Exam #2 Rework

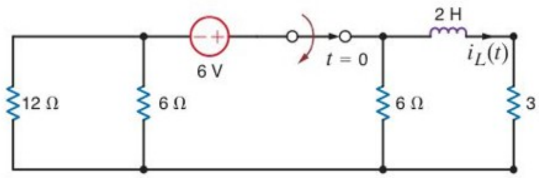
by Zack Smith

April 26th, 2013

ECE 2322: Electrical Engineering Science II

Problem #1

1.Find $i_L(t)$ for t > 0.



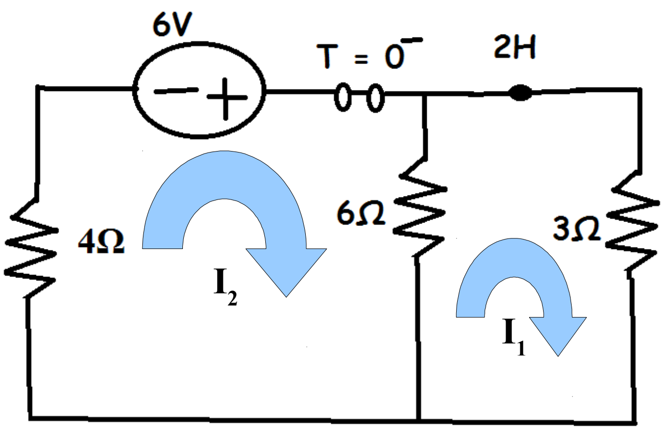
We can approach this problem as we have done before, looking at the current through the inductor at Time 0-, 0+, and at ∞.

At t = 0- :

Inductor acts as short circuit, we must find current through the inductor. By combining the two left most resistors, we can do a two loop KVL to determine the current through the inductor.

$$\frac{12*6}{12+6}=\frac{72}{18}=4\,Ohms$$

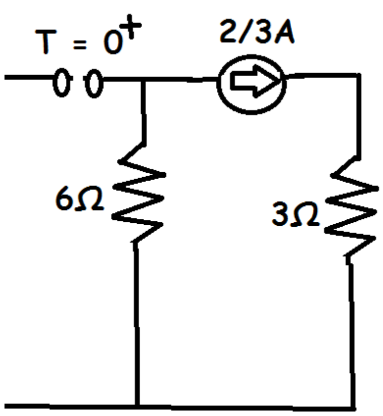
KVL:



$$3I_1+4(I_1+I_2)=6V$$
$$6I_2+4(I_1+I_2)=6V$$

By solving these equations, we get I_1 = 2/3A, so the current through the inductor at T = 0^- is 2/3A.

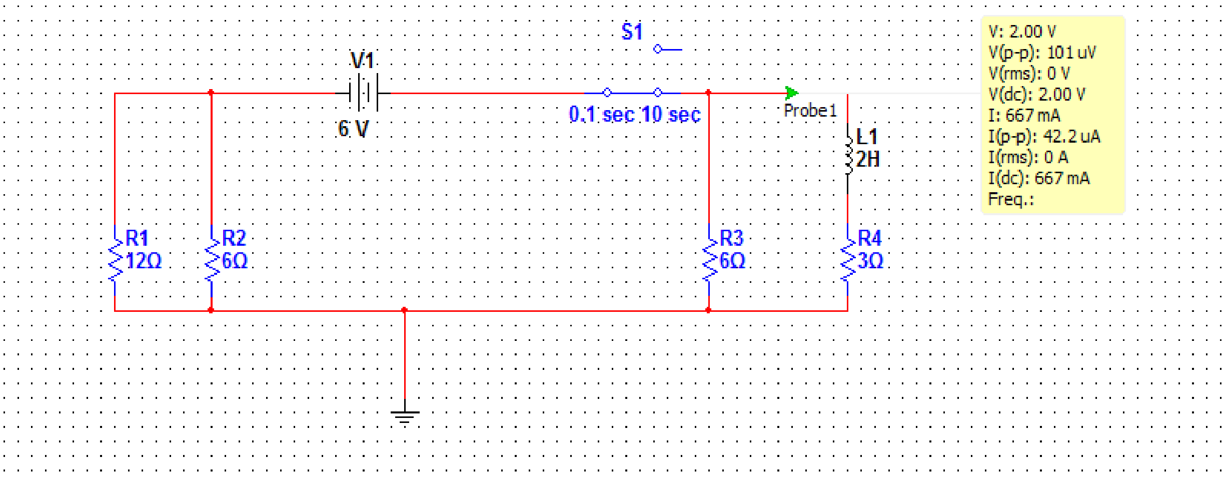
At T = 0^+, the switch is flipped, so we get a new transient equivalent circuit that looks like this:



