

ET 304b
Laboratory 2
Pspice Simulation-Transmission Line Simulation

Objectives: Use a Pspice simulation program to solve an ac network. Compare the simulation results to the a hand calculation and the laboratory measurements. Simulate the effects of a electric transmission line using a Pi-section model. Compute node voltages of the model and compare the results to the laboratory measurements. Observe the effects of frequency on the voltage profile of the circuit.

Theoretical Background

Engineers use circuit analysis software to solve most industrial circuit analysis problems in practice. This software gives quick and accurate solutions to complex circuits that include linear and non-linear elements. Most of the packages use Pspice as the computational engine. The Pspice engine can solve for both the ac or dc response of a circuit. For ac circuits, the user can output the rms voltage and current values or the transient analysis plots. The transient analysis plots are the analytical equivalent of oscilloscope traces from the lab.

Phasor analysis of circuits finds the magnitude and phase angle of voltages and currents. Pspice will compute the magnitude of the ac voltages and currents directly, but finding phase angles requires the time differences from the transient plots and a simple hand calculation. To find the phase angle of a circuit quantity, select a voltage reference phasor. The source voltage of the circuit is usually set to zero and should be use as the reference phasor for other measurements. Display the transient analysis plots of the source and the quantity to measure. Use the cursors to measure the time difference between the two quantities, then convert the time difference to phase shift using the equation below.

$$\phi = \frac{\Delta t}{T} \cdot 360^\circ$$

Where : Δt = the time difference between the positive going zero crossings.
 T = the period of the sine wave, which is $1/f$ with frequency in Hertz.
 ϕ = phase shift in degrees.

Make sure that the simulation time and the time step for the transient analysis have appropriate values so that the plots are smooth sinusoids.

One application of Pspice is the analysis of electric transmission lines. A transmission line can supply high power, such as those that connect generating plants, or low power like those associated with communication applications. The communications transmission line models provide a wide range of circuit analysis problems because at high frequencies the interaction of the capacitive and inductive effects of the line are distributed along its entire length.

Coaxial cable is an example of the type of wiring used in high frequency transmission lines. In coaxial cables, signals travel through a central conductor and a circular outer braided conductor. With this arrangement of conductors, a capacitance exists between the two

conductors that is distributed over the cable length. The magnetic coupling between the conductors causes an inductance that is also distributed along the cable. Figure 1 shows a representation of this phenomena using a Pi-model circuit. In the Pi-model, the distributed capacitance of a length of cable is lumped into two capacitors labeled C_1 and C_2 . The inductor L_1 represents the distributed inductance of the length of cable. The resistance of the conductors is neglected for very high frequency applications.

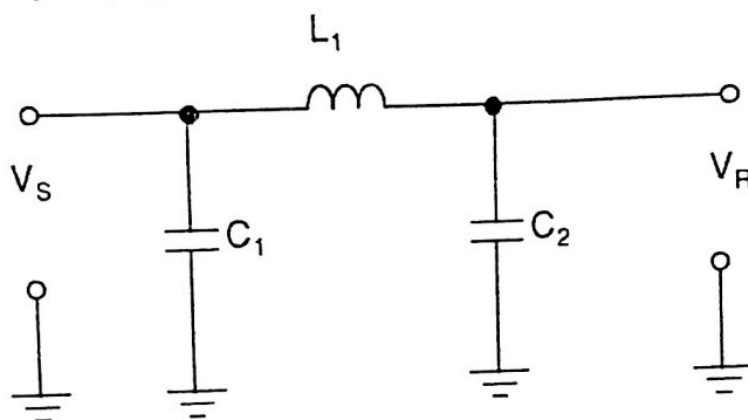


Figure 1. Single Pi-model Section of Transmission Line

The voltages at the sending and receiving ends of the segment are defined as V_S and V_R respectively.

In high frequency communication applications, the voltages and currents are a function of both the terminal conditions and the distance from the sending end of the line. The magnitude and phase angle changes with the distance from the sending end. A number of series connected Pi-model sections approximate this phenomena. As the number of sections increases, the circuit representation will more closely match the actual response of a line. Figure 2 show a three section line model.

