/3, which is what we calculated earlier. As you can see, Multisim's power factor calculations verify my own load calculations.

Problem #3

12.54. Determine the value of C in the network shown for the circuit to be in resonance



We know that from this circuit diagram that the frequency = 2 rad/s. We also know from the definition of resonance is when the impedance is purely resistive, so the imaginary impedances cancel. The easiest way to deal with this problem is to deal in admittances, since the two branches are in parallel. The equivalent admittances for this circuit is then

$$Y_{Total}=\frac{1}{\frac{1}{jwc}+4}+\frac{1}{6+4jw}$$

$$Y_{Total}=\frac{2C}{8C-j}+.06-.08\,j$$

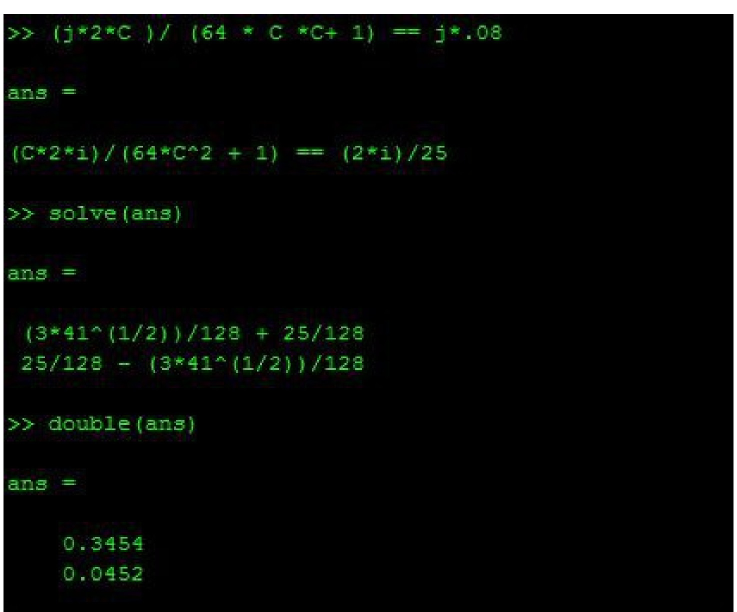
We can discard the real values, because we only need the imaginary parts to cancel. To further simplify the complex fraction, we can multiply by the complex conjugate. This gives us:

$$Y_{Total}=\frac{16C^2+jwc}{64C^2+1}$$

Since the denominator is completely real, we can cancel out the real term $16C^2$, giving us

$$\frac{2jC}{64C^2+1}-.08\,j=0$$

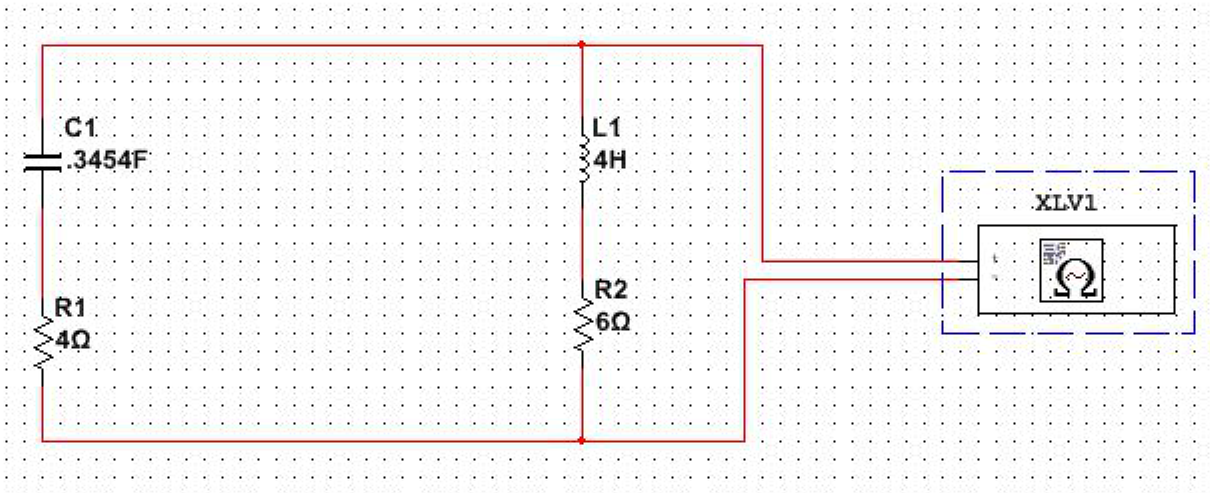
Solving for C gives us C = .3454 and C = .0452.