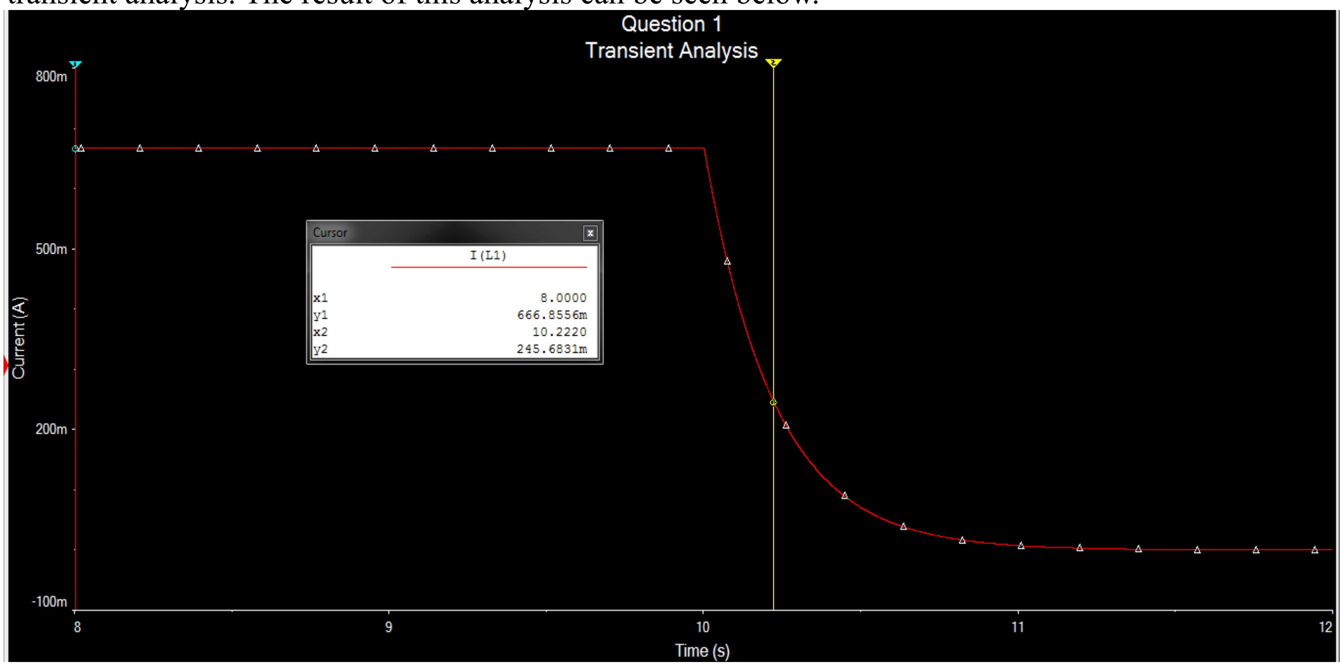
From this, we can infer the time constant of L/R is equal to 2 H / 9 Ohms. We also know the form of the solution is $C1 + C2e^{-9t/2}$. From this circuit, we can get the equation of C1 + C2 = I_L = 2/3A.

By looking at the circuit at T = ∞ we can get the second equation to find our constants. At T = ∞, Our inductor is a short once again, and since it is cut off from any source the current through it must be zero. Therefore, C1 = 0, and the final solution is: $I_L=\frac{2}{3}e^{(\frac{-9t}{2})}$

Now, let's verify that answer with Multisim.



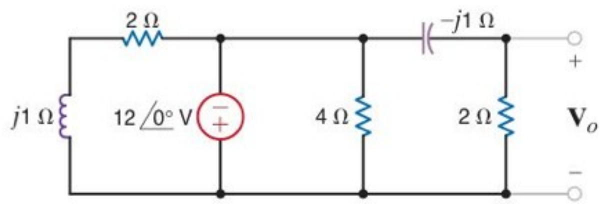
This is the Multisim circuit diagram, the probe shows steady state (before the switch is flipped), and shows 2V and .667 mA. This verifies our steady state analysis of the circuit, but it does not show the whole picture. To see the entirety of the circuit's transient response, we need to use Multisim to do a transient analysis. The result of this analysis can be seen below.



