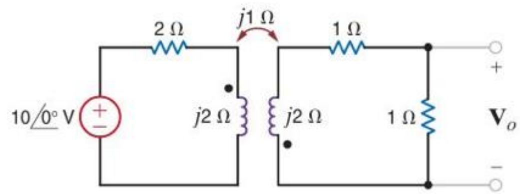
Jarel Benjamin

Reworked Exam No.3

1. Find V_o in phaser form



The first step to obtaining the answer is by identifying the method that will be used in solving this circuit. In this problem the method of choice is to use mesh analysis for the two loops. Upon analyzing the circuit one notices that we have two unknown values, I_1 and I_2, thus we need two loop equations. Another important detail to pay attention to is the mutual inductance between the two coupled inductors and the orientation of the two inductors. As seen, the second inductor is oriented in the same manner as the first inductor thus the mutual inductance in our equation will be positively oriented. The equation for the first loop is:

$$-10V + 2(I_1) + j2(I_1) + j1(I_2) = 0$$

Rewriting the equation gives us:

$$[2 + j2]I_1 + [j1]I_2 = 10V$$

Solving for the second equation gives:

$$1(I_2) + 1(I_2) + j2(I_2) + j1(I_1) = 0$$

Rewriting the equation gives:

$$[2 + j2]I_2 + [j1]I_1 = 0$$

After writing our equations for our two unknowns the next step is to simply solve for our unknown variables. Using MATLAB one can simple enter in these equations as vectors and solve for both unknowns at the same time. Using MATLAB gives us:

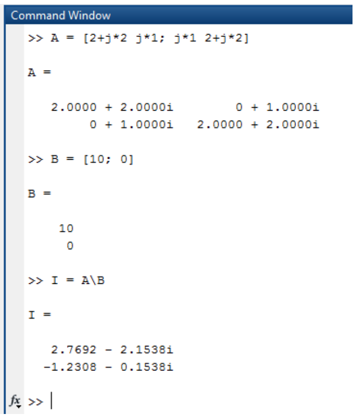


Figure 1. Solving for currents in MATLAB

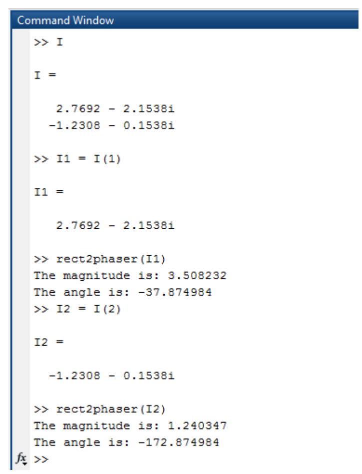


Figure 2. Converting currents to phaser form

Using MATLAB gives us I_1 and I_2 in phaser form as:

$$I_1 = 3.508232\langle -37.874984^o A$$

$$I_2 = 1.24034\langle -172.874984^o A$$

Solving for V_o is a just simply multiplying the resistance by the current which gives:

$$V_o = 1\Omega * I_2 = 1.24034\langle -172.874984^o V$$

Simulating the circuit in Multisim:

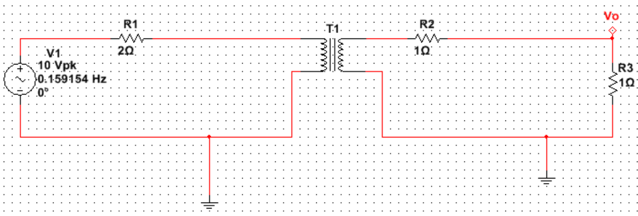


Figure 3. Multisim circuit

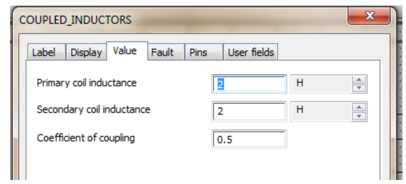


Figure 4. Mutual Inductance parameters

As the Mutual inductance between two inductors is described by the equation:

$$M = K\sqrt{L_1 + L_2}$$

where K is the coefficient of coupling, one can achieve a Mutual inductance of 1H and using a voltage source with an arbitrary frequency the desired impedance for both inductors and the Mutual inductance are achieved. Using a single frequency analysis the results for voltage output is:

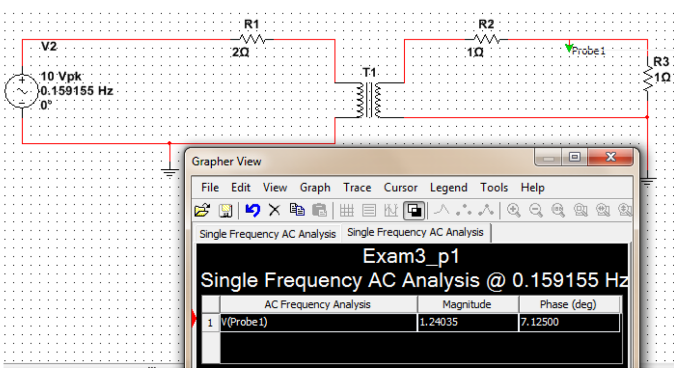
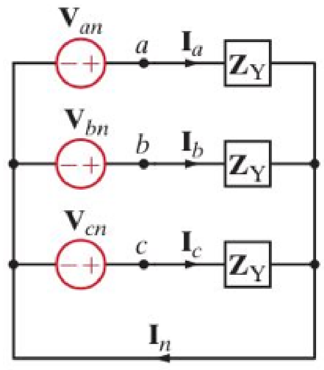


Figure 5. Multisim results

The only difference between the Multisim answer and our answer is that the phase is positive due to the fact that Multisim allows one order for coupled inductors.

2. Determine the load impedance



First thing to do here is to understand that the power loss in all three lines for the circuit is 400W and not 400W in each line. Seeing as how we have three lines, the next step to do is solve for the currents in each line knowing that each current will be the same:

$$P = 3I^2R$$

Solving for I:

$$I_M = \sqrt{\frac{P}{3R}}$$

$$I_M = 8.1649A$$

Knowing that the phase of the voltage and the power factor of the load, one can find the angle for the current:

$$\theta_z = cos^{-1}(0.77) = 39.646^o$$

$$\theta_i = \theta_v - \theta_z = -8.086^o$$

$$I_a = I_b = I_c = I_L = 8.1649\langle -8.086^o A$$

Solving for the load impedance:

$$Z_{Load} = \frac{105.28\angle 31.56^{o} V}{8.1649\angle -8.086^{o} A} = 12.89\angle 39.646^{o} \Omega$$

Using a generated circuit in Multisim to get our values gives us this circuit:

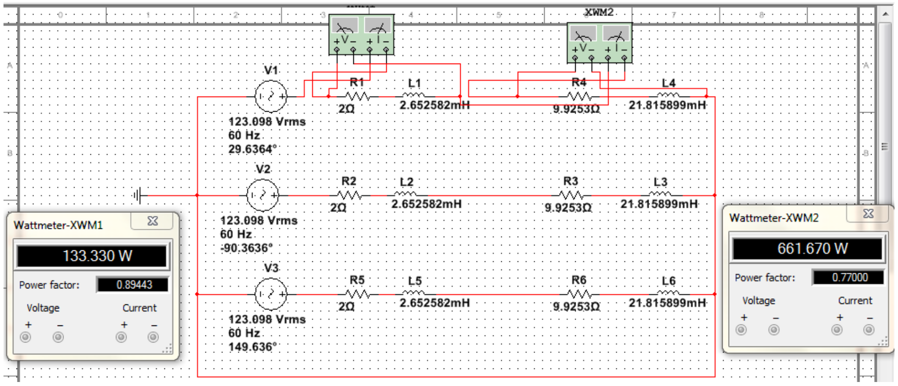


Figure 6. Three Phase simulation

Upon observing the image one notices that the power factor of the load is 0.77. Another important thing to pay attention to is that the impedance of the first line is 133.33W which is a third of the power loss in lines all together. One must note that since this is a three phase power system so the phases for each line is shifted by 120^{o}. Also to be noted is that phase voltage for each line is a result of the voltage across the load plus the voltage loosed in the lines. Solving for the voltage in the lines gives us:

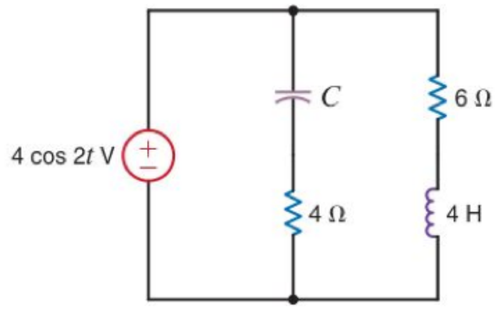
$$V_{Line} = [2 + j1] * I = 18.257271\angle 18.479051^{o} V$$

Combining this with the voltage across the load gives:

$$V_s = 105.28\angle 31.56^{o} V + 18.257271\angle 18.479051^{o} V = 123.089\angle 29.6364^{o} V$$

It is to be noted that this phase voltage if just for one line and not for all three line voltages.

