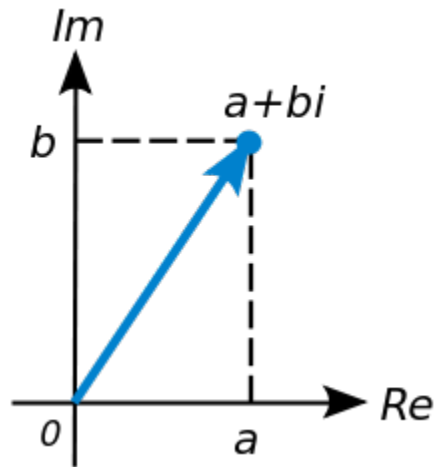


Exam #1 Rework

by Zack Smith

November 30th, 2012

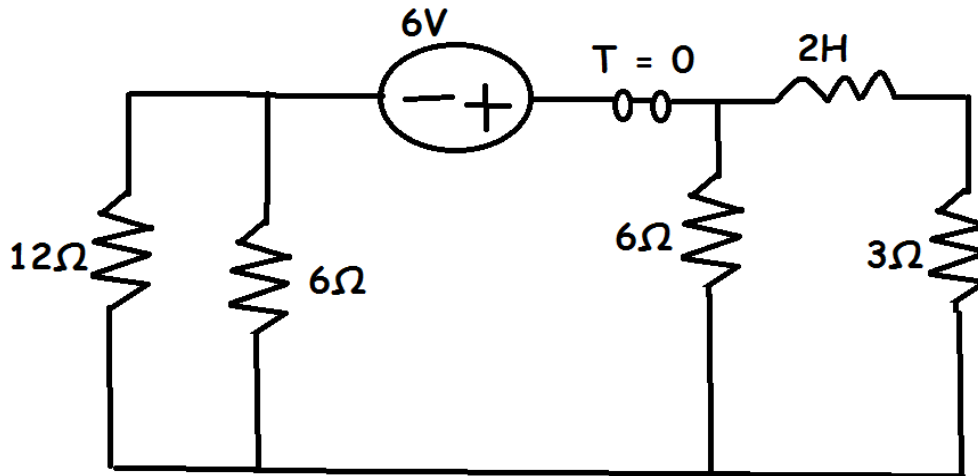
ECE 2312: Electrical Engineering Science I



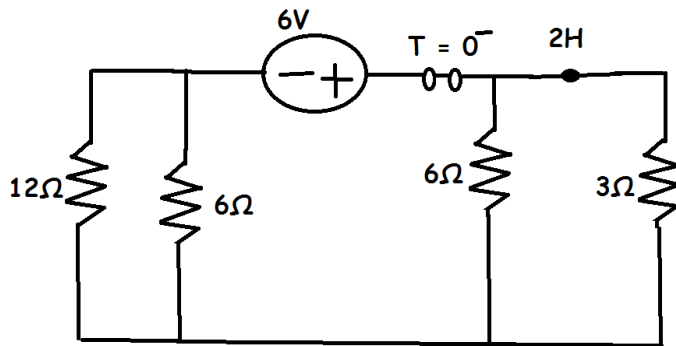
Problem #1

1. Determine the current through the inductor, $i_L(t)$, as a function of time (derive an equation).

Sketch your result for $t = [-1.0, 10]$ and label as much of the plot as possible (e.g., time constant, initial value, final value). (Hint: Write the form of the response and solve for the constants.)



The original circuit looks something like this, and we're safe to assume that the switch has been on for a long time, so the circuit is in steady state. The first step in this case is to analyze the circuit at $T = 0^-$. Since the circuit is in steady state, the inductor acts as a short circuit, so the circuit looks like this :



