Washington State University School of Electrical Engineering and Computer Science

EE 352 Electrical Engineering Laboratory

Lab # 2

State Variable Modeling and Mutual Inductance

Name: Sarah Rock

Partner: Isobel Baetz

Due Date: 02/08/21

Lab Overview

For the following experiments, state variable modeling was explored in first modeling a mutual inductance circuit then in an underdamped RLC circuit. State variable modeling is used to simplify the analysis and the circuit of initially complex systems with multiple inputs and outputs (MIMO). Using LTSPICE, the first experiment focused on using a simplified model for the mutual inductance (M). This mutual inductance was then compared with the average of the two experimental measurements which was then used to find an experimental coupling coefficient (k). The plots generated from LTSPICE was then compared to simulated graphs produced in MATLAB. The second experiment focused on modeling an underdamped RLC circuit with state variables in LTSPICE, which was also compared to simulated graphs in MATLAB.

Experiment #1 Mutually Coupled Transformer Circuit

1.1 Purpose

The purpose of experiment one was to estimate the mutual inductance (M) using phasor domain analysis for a sinusoidal input then for a triangular pulse input both using the time domain. This was experimentally done in LTSPICE with as simplified mutual inductance model circuit.

1.2 Theoretical Background

Mutual inductance occurs when two inductors are placed in close proximity and share a magnetic flux that causes a mutual exchange between inductors where each produce a flux by the generated current which induces a voltage across the other inductor. Variables such as permeance of the medium, the flux linkage, the number of turns of each inductor, the self-inductances and the distance between the two inductors are all functions of mutual interaction. The voltage induced from the mutual interaction is caused by the rate of change of the current from the other inductor multiplied by the mutual inductance (M). The voltage cause by the self-inductance is expressed as:

$$v_1 = L_1 \frac{di_1}{dt} + M \frac{di_2}{dt} \qquad (1a)$$

$$v_2 = M \frac{di_1}{dt} + L_2 \frac{di_2}{dt}. \qquad (1b)$$

A way to figure out how to estimate mutual inductance experimentally is to connect two inductors in a transformer as shown in Fig.1. When a time varying signal is applied, known as u(t) in this experiment, and induced voltage is observed at the output. The secondary current as well as its derivative is zero when Ro indicates an open circuit. To measure i_1, R_1 is used and the effect of R_{L1} is ignored since it is assumed to be much less than R_1.

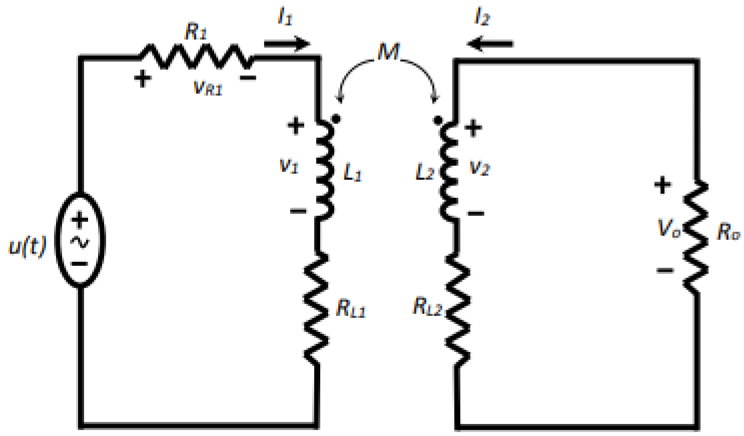


Figure 1. Mutual inductance in a transformer

Equations (2a) and (2b) are equations derived from equations (1a) and (1b) in that they are the product when a sinusoidal signal is applied to the circuit in Fig.1. The variables of (2a) and (2b) are V_1, V_2, I_1, I_2 which are all phasors. For the sinusoidal signal, it allows the use of a phasor analysis and it is one of the two signals that can be easily applied to find an analytical solution.

$$V_1 = j\omega L_1 I_1 + j\omega M I_2 \qquad (2a)$$

$$V_2 = j\omega M I_1 + j\omega L_2 I_2, \qquad (2b)$$

Equation (3) represents how an analytical solution can be achieved for the second signal that can be easily applied, which is a triangular pulse. As shown in equation (3), the pulse is estimated using the linear approximation $\frac{\Delta I}{\Delta t}$ of the derivatives given a certain frequency f.

$$\frac{di_1}{dt} = 2f\Delta I_1 \quad \text{and} \quad \frac{di_2}{dt} = 2f\Delta I_2. \ (3)$$

When calculating the theoretical values for the equations above, the given parameters in the prelab were self-inductance as $L_1 = L_2 = 200$ mH, $R_0 = \infty$ and $R_{L1} = R_{L2} = 50\ \Omega$, u(t) as a sinusoidal signal with $f = 400\ Hz, \omega = 2513.3\ rad/sec$, and $R_1 = 10\ k\Omega$.The expressions for u(t) and $v_2(t)$ to find an expression that could be used to measure M, were as follows:

$$\mathrm{u(t)} = 5\sin(800\pi t)\ (4)$$

$$\mathrm{v_2(t)} = \mathrm{V}_m \sin(800\pi t + \theta)(5)$$

The calculations to determine the results are shown in Appendix E. $\mathrm{v}_2 = 36.39mV$, $I_1 = .4969 < -288°$. From these results and equation (6) below, M was determined to be 47.38 mH, all for part a of the prelab.

$$M = \frac{|V_2|}{j\omega I_1} \qquad (6)$$

The results for part b were determined using a triangular pulse of 10V peak-to-peak at $f = 400\ Hz$ and $v_{R1}(t) \approx u(t)$. The determined expression for $\frac{\mathrm{di}_1}{dt} = \frac{1}{R_1} = \frac{\mathrm{du(t)}}{dt}$ where it would be .8 A/s between time of 0 and 1.25 ms and -.8 A/s between time 1.25 and 2.5 ms. The determined expression for M is as follows:

$$M = \frac{|V_2|}{\frac{\mathrm{di}_1}{dt}} \qquad (7)$$

Where if the measured $\mathrm{v}_2 = 36.976\ mV$, then M= 46.22 mH. The theoretical value of M was found in equation (8) where k=.25, L1= .18 H, L2= .19 H.