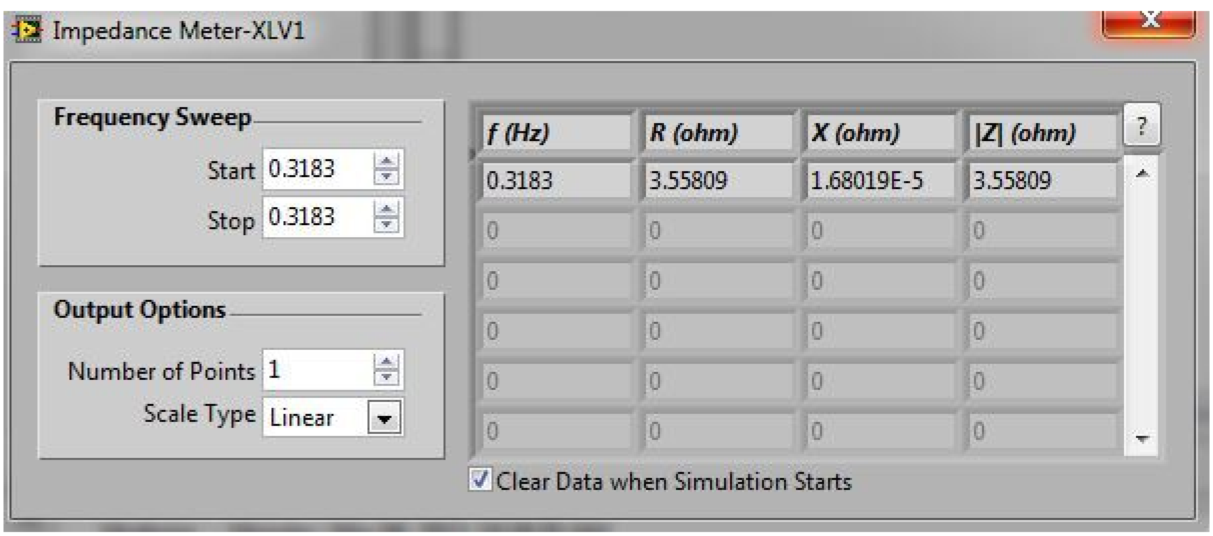
```
>> (j*2*C )/ (64 * C *C+ 1) == j*.08

ans =

(C*2*i)/(64*C^2 + 1) == (2*i)/25

>> solve(ans)

ans =

 (3*41^(1/2))/128 + 25/128
 25/128 - (3*41^(1/2))/128

>> double(ans)

ans =

    0.3454
    0.0452
```

We can verify our calculations in Matlab like so:

Finally, we can check the answer in Multisim using the impedance meter. At resonance, the imaginary impedance should be zero giving us a purely resistive circuit. Plugging our values into Multisim for our calculated capacitance, we get a circuit that looks like this:



Using our impedance meter, we can check the equivalent impedance of the circuit. The reading from the meter shows:



Our imaginary impedance is at 1.69e-5, which is pretty close to zero as far as it can be concerned. Unfortunately, our resistive aspect of the circuit is not perfect. If it were purely resistive, the impedance would be the parallel combination of the 6 and 4 ohm resistor, which would be 2.4 Ohms. Our capacitor value isn't exactly precise, but it is close enough to at least show that the imaginary impedance is nearly zero and the circuit is therefore in resonance.