This circuit is just a simple circuit with 4 resistors. The purpose of analyzing at $T = 0^-$ is to determine the initial current running through the inductor, so we must solve for I_L , which is the same as the current through the 3Ω resistor. Combining all the resistors:

$$R_{eq} = R_{12} + R_{34}$$

$$R_{12} = (3\Omega \parallel 6\Omega) = 2\Omega$$

$$R_{34} = 12\Omega \parallel 6\Omega = 4\Omega$$

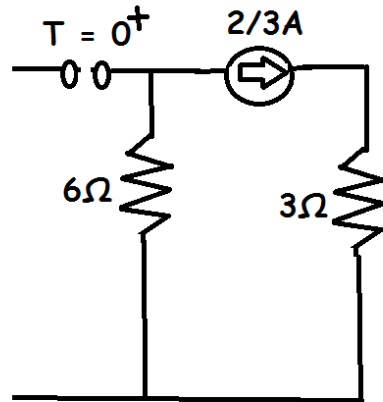
$$R_{eq} = 6\Omega$$

Therefore, $I_S = \frac{V_S}{R_{eq}}$, $I_S = \frac{6V}{6\Omega} = 1A$ Using KCL, we know $I_1 + I_2 = I_T$, so:

$$\frac{V_1}{R_1} + \frac{V_1}{R_2} = 1A \quad \frac{V_1}{3} + \frac{V_1}{6} = 1A \quad 2V_1 + V_1 = 6A \quad V_1 = 2V$$

$$I_1 = \frac{2V}{3\Omega} = \frac{2}{3}A$$

We know, therefore, that I_L must be $2/3A$ as the switch is thrown. Just as the switch is opened, at a time slightly past $T = 0$, we have this circuit:



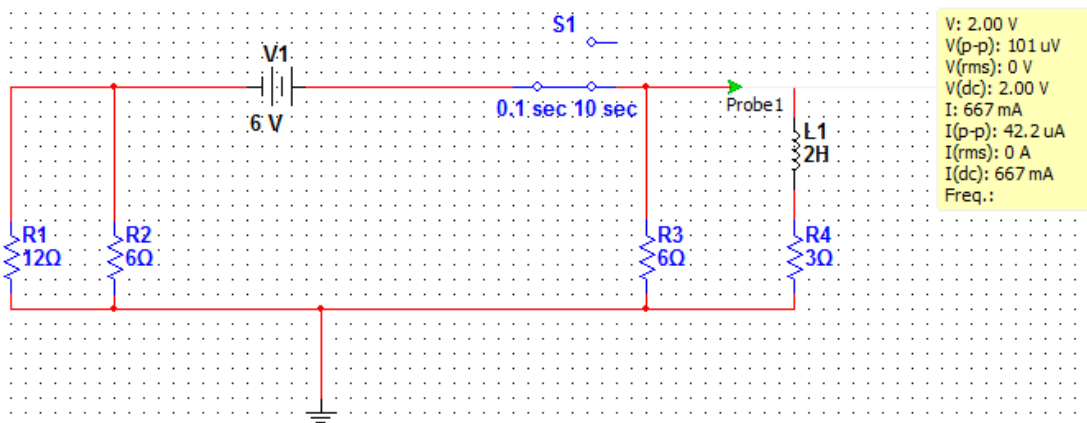
The inductor which was fully charged begins to discharge its stored potential into this new circuit, which consists of just two resistors in series. We can say that at $T = 0^+$ the current through the inductor is $2/3A$, and it can be written as a $2/3A$ source at $T = 0^+$. We can express the form of the solution as: $K_1 + K_2e^{-t/\tau}$. From this $T = 0^+$ point, we realize that $K_1 + K_2 = 2/3$.

To get our second equation for the constants K_1 and K_2 , we must look at the point of the circuit where the inductor is once again in steady state, at $T = \infty$. The circuit at $T = \infty$ is the same as the circuit at $T = 0^+$, but it is in steady state, with the inductor fully discharged, so it does not act as a source, but as a short circuit. Since there is no other source in the circuit, the current through the capacitor must be 0.