$$M = k\sqrt{L_1 L_2} = 46\ mH \qquad (8)$$

1.3 Procedure

1. Values for the following were randomly picked L1 and L2 to be ranged from 0.17 H- 0.2 H, RL1 and RL2 to be ranged from 140 Ω to 160 Ω, and a value for the mutual coupling coefficient (k) to be ranged from 0.2 to 0.5, where $k = \frac{M}{\sqrt{L_1 L_2}}$.
2. LTSPICE was used to construct the transformer circuit as shown in Figure 1, where Ro = 1 MΩ. Then the dots on the inductors were rotated so that both were oriented on the top of the inductor.
3. Then the mutual coupling was modeled by using the .op icon where the command K1 L1 L2 k was entered with the picked value k which was .25.
4. Once the circuit was set up a triangular pulse was applied with 10 V peak-to-peak centered at zero at 400Hz, a Vpulse signal with Vin=-5V, the Von= 5V, time rise and time fall at 1.25 ms, the pulse width at 0 and the period at 2.5 ms.
5. Then a transient analysis was applied to the circuit with 25ms the stop time and 1 μs for the max step size.
6. A simulation was run to plot I1 and Vo. Then the plot was sketched in the notebook.
7. Next the mutual inductance was estimated based off equation (1b) and the solution to question (b) of the pre-lab.
8. A 10 V peak-peak (5 V zero to peak) sine wave at 400 Hz was then applied to the circuit. Also, a transient analysis to show around 10 cycles was applied to simulate a plot for Vin, the voltage VR1, Vo and I1. These plots were then sketched.
9. The mutual inductance M was then estimated using the sinusoidal wave measurements and the solution to question (a) of the pre-lab.
10. The average for M of the two estimated cases was determined from step 7 and step 9 and used to estimate the coupling coefficient k from M, L1 and L2.
11. Using the sinusoidal measurements, L2 was flipped so that the dot on L2 was oriented at the bottom and the dot of L1 was at the top. This was done to show dot convention changes relative to the direction of the coil windings.

Post-Lab Procedure

1. For the MATLAB m-file provided in Appendix B, the variables R1, Ro, L1, L2, M, RL1 and RL2 was set to match the values picked in the lab.
2. Then checked the dot convention used in the state space model to ensure it was consistent with the dot convention of the circuit constructed in the lab.
3. Once all was correct, the MATLAB m-file was simulated to produce the responses for both the sinusoidal and triangular inputs.
4. Lastly, the measure and MATLAB simulated responses were compared.

1.4 Results & Analysis

When comparing the experimental and theoretical results, they ended up being within close porximity of eachother. Something that must be taken in consideration when comparing the chosen values from the values in the prelab is that the range to choose from changed right when the lab started to match the prelab values. Considering the degree of influence to achieve the theoretical answers for the experimental, it proved to have little influence.

In order to satisfy the directed range values for the variables stated in step 1 of the procedure, $L_1 = .18\ H, L_2 = .19\ H,\ R_{L1} = 160\Omega, \mathrm{R}_{L2} = 170\ \Omega, \mathrm{R}_0 = 1\ \mathrm{M}\Omega$ and $\mathrm{k} = .25$. Fig.2, as shown below, is the result of applying these values along with the transient anaylsis and V pulse with values directed in step 4 of the procedure.

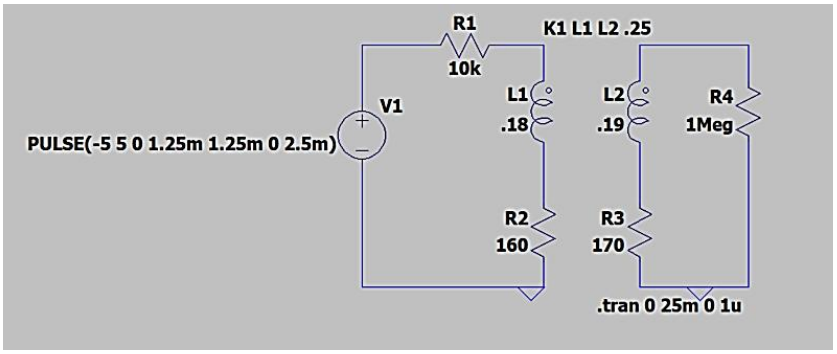


Figure 2: Circuit for Triangular Pulse

When the circuit in Fig. 2 was simulated using LTSPICE, Fig.3 shows the resulting behavior of I1, of R1, and Fig 4 shows the resulting behavior of Vo (voltage output).