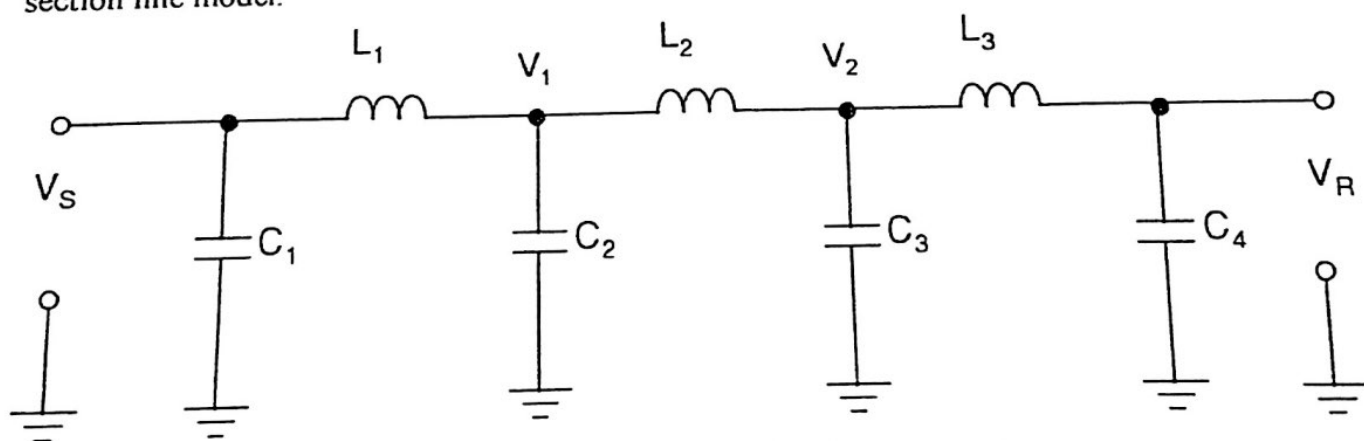


Figure 2. Three Section Pi-Model of a Transmission Line.

The voltages V_1 and V_2 approximate the actual voltages at that point on the cable.

Notice that the repeated Pi-model sections form a ladder network that can be solved using simple circuit analysis techniques, but this method quickly becomes impractical as the number of sections increases. Solving the transmission line model using nodal analysis easily finds the voltages along the line even when the number of sections is large. A computer program can efficiently develop the equations and solve them when component values are given.

Use the same formula to find the error between the calculated values from Table 8, the circuit including resistances, and the measured values. Include these results in the lab report and discuss how the changes affect the errors.

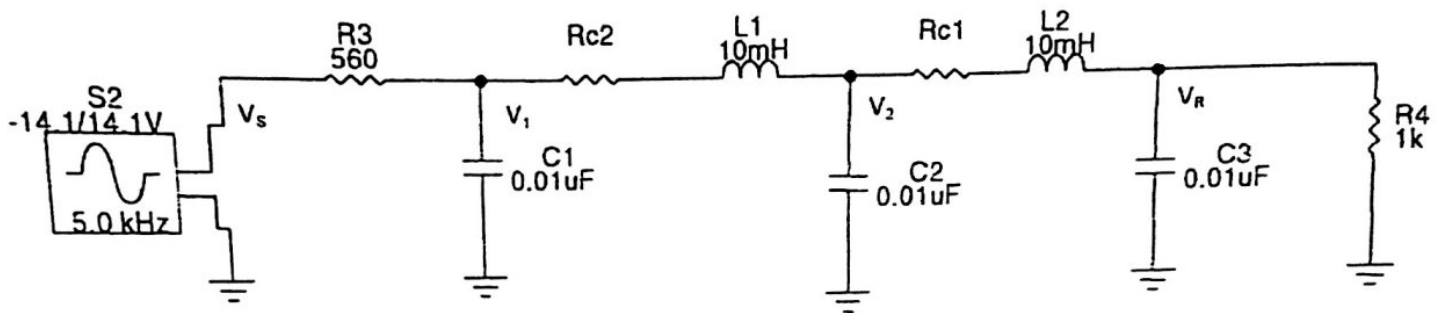


Figure 4. Transmission Line Model Including Coil Resistances.

Graphs and Plots

Construct the following plots for the lab report.

- 1.) Location number (x-axis) Vs Table 1 voltage magnitudes (y-axis)
- 2.) Location number (x-axis) Vs Table 2 voltage magnitudes (y-axis)
- 3.) Location number (x-axis) Vs Table 3 voltage magnitudes (y-axis)

Discussion Points

How well do the experimental results match the theoretical calculations done by nodal analysis? How well do the experimental results compare to the Pspice model with resistance neglected? Does including the resistance in the circuit model give a better match with the measured values? Are the voltages constant along the nodes of the circuit? Do the nodal voltage profiles of the measured values change with frequency? What causes the change in the voltages as frequency changes?

Transmission Line Pspice Calculation Results (cont.)

Table 6. Pspice Voltages $f=10$ kHz.

Location	$ V $ (V)	Δt	Phase (Degrees)
1 (V_S)			
2 (V_1)			
3 (V_2)			
4 (V_R)			

Table 7. Pspice Voltages $f=20$ kHz.