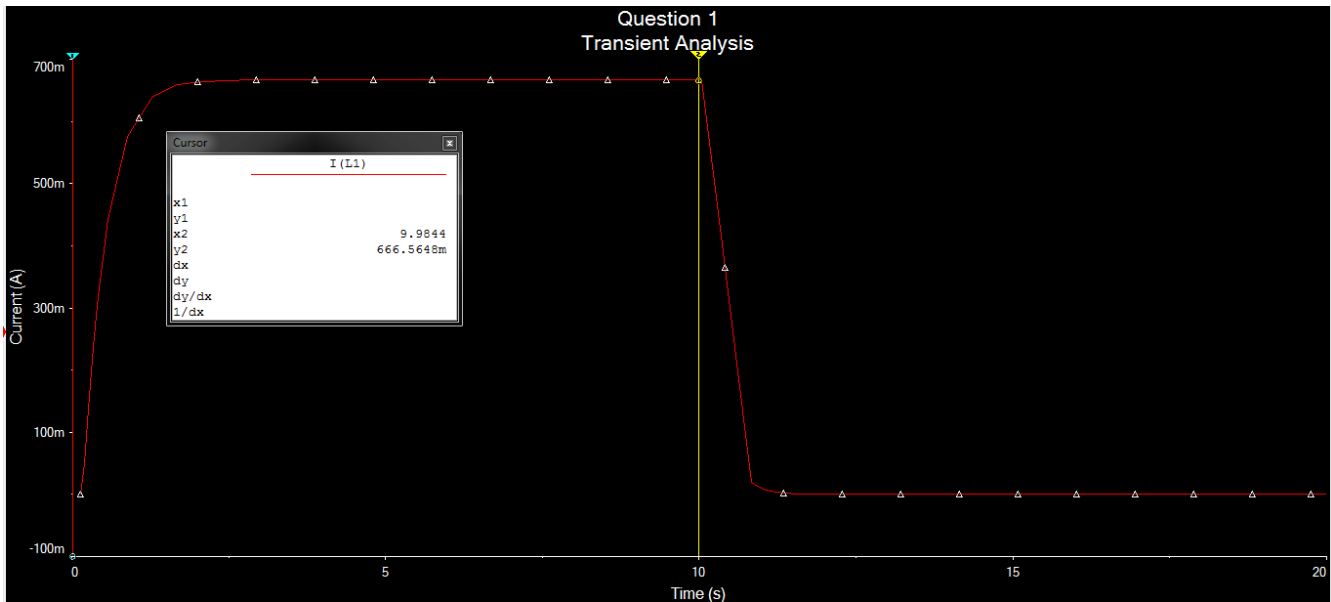
Therefore, $K_1 = 0$, and $K_2 = 2/3$. This tells us the solution is $\frac{2}{3}e^{-t/\tau}$, and we know that the time

constant, $\tau = \frac{L}{R}$, so finally we get $I(t)_L = \frac{2}{3}e^{-9t/2}$. The graph is simple, before the switch is open, the inductor current is at its maximum, $2/3A$. So before $T = 0$, the current is a straight line at $2/3$. When the switch is opened, the equation of the line is $I(t)_L = \frac{2}{3}e^{-9t/2}$, with a time constant of $\frac{2}{9}$.

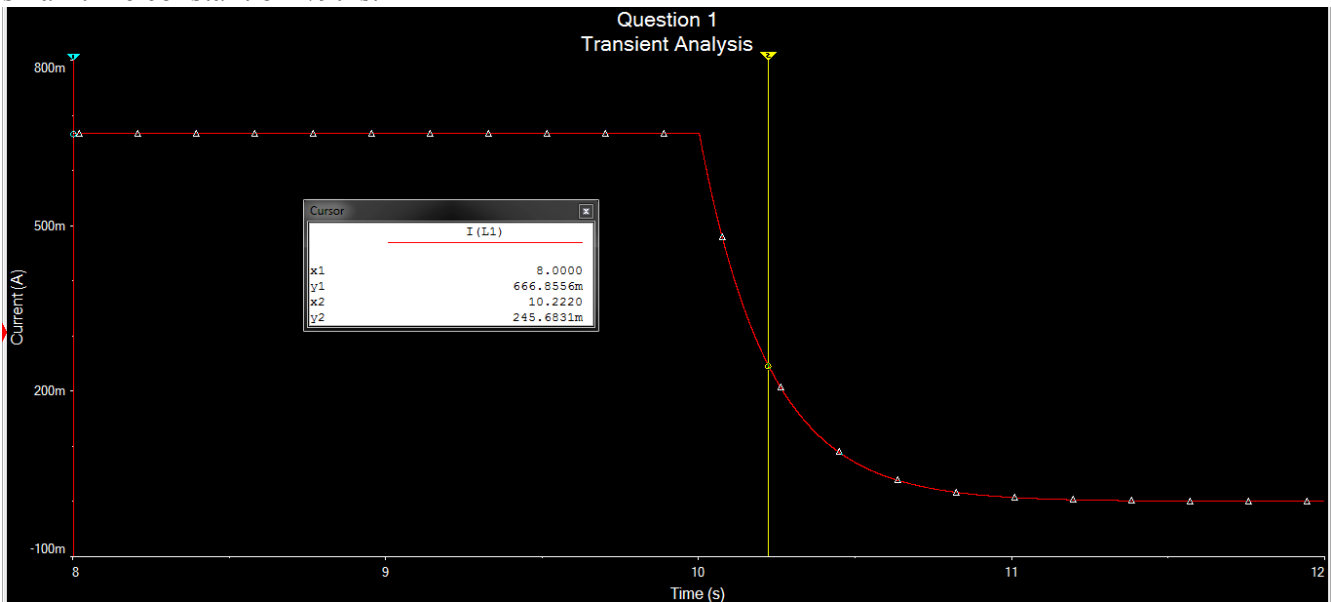
Multisim verifies this, and provides the graphing abilities.