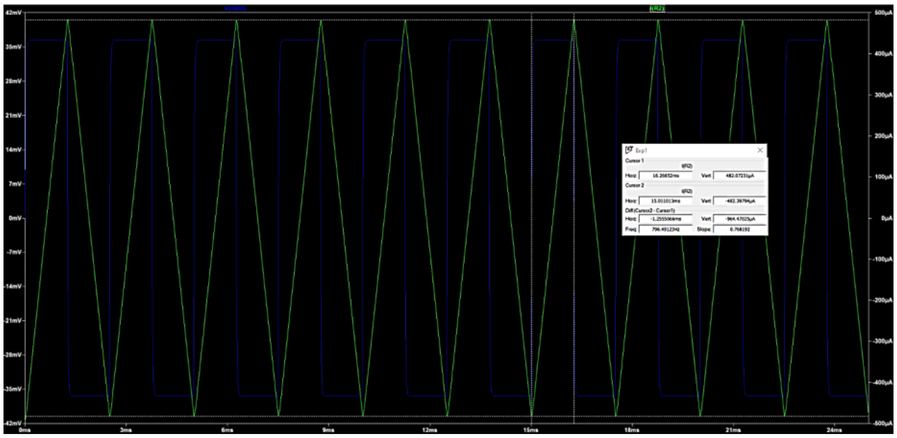


Figure 3: Plot showing I1 for Triangular Pulse

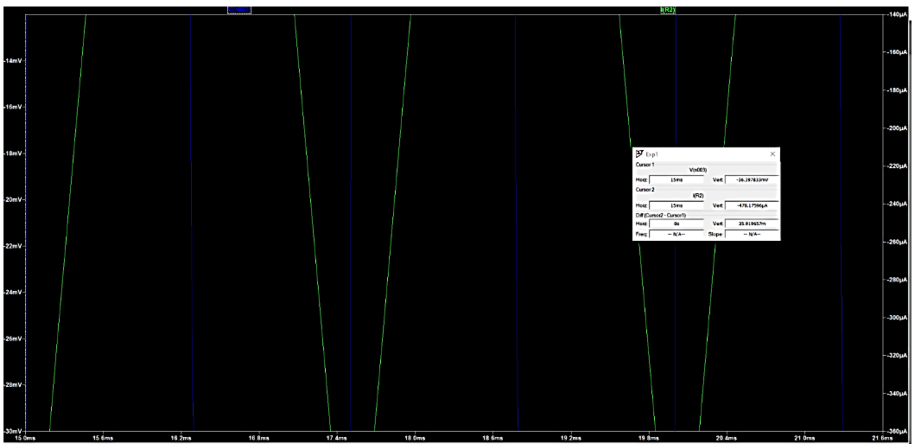


Figure 4: Plot showing Vo

From the resulting behaviors of I1 and Vo, the slope of I1 was determined to be $\frac{964.7uA}{1.25ms} = .772\frac{A}{s}$ and Vo which is equivalent to the absolute value of V2 is 36.4 mV. So, M (mutual inductance) is then determined to be .04716 H. Full calculations are shown in Appendix E.

After completing the post lab, the plots of Fig.5 show the product of the MATLAB m-file simulated from Appendix B. From the human eye, the responses from the experiment and from MATLAB are basically identical. The only difference that is humanly visible is the chosen axis max and min values to depict the plots.

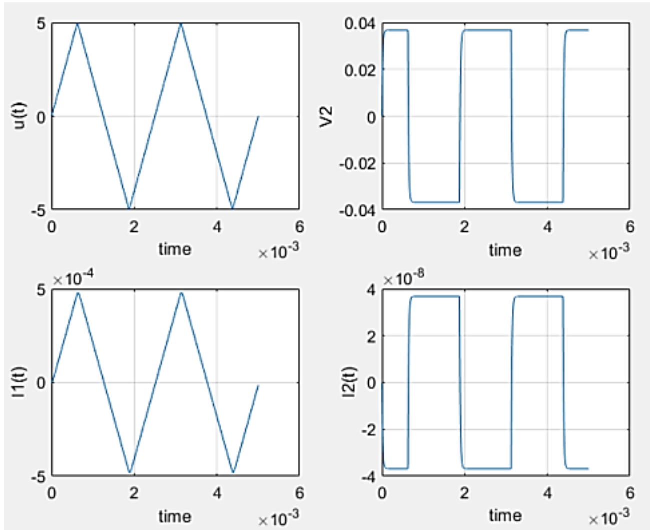


Figure 5: MATLAB graph of Triangular Pulse from code in Appendix B.

The second half of this experiment was to analyze the circuit with a sine wave signal with the parameters stated in step 8 of the procedure, the resulting circuit is shown in Fig.6.

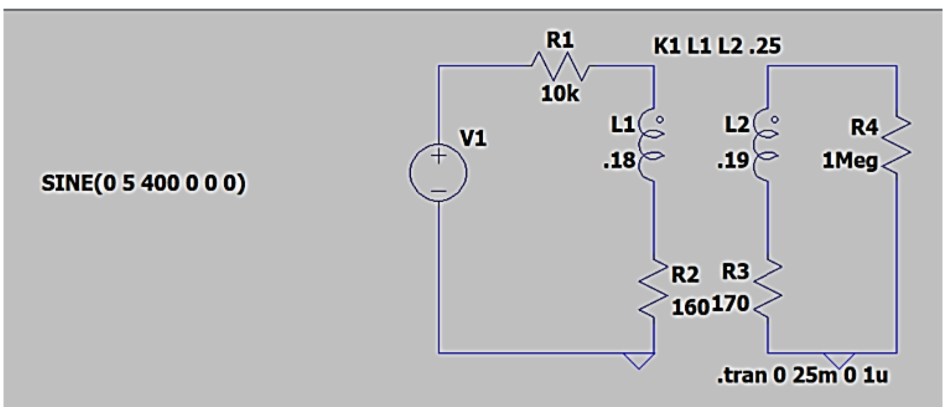


Figure 6: Circuit of figure 1 with inputed values with a sinosoidal input

When Fig. 6 was simulated using LTSPICE, the plot in Fig. 7 shows the plotted responses of Vin, the voltage VR1, Vo and I1.

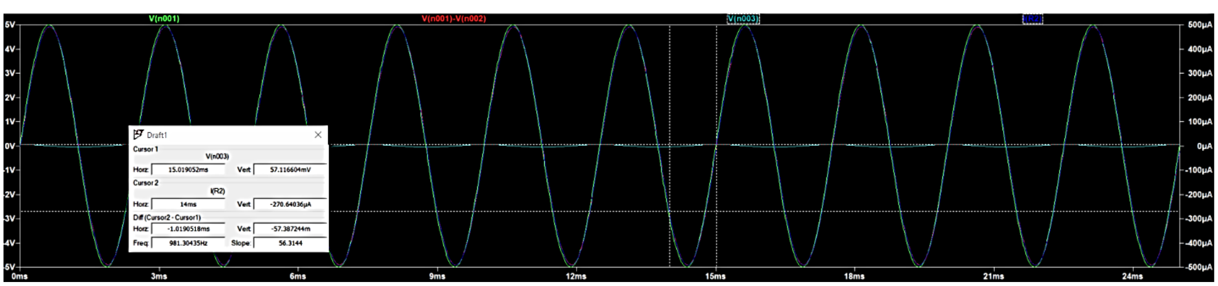


Figure 7: Sinusoidal plot showing Vo,Vr1,I1,Vin

Using the plot in Fig.7, the Vo which is still the absolute value of V2 is 36.39 mV. The absolute value of I1 was determined to be $490.12\ \frac{uA}{s}$ and $\omega = 2\pi f\ = 2\pi 400 = 800\pi rad/s$. As shown in Appendix E, these values were then used to calculate M as .0464 H.

This meant that the average of the two M's became .04677 H which is only a 1.16% error from the theoretical value of .04622 H determined in the prelab. Then k was determined to be .253, using the experimental average of M. This meaning the experimental k compared to the theoretical k only has a 1.2% error. These errors are due to a measuring error when navigating the max values for all the measured signals.