We can see from this picture that I leave the circuit in steady state from 0 to 10s, and the current through the inductor is measured at a steady 2/3A. At 10s, I flip the switch, and the transient response occurs. I placed the cursor at the first time constant past 10s. The time constant is 2/9ths, so I checked out the current value at 10.222s. After one time constant, 63.2% of the current should have discharged. Multisim gives us a value of 245.7mA, which is 36.9% of the initial starting value of 666mA. This means about 63% of the current was discharged, and this checks out with the 63.2% ideal.

Problem #2

2.Find Vo (as a phasor).



The first step of this problem is to realize that through each branch there is equal voltage, because they are all in parallel. This is fairly clear just by inspection. That simplifies the circuit beyond imagination, because now we can just do a voltage divider on the branch with V_o. The voltage divider equation will be:

$$V_0 = \frac{2}{2 - j1} * -12\text{V} \quad .$$

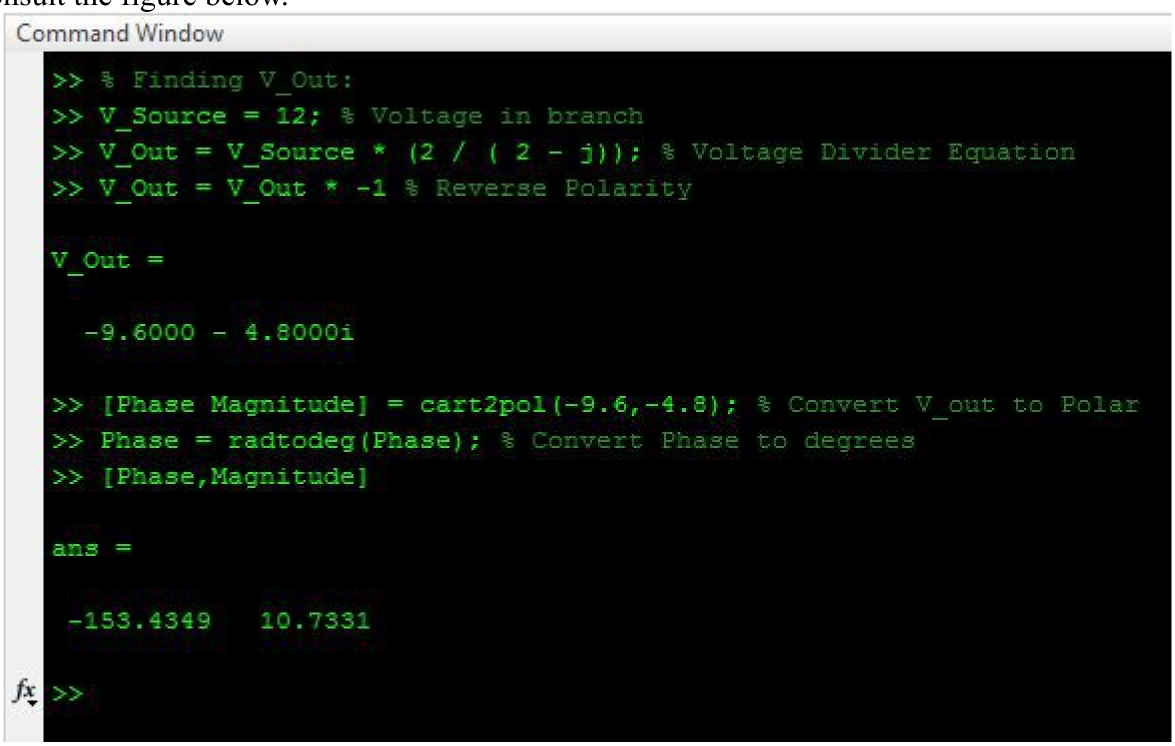
Doing the math out, we will get $V_0 = -9.6 - 4.8j$. To get the magnitude, we will simply take the square root of the real part plus the imaginary part squared. The phase will simply be the arctangent of the imaginary part over the real part.