The Multisim pictured above verifies my calculations. The simulation was running for a few seconds in Multisim time, and reached close to steady state, showing from the probe that there was 2V going through that resistor, along with 667mA, or $2/3A$.



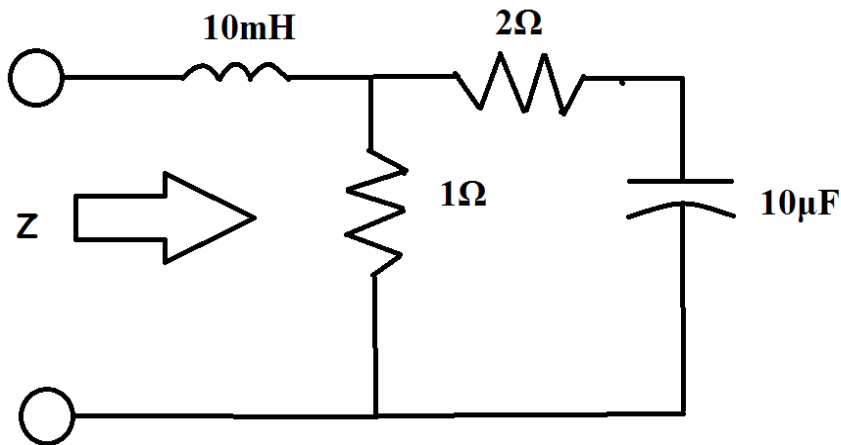
This graph shows the entire gamut of the circuit's response : First, at $T = 0$, the circuit is started and the inductor begins to charge. I allow it to charge for 10 seconds, and it reaches its steady state, at 666mA on the inductor. Then, at $T = 10s$, I open the switch and Multisim shows a sharp drop, related to our small time constant of $2/9ths$.



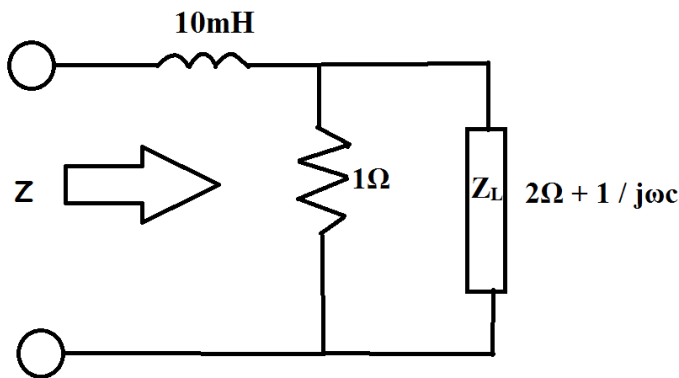
Thanks to Multisim we can look at a specific time, so in this picture I checked out the time constant value – $2/9ths$. At 10.222s, we are at our first time constant, and about 63.2% of the current should have discharged from the inductor at this point. Looking at our Multisim values we can see that the current value is 245.7 mA, which is 36.9% of 666mA. Meaning that approximately 63% of the current was discharged in that time constant, so everything checks out.