Part 2 of the MATLAB m-file simulated in the post lab produced the same plots as expressed in Fig.5 but as sine waves as expressed in Fig. 8. As stated in comparing the pulse signals between LTSPICE and MATLAB, the sinusoidal signal plots are also the same except the chosen min and max values of the axes.

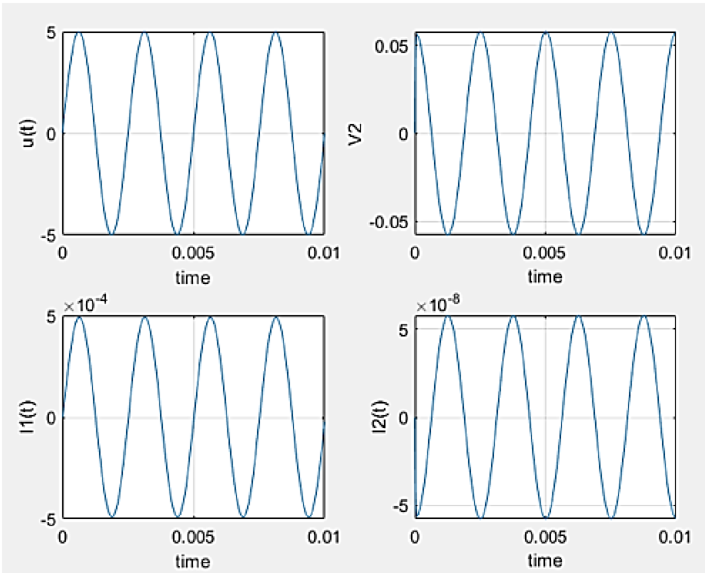


Figure 8: MATLAB graph of Sinusoidal Pulse from code in Appendix B

1.5 Conclusion

To conclude this experiment, the experimental mutual inductance and k value was determined to be less than a 2% error. This error is simply due to a human measuring error. The plots of I1, Vo, VR1 and Vin of both signals proved to be basically identical when comparing between LTSPICE and MATLAB. This saying that using these software programs allows for close to accurate results of an ideal mutual inductance circuit in a transformer.

Experiment #2 State Variable Modeling of RLC circuit

2.1 Purpose

The purpose of this experiment was to determine $Vc(\infty) and\ I_L(\infty)$ where time (t) is approaching infinity using a steady state response. From these determined values, the last thing is to find the state trajectories as time approached infinity.

2.2 Theoretical Background

For state variable modeling of an RLC circuit it is important to that if the state variable signals are known, the any voltage and any current of that circuit can be solved. An ideal approach in picking state variables is to use the capacitor voltages and the inductor currents because they have continuous derivatives. To measure the inductor current, R2 as shown in Fig.9 and Fig 11, is used. When R2 is selected to be about 1 Ω, V_{R2} and I_L can be considered about equal. To simulate and measure the state trajectories, the first state variable $x_1(t) = v_c(t)$ and the second state variable is $x_2(t) = i_L(t)$. For the circuits below, the output is then $v_c(t)\ and\ v_{R2}(t)$. The state trajectory is simply a plot comparing $x_1(t)\ vs.\ x_2(t)$ as time (t) approaches infinity. To achieve accurate measurements is it important to have R_L, which is the internal resistance of the inductor be included as shown in Fig.10 and applied in Fig.11. Another thing important to be included is the internal resistance of 50 Ω added to the signal generator. As a summary, the circuit to be simulated using LTSPICE and MATLAB will be the actual circuit shown in Fig.11, given that Fig.9 is the ideal circuit.

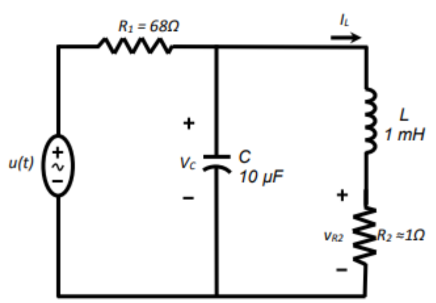


Figure 9: Ideal RLC circuit

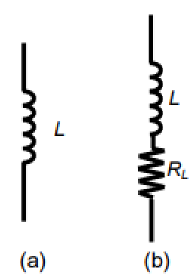


Figure 10: (a) Ideal inductor. (b) Practical inductor with series resistance.

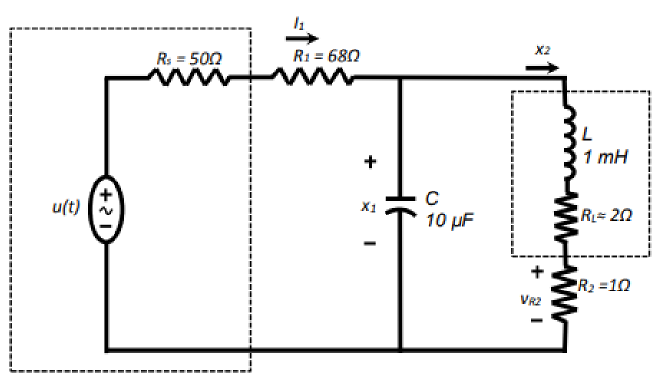


Figure 11: Actual RLC circuit

The state variable equations were determined to be (C6) and (C9) found in Appendix C. $Vc(\infty) and\ I_L(\infty)$ were found to be .2479V and 82.65 mA as shown in equation (9) and (10) below.

$$Vc(\infty) = u(t)\frac{R_L+R_2}{R_{eq}} = 10\left(\frac{3}{118+3}\right) = .2479\ V \qquad (9)$$

$$I_L(\infty) = \frac{u(t)}{R_{eq}} = \frac{10}{118+3} = 82.65\ mA \qquad (10)$$

2.3 Procedure

1. Using LTSPICE, the circuit shown in Fig. 11 was built where Rs was the internal resistance of the function generator, R_L is the internal resistance of the inductor and R_2 is a sense resistor used to determine the current through the inductor. The input voltage source is a Vpulse with the following parameters was applied, V_{in}= 0V, Von =10V, Time rise = 1n, Time fall = 1n, Ton = 5ms, and Period = 10 ms.
2. A transient analysis for 20 ms and set the maximum step size of 100 ns was then applied and the capacitor voltage $x_1(t)$ and the sense resistor voltage $V_{R2}(t)$, and current $x_2(t)$ where all plotted. Then the steady state response for both signals were compared to the prelab lab values.
3. Lastly the measured state trajectory using the $x_1(t)$ and $x_2(t)$ waveforms were plotted.

