EEE 126: Electrical Circuit Simulation Laboratory

Experiment # 05: Analysis of AC Circuits, Observing variables as functions of time using tran.(mutual inductance)

Objective:

The experiment is designed to enable you to study ac circuit analysis in considerably more detail than would be possible without the aid of digital computation technique. Magnetic circuits will also be discussed in this experiment. It is assumed that you are already familiar with the PSPICE and have used PSPICE to analyse DC circuits and transients.

Introduction:

In addition to DC circuit analysis and transient analysis, PSpice can be used to analyse steady-state phasor problems. The syntax for an AC sources are very similar to DC sources. Sources in AC circuits are time-variant. They are alternating current and have both magnitude and phase displacement. Practical sources are generally sinusoidal or near sinusoidal. The behaviour of a circuit is normally evaluated with a sinusoidal source. The steady state voltages and currents that are normally used to evaluate the performance of a circuit can be calculated by applying the circuit laws that are applicable to dc circuits.The transient behaviour is also important under some conditions, but the analysis become more complex compared to steady–state analysis.

SPICE is an ideal software tool for simulating a circuit and for studying the behaviour of voltages,currents and power flow under steady-state and transient condition. We have discussed the SPICE representations of circuit elements and sources in previous experiments. In this Experiment we will simulate the steady-state analysis of a circuit and include mutual inductances. Students are encouraged to apply the basic circuits laws and to verify the SPICE results by hand calculation

Mutual Inductances in PSpice

Users of PSpice often need to model inductors that are magnetically coupled. This may occur in steady-state power system simulations, or in power electronic transient circuit simulations where linear or nonlinear transformer models are used. In some cases it is necessary to model weakly coupled inductors. This tutorial will address the issues of modeling magnetic coupling in these circumstances.

Basic Linear Coupled Inductors
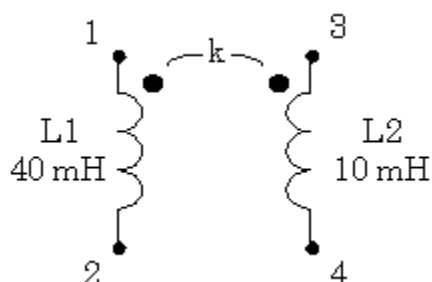
In the above figure two inductors are coupled by a coefficient of coupling, k. Their nodes are designated by small integers, and polarity marks have been added. The polarity information is passed to PSpice by the order of the nodes. If the coefficient of coupling is k = 0.8, a valid PSpice coding could be:

```
*name node1 node2 inductance (comment line)
L1    1     2         40mH
L2    3     4         10mH
*name ind1  ind2  k (comment line)
K12   L1    L2    0.8
```

Note that the polarity marks in the figure are beside nodes 1 and 3 of the inductors. In the listing, these are entered as the leftmost nodes. An equivalent polarity relationship could be indicated by reversing both nodes on both inductors. The coupling of the coils is entered by including a new part that must begin with the letter, K. The "K" part name is followed by a list of the coupled inductors, then by the value of the coefficient of coupling. The coefficient of coupling must occupy the range, $0 \leq k \leq 1$.

Multiple Couplings with Different Values
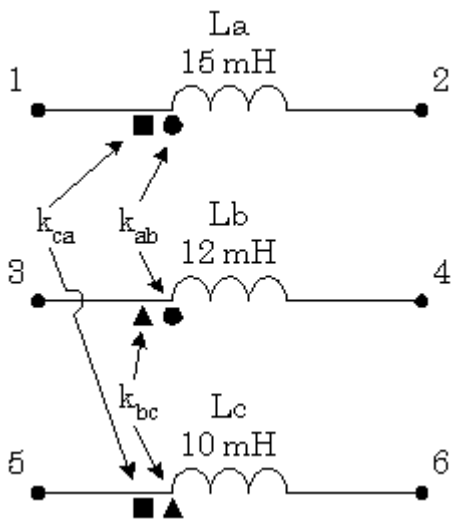
In the above figure, each inductor has mutual coupling with more than one other inductor, but with different coupling coefficient. In this case, PSpice requires a separate "K" part for each coefficient of coupling as shown in the following code:

```
La      1       2       15mH
Lb      3       4       12mH
Lc      5       6       10mH
Kab     La      Lb      0.08
Kbc     Lb      Lc      0.075
Kca     Lc      La      0.04
```

Note that the polarities of the inductors, and therefore the sense of the mutual coupling is accounted for by the order of the nodes entered for the self inductance parts. In this case, different symbols have been used in the figure to assure that it is understood which pairs are coupled and in what sense. In general, if there are n coils, there will be ½ n(n-1) "K" parts needed.