Problem #2

2. Find the equivalent input impedance assuming a frequency of 60Hz.



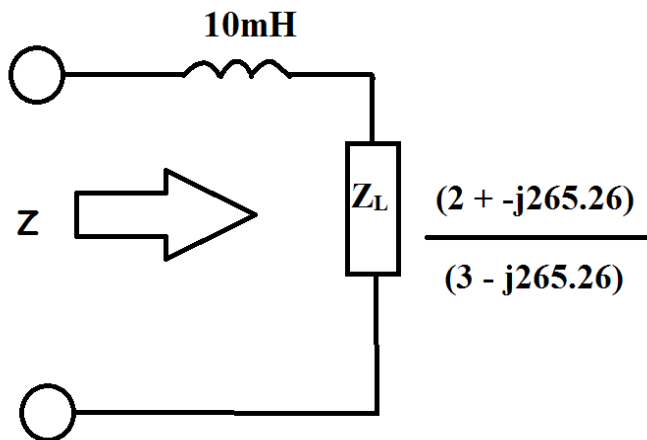
This problem is a simple equivalent impedance question, which is basically an equivalent resistance problem with complex numbers. Since our tools like Matlab can easily deal with complex numbers, it makes this question just a glorified resistance problem. It is just important to understand what the impedance for capacitors and inductors is. For an inductor, its impedance is $j\omega L$, and for a capacitor its impedance is $\frac{1}{j\omega c}$. It is also critical to know that $\omega = 2\pi f$ where f is the frequency. For the first step, the capacitor and resistor on the far right are in series. This gives us an equivalent impedance of $2 + \frac{1}{j2\pi fc}$. This gets us the circuit shown below:



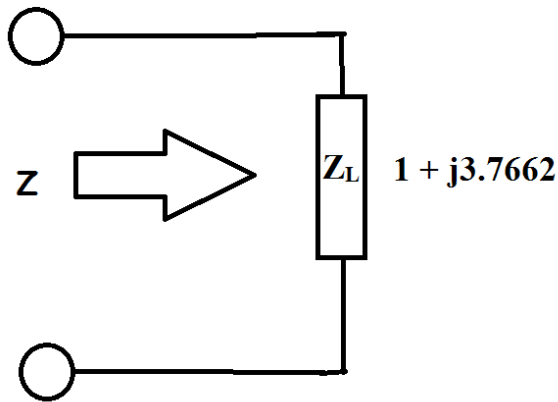
Next, the Z_L and the 1Ω resistor are in parallel, and it is simply another resistor combination

$$\frac{Z_L * 1\Omega}{Z_L + 1\Omega}$$

This gives us a new circuit:



The final circuit is obviously just the inductor and Z_L in series, so the final calculation is $Z_L + j\omega L$, giving us this final circuit and solution :



```

Command Window
>> Zc = 1 / ( j * 2 * pi * 60 * 10e-6)
Zc =
    0 -2.6526e+002i
>> Zinductor = j*2*pi*60*10e-3
Zinductor =
    0 + 3.7699i
>> Zload1 = 2 + Zc
Zload1 =
    2.0000e+000 -2.6526e+002i
>> Zload2 = Zload1 / (Zload1 + 1)
Zload2 =
    1.0000 - 0.0038i
>> ZloadFinal = Zload2 + Zinductor
ZloadFinal =
    1.0000 + 3.7661i
    
```

The complex number division required to get to this step may seem confusing, but we didn't need to know all about complex number arithmetic for this question (although it would help), we have matlab at our disposal. Simply plugging the steps I described above in to Matlab would result in our final Z_{eq} being $1 + j3.766$.