Post-Lab Procedure

1. Using the state space model of the circuit in Figure 11 calculated in Appendix C, a MATLAB m-file was written to set up the state-space matrices A, B, C, and D using the circuit variables provided in the lab.
2. Then the MATLAB m-file was used to simulate the step response of the circuit of Figure 11 using the step command:
   ```
   sys = ss(A,B,C,D);
   [Y,T,X]=step(sys);
   ```
3. Next the simulated system states $x_1(t)$ and $x_2(t)$ were simulated as a function of time, using the T vector as the time variable.
4. Then the simulated state trajectory, $x_1(t)$ vs $x_2(t)$, was plotted.
5. Lastly, the state variable plots generated from the lab were compared to the plots generated in MATLAB.

2.4 Results & Analysis

When comparing the results between LTSPICE and MATLAB, the graphs and calculations produced the same values and pattern of behavior. This being due to fact that both software programs are designed to simulate ideal outcomes.

To simulate the required plots, the circuit below was generated based on the circuit shown in Fig. 11 and the values indicated in step 1 and 2 of the procedure, the product of combining them is shown in Fig. 12.

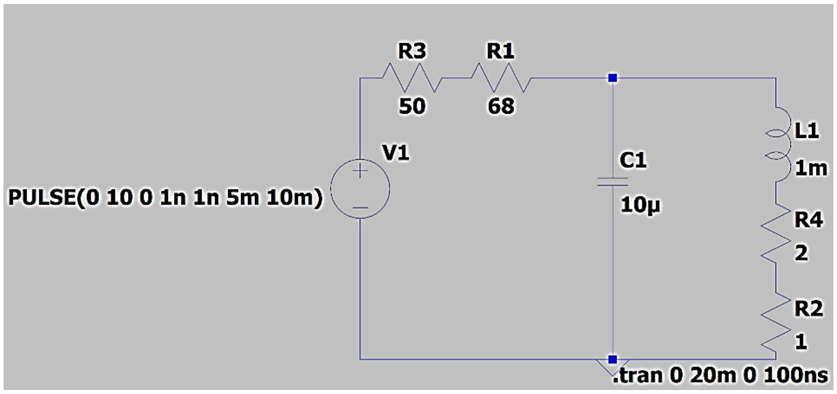


Figure 12: RLC Circuit simulated with Required Values.

Fig.13 (a-c) show the different signals measured in the circuit shown in Fig.12. These signals being I_1 which was probed at R_1, V_{R2} (used to estimate I_L since they are similar as stated in the theoretical background and is also the second state variable), and V_C (the first state variable).

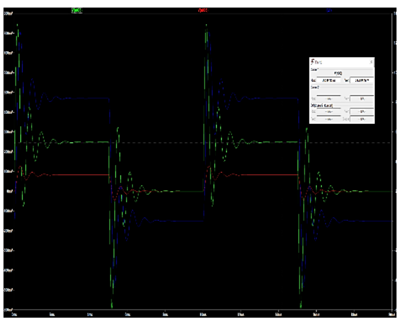


Figure 13(a): V_C values

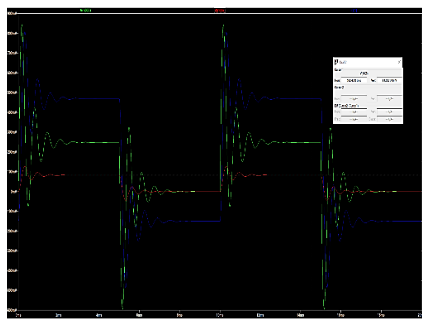


Figure 13(b): V_{R2} values

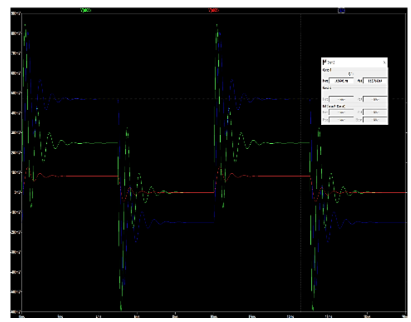


Figure 13(c): I_1 values

From Fig.13, the final value steady state response were found to be 82.65 mA for I_1, 247.89 mV for V_C and 82.64 mV for V_{R2}. When comparing these values to the theoretical values, there is a 0% error.

When comparing the plots above to the plots generated in MATLAB for the post lab, the plots seemed to produce the same values as in LTSPICE. The plots of MATLAB for experiment 2 is shown below in Fig. 14.

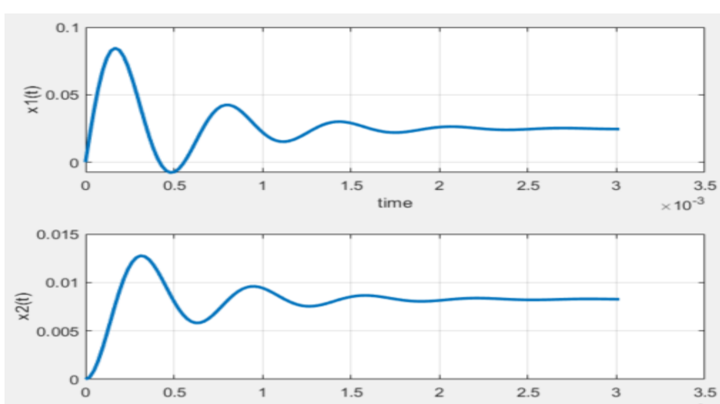


Figure 14: $x_1(t)$ and $x_2(t)$ vs. time

Below in Fig.15(a-b) represents the plotted result when plotting $x_1(t)$ vs $x_2(t)$ as time (t) approaches infinity. The swirls into the points represent degree of oscillation approaching steady state. As shown in both figures, the points where steady state is approached, the values are the same as the theoretical values calculated in the prelab and found in Fig. 13 (a-c).