Location	$ V $ (V)	Δt	Phase (Degrees)
1 (V_S)			
2 (V_1)			
3 (V_2)			
4 (V_R)			

Pspice Calculation Results-Resistance Included

Table 8. Pspice Voltages, Resistance included $f=5$ kHz

Location	$ V $ (V)	Δt	Phase (Degrees)
1 (V_S)			
2 (V_1)			
3 (V_2)			
4 (V_R)			

Experiment 2 Data

Transmission Line Model Voltage Lab Voltage Measurements

Table 1. $f=5000$ Hz

Location	1 (V_S)	2 (V_1)	3 (V_2)	4 (V_R)
$ V $ (V)	10			
Phase (Degree)	0			

Table 2. $f=10,000$ Hz

Location	1 (V_S)	2 (V_1)	3 (V_2)	4 (V_R)
$ V $ (V)	10			
Phase (Degree)	0			

Table 3. $f=20,000$ Hz

Location	1 (V_S)	2 (V_1)	3 (V_2)	4 (V_R)
$ V $ (V)	10			
Phase (Degree)	0			

Transmission Line Hand Calculations

Table 4. Nodal Analysis Voltage Values $f=5000$ Hz.

Location	1 (V_S)	2 (V_1)	3 (V_2)	4 (V_R)
$ V $ (V)	10			
Phase (Degree)	0			

Transmission Line Pspice Calculation Results

Table 5. Pspice Voltages $f=5000$ Hz.

Location	$ V $ (V)	Δt	Phase (Degrees)
1 (V_S)			
2 (V_1)			
3 (V_2)			
4 (V_R)			

Procedure

- 1.) Construct the circuit shown in Figure 3. Use the inductor substitution boxes in the lab for the inductors. Set the function generation output to a 10 Vac rms sine wave and connect it to the circuit.

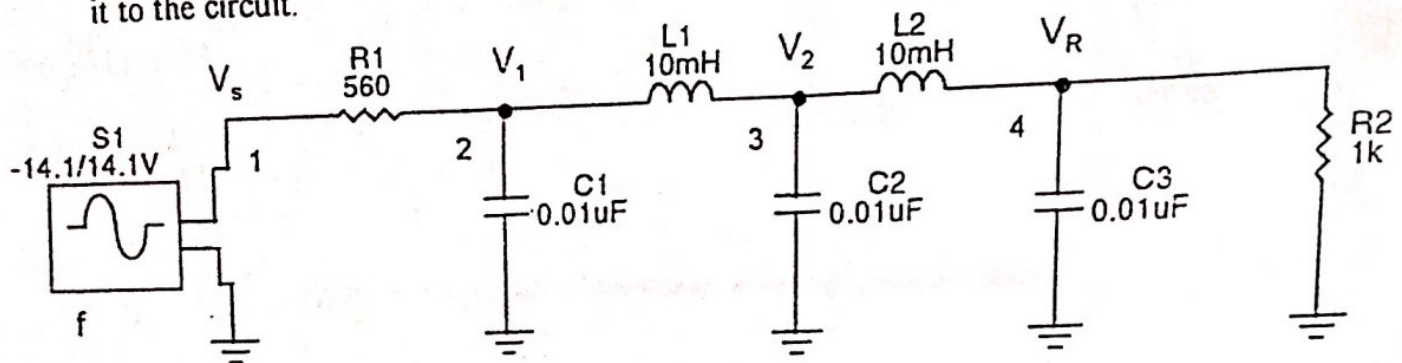


Figure 3. Transmission Line Approximation Circuit Excluding Resistance.

- 2.) Set the source frequency to 5000 Hz and measure the magnitude and phase angles of the voltages V_s , V_1 , V_2 , and V_R . Record these values in Table 1 of the data sheet.
- 3.) Set the source frequency to 10 kHz and repeat step 2 entering the values in Table 2 of the data sheet.
- 4.) Set the source frequency to 20 kHz and repeat step 2 entering the values in Table 3 of the data sheet.
- 5.) Solve for the voltages in Figure 3 with $f=5000$ Hz using nodal analysis. Enter these values into Table 4. Note the differences in the measured and calculated voltages and try to explain the differences in the report discussion.
- 6.) Enter the schematic of Figure 3 into the Pspice simulation software provided and find the magnitudes and phase angles of V_s , V_1 , V_2 , and V_R . Find the phase angles from the transient analysis plots by measuring the time difference between the positive going zero crossing of V_s and the other voltage waveforms. Use the formula presented above to calculate the phase angle. Enter these values in Tables 5-7 for each of the test frequencies.
- 7.) For each of the test frequencies, print transient analysis plots that show both the voltage V_s and V_R waveforms. Note the phase shift and change in magnitude between the sending and receiving voltages. Comment on the effect that frequency had on these results in the discussion section of the report. Include these three printouts in your lab report.
- 8.) Measure the dc resistance of the inductors using a multimeter. Modify the Pspice model to include the coil resistance of the inductors. The resulting schematic should look like Figure 4.
- 9.) Simulate the circuit of Figure 4 for $f=5000$ Hz only. Find the magnitudes and phase angles of the voltages. Record the results in Table 8 for later use.
- 10.) Calculate the percentage error between the voltages in Table 4, the hand calculations, and the measured values for Table 1 using the following formula:

$$\%error = \frac{V_{hand} - V_{measured}}{V_{hand}} \cdot 100\%$$

$= 314 \angle 90^\circ$