If you want to be extra fancy in Matlab, you can also convert it from its Cartesian $a + bi$ format to the polar form, you can do this :

```
Command Window

ZloadFinal =

    1.0000 + 3.7661i

>> [THETA, RADIUS] =cart2pol(1,3.7661)

THETA =

    1.3113

RADIUS =

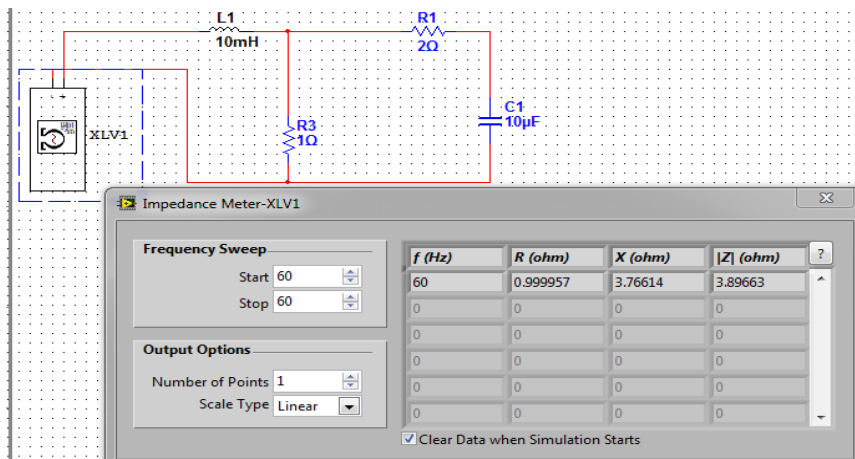
    3.8966

>> radtodeg(THETA)

ans =

    75.1296
```

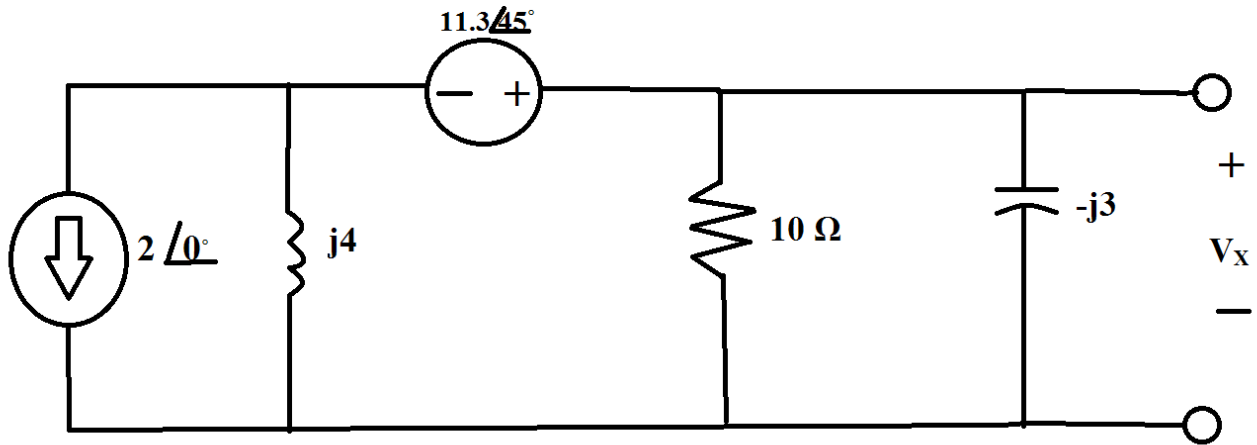
This gives us a polar form of $3.9 / 75^\circ$. Let's check what Multisim has to say about this circuit. With the aid of a labview instrument, the impedance meter, we can measure the impedance of this circuit.