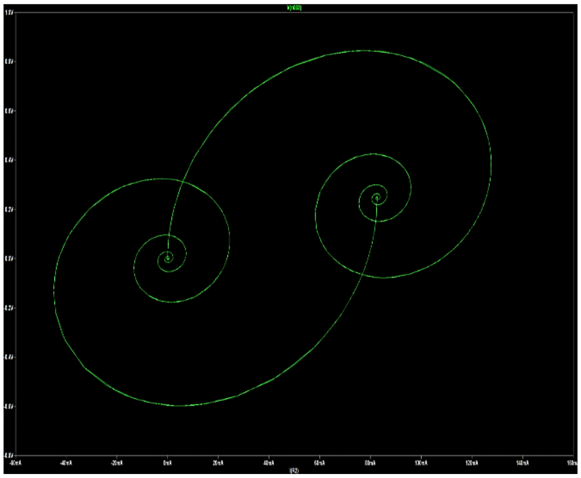
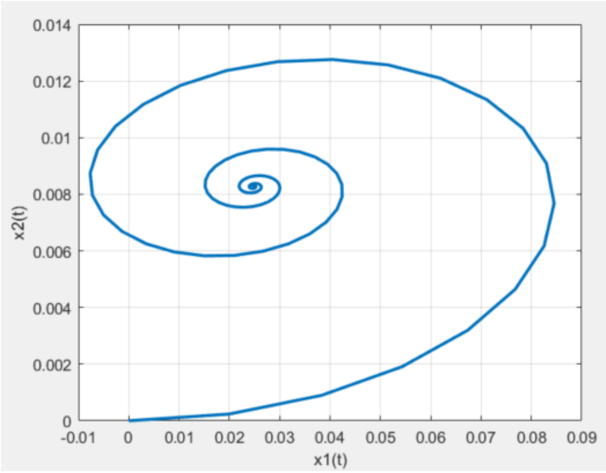


Figure 15(a): $x_1(t)$ vs $x_2(t)$ as time (t) approaches infinity in LTSPICE.



$x_1(t)$ vs $x_2(t)$ as time (t) approaches infinity in MATLAB.

2.5 Conclusion

To conclude experiment 2, when modeling state variables for an RLC circuit, the results are all the same in comparing with 0% error. The differences in axis min and max between the plots reflects parameters given for simulation. LTSPICE is better at showing more of the pattern happening in the circuit, MATLAB breaks down the state variable analysis and the theoretical values serve to validate the accuracy of the two systems.

Appendix

Appendix A

Obtaining the State Variable Model for the Transformer Circuit.

Our objective is to obtain the state variables model for the transformer circuit shown in Figure1.

Here we have two inductors L1 and L2, so let x1(t) = i1(t) and x2(t) = i2(t). We will also add the

loading effect of the scope Ro, which is 1MΩ for 1x or 10 MΩ for 10x. Applying KVL at the primary loop we get

$$u = (\mathrm{R}_1 + \mathrm{R}_{L1})\mathrm{i}_1 + v_1 \qquad \text{(A1)}$$

By substituting eqn. (1) in equation (A1) we get

$$\mathrm{u} = (\mathrm{R}_1 + \mathrm{R}_{L1})\mathrm{i}_1 + \mathrm{L}_1 \frac{\mathrm{di}_1}{dt} + \mathrm{M}\frac{\mathrm{di}_2}{dt}. \qquad \text{(A2)}$$

Following the dot convention, we also use KVL around the secondary loop, to get

$$0 = (\mathrm{R}_0 + \mathrm{R}_{L2})\mathrm{i}_2 + \mathrm{L}_2 \frac{\mathrm{di}_2}{dt} + \mathrm{M}\frac{\mathrm{di}_1}{dt}. \qquad \text{(A3)}$$

Equation (A3) can be solved for $\frac{\mathrm{di}_2}{dt}$ as

$$di2\ dt = -\frac{(\mathrm{R}_0 + \mathrm{R}_{L2})\mathrm{i}_2}{\mathrm{L}_2} - \frac{M}{\mathrm{L}_2}\frac{\mathrm{di}_1}{dt}\ . \qquad \text{(A4)}$$

Substituting (A4) into (A2) gives

$$u = (\mathrm{R}_1 + \mathrm{R}_{L1})\mathrm{i}_1 + \mathrm{L}_1 \frac{\mathrm{di}_1}{dt} + M\left[-\frac{(\mathrm{R}_0 + \mathrm{R}_{L2})\mathrm{i}_2}{\mathrm{L}_2} - \frac{M}{\mathrm{L}_2}\frac{\mathrm{di}_1}{dt}\right]. \qquad \text{(A5)}$$

By grouping the $\frac{\mathrm{di}_1}{dt}$ we get

$$\mathrm{u} = (\mathrm{R}_1 + \mathrm{R}_{L1})\mathrm{i}_1 - \frac{\mathrm{M}}{\mathrm{L}_2}(\mathrm{R}_0 + \mathrm{R}_{L2})\mathrm{i}_2 + \frac{\mathrm{di}_1}{dt}\left[\mathrm{L}_1 - \frac{M^2}{\mathrm{L}_2}\right]. \qquad (A6)$$

Solving for $\frac{\mathrm{di}_1}{dt}$ gives the first state equation

$$\frac{\mathrm{di}_1}{dt} = -\left[\frac{\mathrm{L}_2\,(\mathrm{R}_1 + \mathrm{R}_{L1})}{L_1L_2 - M^2}\right]\mathrm{i}_1 + \left[\frac{\mathrm{M}(\mathrm{R}_0 + \mathrm{R}_{L2})}{L_1L_2 - M^2}\right]\mathrm{i}_2 + \left[\frac{L_2}{L_1L_2 - M^2}\right]\mathrm{u}\ . \quad (A7)$$

Substituting for (A7) in (A4) we obtain $\frac{\mathrm{di}_2}{dt}$

$$\frac{\mathrm{di}_2}{dt} = -\frac{(\mathrm{R}_0 + \mathrm{R}_{L2})}{L_2}\mathrm{i}_2 + [[\frac{\mathrm{M}(\mathrm{R}_1 + \mathrm{R}_{L1})}{L_1L_2 - M^2}]\mathrm{i}_1 - \frac{M}{L_2}[\frac{M(\mathrm{R}_0 + \mathrm{R}_{L2})}{L_1L_2 - M^2}]\mathrm{i}_2 - [\frac{M}{L_1L_2 - M^2}]\,\mathrm{u}. \text{(A8)}$$