Multiple Couplings with Same Values
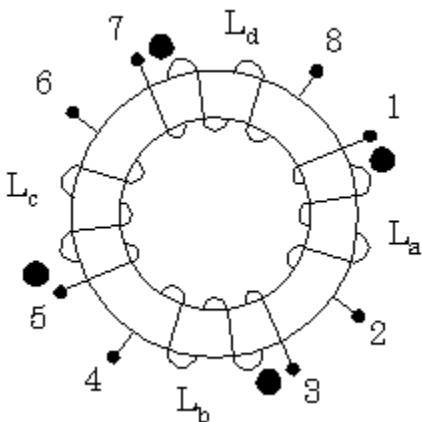
In the above example, we assume that all inductors share identical coefficients of coupling. This is a reasonable assumption when coil symmetry exists and all coils are wound on a common core. Under these conditions, PSpice allows a single "K" part to describe all the coupling.

```
La    1    2     25uH
Lb    3    4     50uH
Lc    5    6    100uH
Ld    7    8    200uH
Kall  La  Lb  Lc  Ld  0.98
```

Again, the polarity information is entered by the order of the nodes for the self inductances. Since all the coupling coefficients were the same, only one "K" part was needed instead of six.

Circuit Example
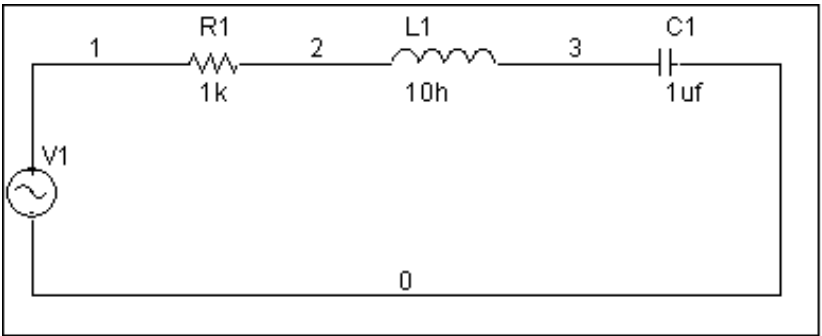
Given the RLC series circuit shown in fig.5.1

FIGURE 5.1

a) Construct the circuit in Figure 5.1 for both netlist and schematic.
b) Setup your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time. Take V1 to be of Magnitude 100V.Choose a suitable frequency.
c) Observe the waveshapes of currents through and voltages across through the resistor ,inductor and capacitor and mention their phase relationship.
d) Observe the mentioned waveshapes for the series resonance condition.

*Schematics Netlist *

```
V_V1      1 0 DC 0 AC 0  SIN 0 100v 1k 0 0 0
C_C1      3 0  1uf
R_R1      1 2  1k
L_L1      2 3  10h
.tran
.probe
.END
```

Report:
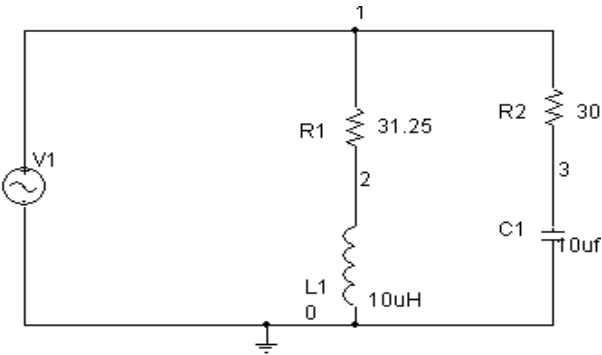**Practice Problem 1:**

FIGURE 5.2

a) Construct the circuit in Figure 5.2 for both netlist and schematic.Take V1 to be of Magnitude 100V.Choose a suitable frequency.

b)Setup your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time.

c)Observe the waveshapes of currents through and voltages across the resistor R1 & inductorL1 and Resistor R2 & capacitor C1 and mention their phase relationship.
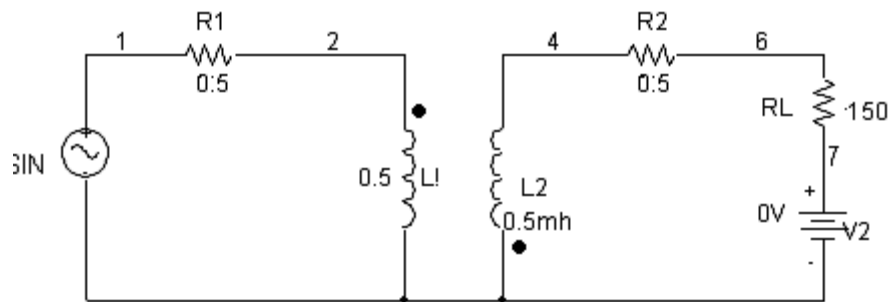
Practice Problem 2:

FIGURE 5.3

a) A circuit with two mutually coupled inductors is shown in fig 5.3. Write down the netlist of the above circuit. If the input voltage is 120v peak with a suitable frequency (say 60Hz,90Hz etc). Calculate the magnitude and phase of the output current. The coefficient of coupling for the transformer is 0.999.

b) Observe the voltage waveshapes at point 2 and 4.

c) Perform the above problem by changing the position of the DOT on the Secondary inductor.