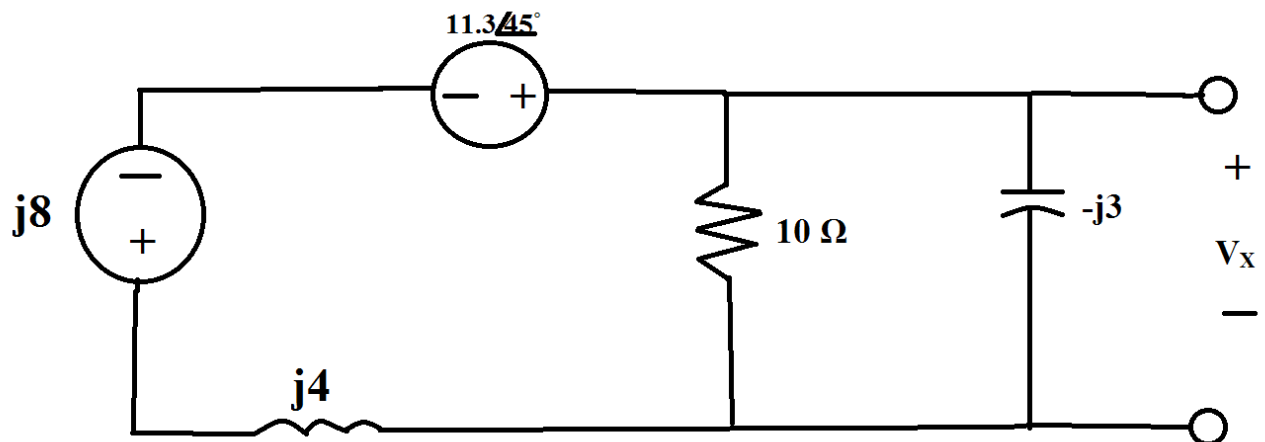
Multisim shows that the real part of the impedance is nearly 1, and the imaginary part 3.766, which is what I showed above. $|Z|$ Indicates its magnitude in polar form, which was also what Matlab showed in its conversion.

Problem #3

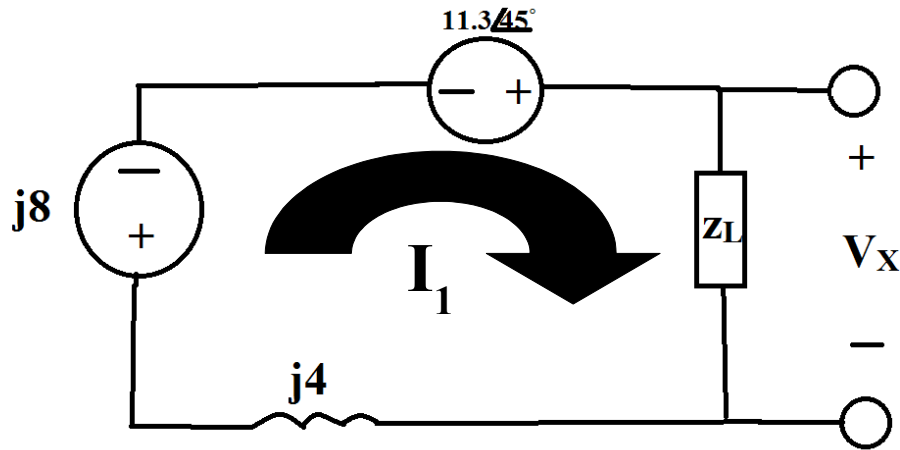
3. Find V_x (Hint: Use Norton's Theorem to convert the current source to a voltage source).



This problem was also fairly straight forward with a tip that should start your train of thought on the problem. The first step we'll take is to take the advice of the hint, and convert the current source and parallel impedance into their equivalent voltage source and series impedance. Using Ohm's Law, we know $V = IZ$, so the new voltage source must be $2A * j4 = 8j$ V, or $8/90^\circ$. The new circuit would look like this:



Also, let's combine the two impedances on the right in parallel, to make an easy, single loop KVL. We know that the Z_{eq} of that combination will have the same voltage drop as either the resistor or the capacitor, in other words the voltage drop across that Z_{eq} is V_x . The new circuit would look like this :



Our equivalent impedance expressed as Z_L is equal to $\frac{10 * (-j3)}{10 - j3}$ we plug that in to matlab and get the Z_L is equal to $.8257 - j2.752$. Let's also convert the voltage source in to its Cartesian $a + bi$ format, which can be expressed as $7.99 + j7.99$ according to Matlab.