Reorganizing the terms gives the second state equation

$$\frac{\mathrm{di}_2}{dt} = \left[\frac{\mathrm{M}(\mathrm{R}_1 + \mathrm{R}_{L1})}{L_1L_2 - M^2}\right]\mathrm{i}_1 - \left[\frac{M(\mathrm{R}_0 + \mathrm{R}_{L2})}{L_1L_2 - M^2}\right]\mathrm{i}_2 - \left[\frac{M}{L_1L_2 - M^2}\right]\mathrm{u}. \qquad \text{(A9)}$$

The output y = V_0 is obtained as

$$y = -R_0 i_2. \qquad \text{(A10)}$$

Appendix B

MATLAB m-file to simulate the transformer response.

```
% Step #1 Enter the actual components values;
R1= 10000; Ro=1000000; L1=0.18; L2=0.19; M=0.04677; RL1=160; RL2=170;
f=400;
k=M/sqrt(L1*L2);
% Step #2 obtain A, B, C, D matrices
% These matrices assume that the dots are on same side of L1 and L2
% If dots on opposite sides than replace M with -M in the A and B matrices
Den=(L1*L2-M*M); % this is the common denominator
A=[-L2*(R1+RL1) M*(Ro+RL2); M*(R1+RL1) -L1*(Ro+RL2)]/Den; % A matrix
B=[L2; -M]/Den; % B matrix
C=[0 -Ro]; % C Matrix
D=[0]; % D Matrix
% Step #3 Entering the state space model
SYS = ss(A,B,C,D);
% Step #4 define the 400 Hz triangular pulse for two periods
u1= 0.001:0.001:1; %up
u2=1-(0.001:0.001:1-0.001); %down
u1p=[0 u1 u2 0 -u1 -u2]*5; %u1p is one period
u = [u1p u1p 0]; %u has two periods
l=length(u)-1;
time=2*(0:l)/(l*f); % time period is (2/400) seconds.
% Step #5 simulate the triangular pulse at 400 Hz
[y1,t1,x1]=lsim(SYS,u,time); % y is the output, t is the time, x is the states
subplot(2,2,1)
plot(t1,u);
xlabel('time');ylabel('u(t)');grid on;
subplot(2,2,2)
plot(t1,y1);
xlabel('time');ylabel('V2');grid on;
subplot(2,2,3)
plot(t1,x1(:,1));
xlabel('time');ylabel('I1(t)');grid on;
subplot(2,2,4)
plot(t1,x1(:,2));
xlabel('time');ylabel('I2(t)');grid on;
% Step #6 simulate the sine wave at 400 Hz for four periods
time=(0:0.001:1)/100;
u3=5*sin(800*pi*time);
[y2,t2,x2]=lsim(SYS,u3,time); % y is the output, t is the time, x is the states
figure % display new graph
subplot(2,2,1)
plot(time,u3);
xlabel('time');ylabel('u(t)');grid on;
subplot(2,2,2)
plot(t2,y2);
xlabel('time');ylabel('V2');grid on;
subplot(2,2,3)
plot(t2,x2(:,1));
xlabel('time');ylabel('I1(t)');grid on;
subplot(2,2,4)
plot(t2,x2(:,2));
xlabel('time');ylabel('I2(t)');grid on;
```

Appendix C

Derivation of the state equations for the circuit in Figure 4

Let the states x1 and x2 be defined as:

$$x_1 = v_c(t) \qquad \text{(C1)}$$

$$x_2 = i_L(t) \qquad \text{(C2)}$$

To derive the first state equation, use KVL in the left-hand loop to get

$$u = (R_s + R_1)i_1 + x_1 \qquad \text{(C3)}$$

But

$$i_1(t) = i_C(t) + x_2 = C\dot{x}_1 + x_2 \qquad \text{(C4)}$$

Substituting for $i_1(t)$in terms of x_1 and x_2 in (C3) gives

$$u = (R_s + R_1)[C\dot{x}_1 + x_2] + x_1 \qquad \text{(C5)}$$

Rearranging equation (C5) gives the first state equation:

$$\dot{x}_1 = \frac{-1}{C(R_s+R_1)}x_1 - \frac{1}{C}x_2 + \frac{u}{C(R_s+R_1)} \qquad \text{(C6)}$$

To derive the second state equation, use KVL in the right-hand loop to get

$$v_c(t) = v_L(t) + i_L(t)\,(R_2 + R_L) \qquad \text{(C7)}$$

Expressing (C7) in terms of the state variables gives

$$x_1 = Lx_2 \dot{+} x_2(R_2 + R_L) \qquad \text{(C8)}$$

Rearranging (C8), we get the second state equation:

$$\dot{x}_2 = \frac{1}{L}x_1 - \frac{(R_2+R_L)}{L}x_2 \qquad \text{(C9)}$$

Rearranging equations (C6) & (C9) in matrix form (using A, B, C and D matrices) gives:

$$\begin{bmatrix}\dot{x}_1(t)\\ \dot{x}_2(t)\end{bmatrix} = \begin{bmatrix}-\frac{1}{C(R_s+R_1)} & -\frac{1}{C}\\ \frac{1}{L} & -\frac{(R_2+R_L)}{L}\end{bmatrix}\begin{bmatrix}x_1(t)\\ x_2(t)\end{bmatrix} + \begin{bmatrix}\frac{1}{C(R_s+R_1)}\\ 0\end{bmatrix}u$$

$$\begin{bmatrix}y_1\\ y_2\end{bmatrix} = \begin{bmatrix}1 & 0\\ 0 & 1\end{bmatrix}\begin{bmatrix}x_1(t)\\ x_2(t)\end{bmatrix} + \begin{bmatrix}0\\ 0\end{bmatrix}u.$$

Appendix D

MATLAB m-file to state variable modeling of RLC circuit

```
Rs=50; %Ohms
R1=68; %Ohms
R2=1; %Ohms
RL=2; %Ohms
L=.001; %Henry's
C=.00001; %Farads
%Step 1 Obtain A B C and D
A=[-(1/(C*(Rs+R1))) -1/C; 1/L -((R2+RL)/L)]
B=[(1/(C*(Rs+R1))); 0]
C=[1 0; 0 1]
D=[0; 0];
%Step 2 Simulate the step response
SYS = ss(A,B,C,D);
[Y,T,X]=step(SYS);
figure % display new graph
subplot(2,1,1)
plot(T,X(:,1),'LineWidth',2);
xlabel('time');ylabel('x1(t)');grid on;
subplot(2,1,2)
plot(T,X(:,2),'LineWidth',2);
xlabel('time');ylabel('x2(t)');grid on;
figure;
plot(X(:,1),X(:,2),'LineWidth',2);
xlabel('x1(t)');ylabel('x2(t)');grid on;
```

Appendix E

The following are calculations required for the prelab and lab of experiment 1. Here the objective is to figure out the theoretical values for V_2, I_1 and M. Then to find the experimental values of V_2, I_1, M, and k.