3. Determine the value of C that allows the circuit to be at resonance



For a circuit to be at resonance the impedance in the circuit has to be purely resistive. Thus we are looking for a value of the capacitor that gives the circuit a real value for impedance when at the frequency of the source. To solve this problem the first recommended step is to convert the components into their Fourier transformations to solve for C.

Converting the left branch:

$$4-\frac{j}{2C}=\frac{8C-j}{2C}$$

Converting the right branch:

$$6+j8$$

Turning the two into admittances is better due to the fact that admittances combine in parallel:

$$\frac{2C}{8C-j}+\frac{1}{6+j8}$$

Multiplying the left branch by its complex conjugate gives:

$$\frac{16C^2+j2C}{64C^2+1}$$

Simplifying the right branch gives:

$$0.06 - j0.08$$

As previously described, when in resonance the impedance or admittance of a circuit is purely resistive or real, thus we can set write an equation using our admittance values:

$$\frac{16C^2 + j2C}{64C^2 + 1} + 0.06 - j0.08 = R$$

Upon analyzing our equation, one will notice that to have a real value the two complex portions have to cancel each other out, thus a new equation can be formed:

$$\frac{j2C}{64C^2 + 1} - j0.08 = 0$$

Writing in quadratic form:

$$j5.12C^2 - j2C + j0.008 = 0$$

Using MATLAB to solve for C we get:

$$C \ = \ 0.3453F \ and \ C \ = \ 0.0452F$$

Solving for the frequency of the circuit:

$$f = \frac{\omega}{2\pi} = 0.3183Hz$$

Last all we have to do is measure the two values of C at the frequency of the source

Using Multisim for our two values gives:

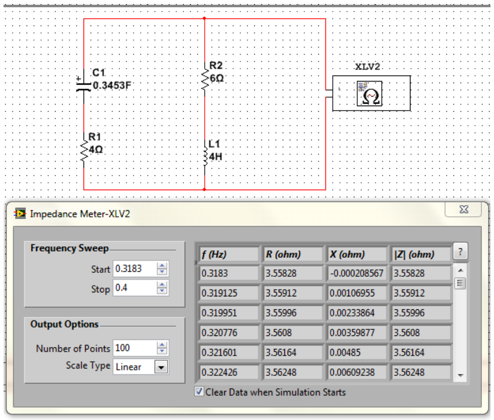


Figure 7. Results for first C value

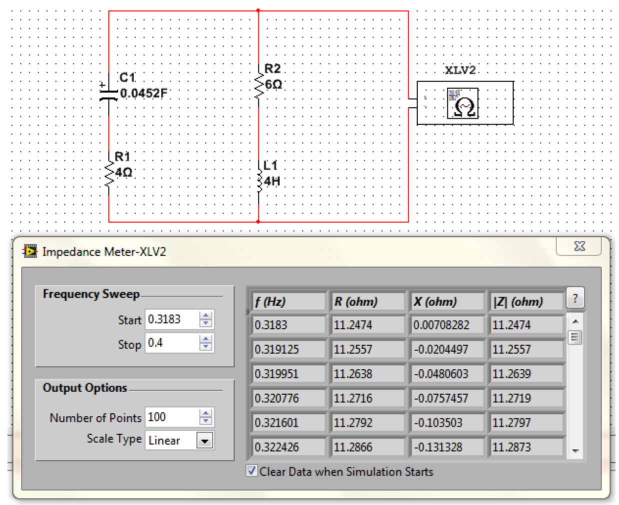


Figure 8. Results for second C value

As we can see the value for the imaginary portion of the impedance is relatively small and when looking at the magnitude of the impedance the value is almost identical to the resistive value of the circuit for both values for the capacitor.