So, let's do a KVL, which is just a singular loop:

$$-(7.99 + j7.99) + (I_1)(.8257 - j2.752) + (I_1)(j4) + j8 = 0$$

Simplifying, we get :

$$I_1(.8257 + j1.248) = 7.99 - j.1$$

Once again plugging in to matlab we get

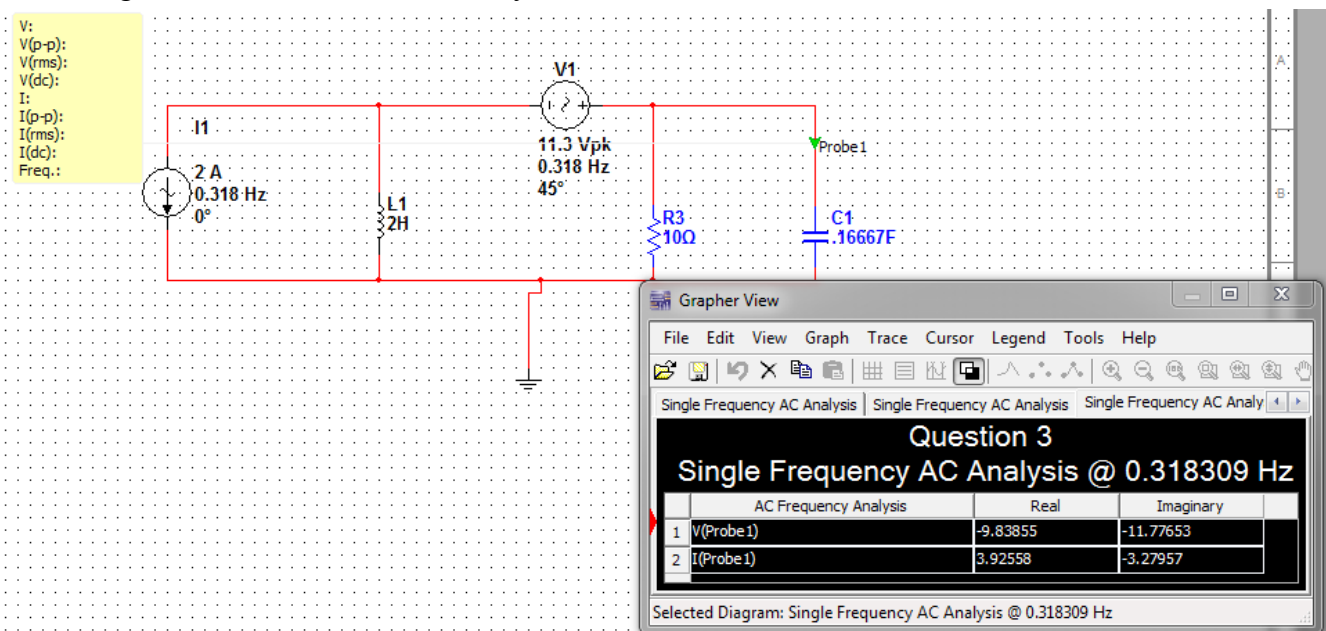
$$I_1 = 3.0019 - j4.4161$$

Finally, $V_x = I_1 Z_L$

$$V_x = (3.0019 - j4.4161)(.8257 - j2.752)$$

$$V_x = -9.822 - j11.7847, \text{ or if we want to express it in polar form : } 15.34 \angle -129.8^\circ.$$

Now, let's go see what Multisim has to say:



The Multisim verifies my values, more or less. Multisim indicates a voltage of $-9.8385 - 11.776j$, which is pretty close to my calculated value of $-9.82 - j11.78$. However, some explanation is necessary for why the Multisim is not exact, and why I have explicit capacitor and inductor values, where the

question posed expresses them as complex impedances. As far as I'm aware, there isn't a simple way to express a capacitor or inductor as a complex impedance rather than as a farad or henry value. So in order for me to get the approximate equivalent impedance, I would need to solve for their values. Of course, we all know by now, that the impedance of an inductor is given by $j\omega L$, and a capacitor's is $\frac{1}{j\omega c}$, where, of course, $\omega = 2\pi f$. Now, I could set a frequency to get a capacitance or inductance value, or I could set the capacitance or inductance to get a frequency. I decided that setting the frequency would be the easiest solution, and even easier, I could set the value of the frequency to be $\frac{1}{\pi}$, the resulting ω would be: $\frac{2\pi}{\pi}$, which makes ω just 2. With this simplification, I could find the equivalent capacitance and inductance to return the impedance values that were written on the question. Because I only took $\frac{1}{\pi}$ to so many decimals, the Multisim analysis is slightly off, but it is close enough to my points that it brings great confidence to my answer.