Experiment 1 Prelab Calculations:

To investigate all parameter first the inductance impedance is solved as follows:

$$Z_{L1} = j\omega L = j * 2513.3 * .2 = 502.67j\Omega \quad \text{(E1)}$$

Thus, to find the polar form of I_1:

$$5 < 0° = I_1 10k + I_1 502.67j + I_1 50 \quad \text{(E2)}$$

$$I_1 = \frac{5 < 0°}{10.05k + 502.67j} = .4969 < -288°$$

When $V_2 = V_m < \theta, I_2 = 0\ A$; also $V_2 = V_m \sin{(800\pi t + \theta)}$; assuming $V_m = abs(V_2)$;

An expression for M for a sine wave can be found as follows:

$$V_2 = j\omega M I_1 + j\omega L_2 I_2 \quad \text{(E3)}$$

$$M = \frac{V_m}{j\omega I_1} = \frac{abs(V_2)}{j\omega abs(I_1)} \quad \text{(E4)}$$

Then when finding an expression for M when V_2 is known, it is 10V peak to peak and $f_0 = 400\ Hz$.

$$V_{R1} \approx u(t), I_1 = IR_1 = \frac{u(t)}{R_1}, and\ \frac{dI_1}{dt} = \frac{du(t)}{dt}\frac{1}{R_1} \quad \text{(E5)(E6)(E7)}$$

$$\frac{du(t)}{dt} = \begin{cases} 8000\frac{V}{s} & 0 < t < 1.25ms \\ -8000\frac{V}{s} & 1.25 < t < 2.5ms \end{cases} \quad \text{(E8)}$$

So dividing by R_1=10kΩ

$$\frac{dI_1}{dt} = \begin{cases} .8\frac{A}{s} & 0 < t < 1.25ms \\ -.8\frac{A}{s} & 1.25 < t < 2.5ms \end{cases} \quad \text{(E9)}$$

Since $I_2 = 0\ A, \frac{dI_1}{dt} = \pm.8\frac{A}{s}$ and a measured V_2; the expression for M for a triangular pulse is as follows:

$$V_2 = j\omega M\frac{dI_1}{dt} + j\omega L_2\frac{dI_2}{dt} \quad \text{(E10)}$$

$$M = \frac{V_2}{\frac{dI_1}{dt}} \quad \text{(E11)}$$

For the prelab, since V_2 is not actually known, the other way was used to find M, given k=.25, $L_1 = .18\ H$ and $L_2 = .19\ H$ as shown:

$$M = \sqrt{kL_1L_2} = \sqrt{.25 * .18 * .19} = .046\ H \qquad \text{(E12)}$$

Experiment 1 Lab Calculations:

Using the measured values for V_2 and I_1 as 36.4 mV and 964.7 uA, M for the triangular pulse was determined as follows:

$$\frac{dI_1}{dt} = \frac{964.7uA}{1.25ms} = .7718\frac{A}{s} \qquad \text{(E13)}$$

$$M = \frac{36.4mV}{.7718\frac{A}{s}} = .04716\ H \qquad \text{(E14)}$$

The experimental M for the sine wave using the measured values of V_2 and I_1 is as follows:

$$M = \frac{abs(36.39mV)}{800\pi(490.12uA)} = .0464\ H \qquad \text{(E15)}$$

Finding the average of the M's is shown below:

$$M_{avg} = \frac{.0464+.04716}{2} = .04677\ H \qquad \text{(E16)}$$

Then taking the average M to find the experimental k is shown below by manipulating (E12):

$$k = \frac{M}{\sqrt{L_1L_2}} = \frac{.04677H}{\sqrt{.18 * .19}H} = .253$$

Appendix F

Lab 2 Checklist

Name:____________________

Lab title and introduction

- Lab title, your name, date, and lab partner.
- Brief introduction (one to three sentences) explaining the purpose of this lab.

I. Mutual Inductance (50 pts total)

1. Diagram of transformer circuit (Figure 1).
2. Measured values for R_1, L_1, L_2, R_{L1}, R_{L2}.
3. *Triangular wave input:* Measured input and output voltage waveforms. Measurement-based estimate of M. Give pre-lab equation used to estimate M.
4. *Sinusoidal wave input:* Measured input and output voltage waveforms. Measurement-based estimate of M. Give pre-lab equation used to estimate M.
5. Calculated value of *k*, including equation used to compute *k*.
6. **DEMO**: Have your TA or instructor check you on blackboard.
    - Circuit operates; responses to triangular and sinusoidal inputs obtained.
    - Response to triangular input consistent with pre-lab analysis. Show results of pre-lab analysis to TA.
    - Response to sinusoidal input consistent with pre-lab analysis. Show results of pre-lab analysis to TA.
    - Explain your dot convention to the TA or instructor.
    - Have the TA initial this sheet, indicating that he/she agrees with your dot convention.
7. Post-lab exercise:
    - MATLAB simulation of response to triangular wave input.
    - MATLAB simulation of response to sinusoidal input.
    - Comparison and discussion of LTSPICE vs. Matlab circuit responses for both triangular and sinusoidal inputs.

II. State Variable Modeling of RLC circuit (50 pts total)

1. Circuit diagram (figure 4).
2. State variable model derived in pre-lab.
3. Measured system states ($x_1(t)$ and $x_2(t)$) plotted vs. time.
4. Measured state trajectory plotted ($x_1(t)$ vs. $x_2(t)$).
5. **DEMO**: Have your TA or instructor check you on blackboard. Indicating that he/she has observed your circuit's operation.
    - Circuit operates; both states measured.
    - TA approves state variable model derived in pre-lab.
    - TA observes the state trajectory.
6. Post-lab exercise:
    - Simulated system states ($x_1(t)$ and $x_2(t)$) plotted vs. time.
    - Simulated state trajectory plotted ($x_1(t)$ vs. $x_2(t)$).
    - Comparison and discussion of LTSPICE simulation vs. Matlab simulation step responses and state trajectories.