$$Magnitude = \sqrt{((-9.6)^2 + (-4.8)^2)} = 10.7331$$

$$Phase = \arctan\left(\frac{-4.8}{-9.6}\right) = 26.5651 \; degrees$$

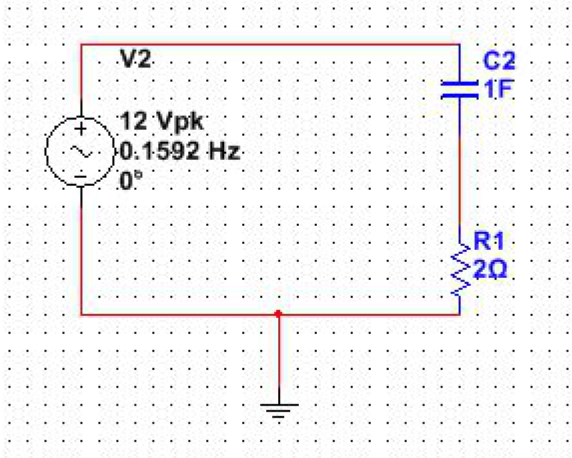
But, since the polarity of the source is backwards, we must subtract 180 degrees from the phase to get it to properly reflect this polarity and the quadrant of the two terms. Thus, the final answer is
$V_0 = 10.7331 \angle -153.435$.

We can corroborate this answer with both Multisim and Matlab. To do the calculations in Matlab, consult the figure below.



```
Command Window
>> % Finding V_Out:
>> V_Source = 12; % Voltage in branch
>> V_Out = V_Source * (2 / ( 2 - j)); % Voltage Divider Equation
>> V_Out = V_Out * -1 % Reverse Polarity

V_Out =

  -9.6000 - 4.8000i

>> [Phase Magnitude] = cart2pol(-9.6,-4.8); % Convert V_out to Polar
>> Phase = radtodeg(Phase); % Convert Phase to degrees
>> [Phase,Magnitude]

ans =

 -153.4349   10.7331

>>
```

The equivalent circuit in Multisim can be seen here:



We can choose a 1 Farad capacitor to replace the -j1 capacitor in the problem because we have set the frequency to 1 / 2pi. This makes the capacitor's impedance $\frac{1}{(\frac{2*pi}{2*pi}*C*j)}$. Meaning a 1 farad capacitor would give us our required impedance of -j1. Doing a single frequency analysis of this in Multisim will give us the values we have already calculated:

Single Frequency AC Analysis @ 0.159155 Hz

	AC Frequency Analysis	Frequency (Hz)	Real	Imaginary
1	V(1)	0.159155	-9.60000	-4.80000

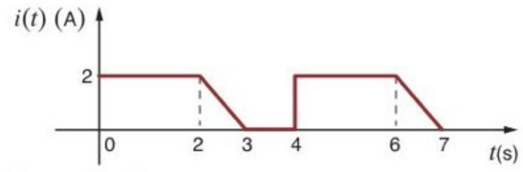
Single Frequency AC Analysis @ 0.159155 Hz

	AC Frequency Analysis	Frequency (Hz)	Magnitude	Phase (deg)
1	V(1)	0.159155	10.73313	-153.43496

Multisim completely verifies our values, showing a non-phasor representation as -9.6 – 4.8j, and a phasor representation as having a magnitude of 10.7331 and a phase of -153.435 degrees.

Problem #3a

3.(a) Calculate the RMS value of the waveform shown. Assume the signal is periodic with a period of 4 secs.



We can look at this signal and break it down in to time periods to integrate in order to find the RMS value. The first time period will be between 0 and 2s, where the value of the current is 2A. The second time period is from 2 to 3s, where the current value is 6 – 2t (equation of the line). The final time period is 3s to 4s, which, because it is 0, is unimportant to calculations. Now, we can apply our RMS equation:

$$RMS=\sqrt{\frac{1}{T}*(\int_{0}^{2} 4\,dt+\int_{2}^{3}(6-2t)^2\,dt)}$$

$$RMS=\sqrt{2+\frac{1}{4}(\frac{4(3^3)}{3}-12(3^2)+36*3)-(\frac{4(2^2)}{3}-12(2^2)+36*2)}$$

By simplifying, we get RMS = 1.5275A.

Of course, we can also do the RMS calculations with Matlab and Matlab will do all the heavy lifting for us, no need to confuse ourselves with integral calculations and algebra.

```
>> %Calculate RMS Value of signal with period of 4s
>> Period = 4;
>> Interval0to2 = 2; %Straight Line
>> syms t
>> Interval2to3 = -2*t + 6; % Line with slope of -2 and y int of 6
>> % Calculate RMS by integrating square, dividing by period, square rooting
>> I_RMS = sqrt(1/Period * ( (int((Interval0to2^2),t,0,2)) + int((Interval2to3^2),t,2,3)))

I_RMS =

(3^(1/2)*7^(1/2))/3

>> Answer = (3^(1/2)*7^(1/2))/3

Answer =

    1.5275

>> double(I_RMS)

ans =

    1.5275

>>
```

Problem #3b

(b) Assume this signal is a current flowing through a 2Ω resistor. How much power is dissipated? How does this compare to a sinewave with a 2V peak amplitude? Explain.

This is a simple power calculation problem, without any manipulation required. The power dissipated by the signal in 3a can is:

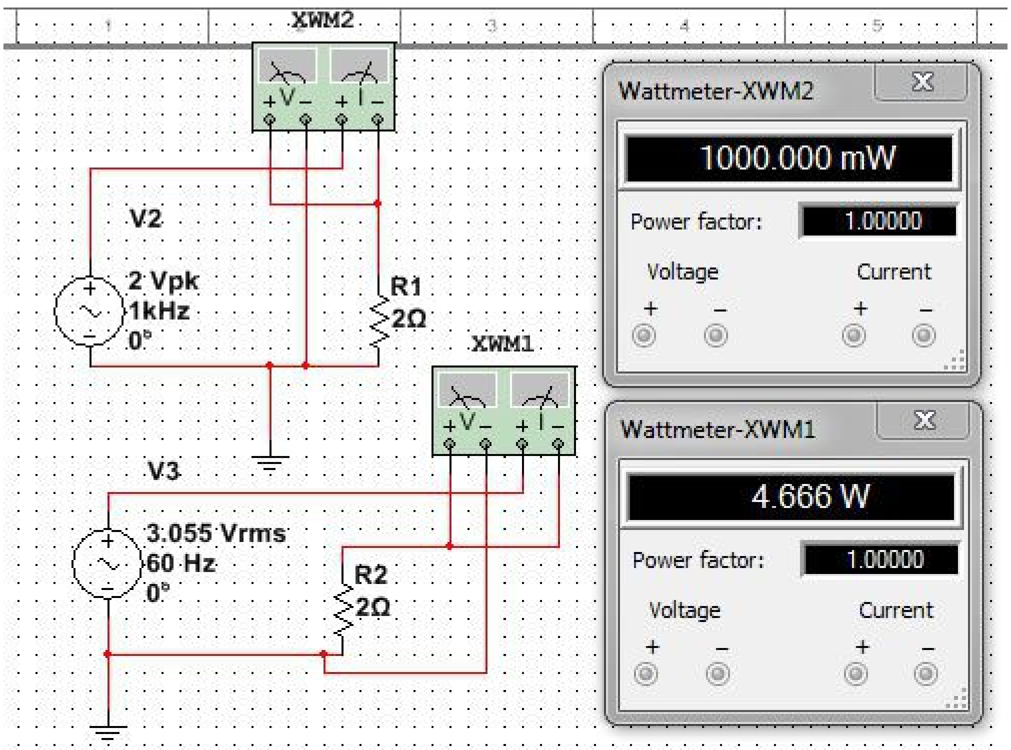
$$P_{3a} = I_{RMS}^2 * R = 1.5275^2 * 2 = \frac{14}{3} = 4.667\, Watts$$

The power of the 2V Peak sine wave needs to be expressed as an RMS value, we can do this by multiplying by .707. This gives us a V_{RMS} of 1.414 Volts. Thus its power is:

$$P_{\sin} = \frac{V_{RMS}^2}{R} = \frac{1.414^2}{2} = .9997\, Watts$$

The first signal dissipates much more power, because it has the highest DC value (which RMS measures).

If we convert the signal from 3a to Vrms from Irms using Ohm's Law, and plug it in to Multisim, we can check our power results rather easily as well.



From this Multisim, we can clearly see that 1 Watt is dissipated in the 2Volt peak signal, whereas 4.66 Watts are shown in the signal from 3a.