TE WHARE WĀNANGA O TE ŪPOKO O TE IKA A MĀUI
VICTORIA
UNIVERSITY OF WELLINGTON

XMUT204 Electronic Design

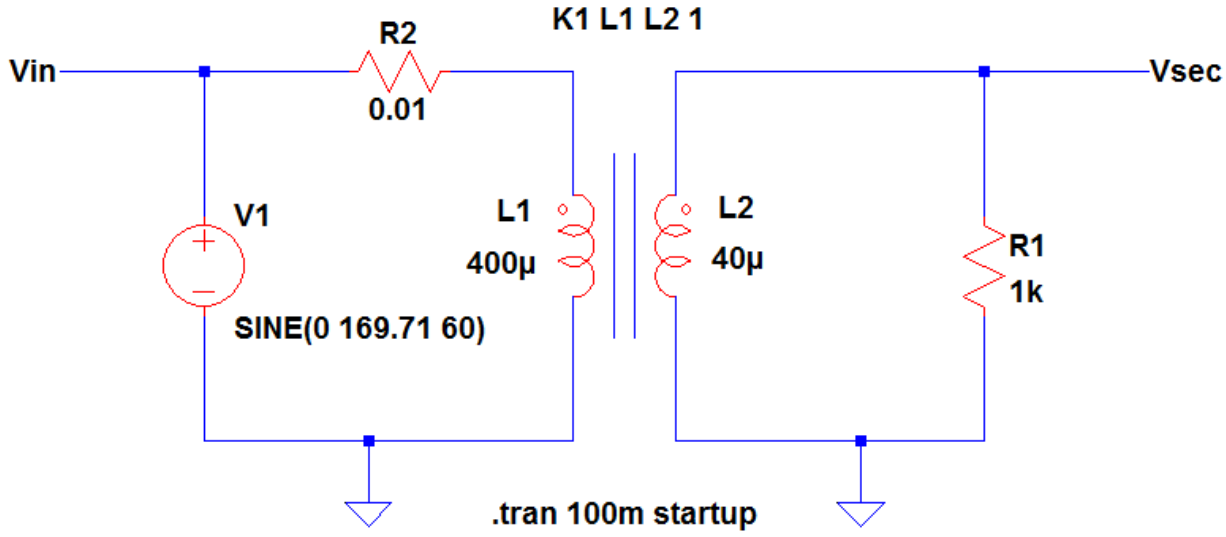
Demo 3ai - Transformer Simulation in LTspice

For ICs, LTspice has many parts available to be used in simulation although most of it is manufactured by Linear Technology (i.e. the maker of the LTspice software). Despite of the mentioned advantages, there are few disadvantages in LTspice. Unlike other circuit simulators, LTspice does not have a transformer part or model in its library. However, there is a special command to come up with a transformer. This exercise is referring on how to make a transformer model in LTspice.

How to Make a Transformer Model in LTspice

A transformer is just coupled inductors. Ideally, the power in the primary will be delivered to the secondary 100%. In real case, there is a reduction of power delivered to the secondary and this is the coupling loss. Coupling effectiveness is represented by a number less than unity to unity in LTspice. Unity means a perfect coupling while less than one is non-perfect coupling.

Below is a transformer model in LTspice environment. L1 and L2 are the primary and secondary inductances. R2 is required for the simulation to run. This can be the primary winding resistance. R1 is the load and not part of the model.



1. Turns Ratio

The turns ratio is very important to transformer. It determines the transformer winding voltages. In most cases, the primary winding voltage is given so the secondary winding voltage will be the unknown term.

$$\text{TurnsRatio} = \frac{\text{Vprimary}}{\text{Vsecondary}} = \frac{\text{Vin}}{\text{Vsec}}$$

For instance, the primary voltage is 120 Vrms, the required turns ratio to get a secondary voltage of 12 V is:

$$\text{TurnsRatio} = \frac{\text{Vprimary}}{\text{Vsecondary}} = \frac{\text{Vin}}{\text{Vsec}} = \frac{120\text{V}}{12\text{V}} = 10\ \text{Turns}$$

2. Inductance and Turns Ratio Relation

LTspice transformer is set in terms of primary and secondary inductances. The relation between inductance and turns ratio is:

$$\text{TurnsRatio}^2 = \frac{\text{PrimaryInductance}}{\text{SecondaryInductance}}$$

For instance, the primary winding voltage is 120 Vrms (169.71 peak), to get around 53 V peak on secondary winding, the inductance ratio should be:

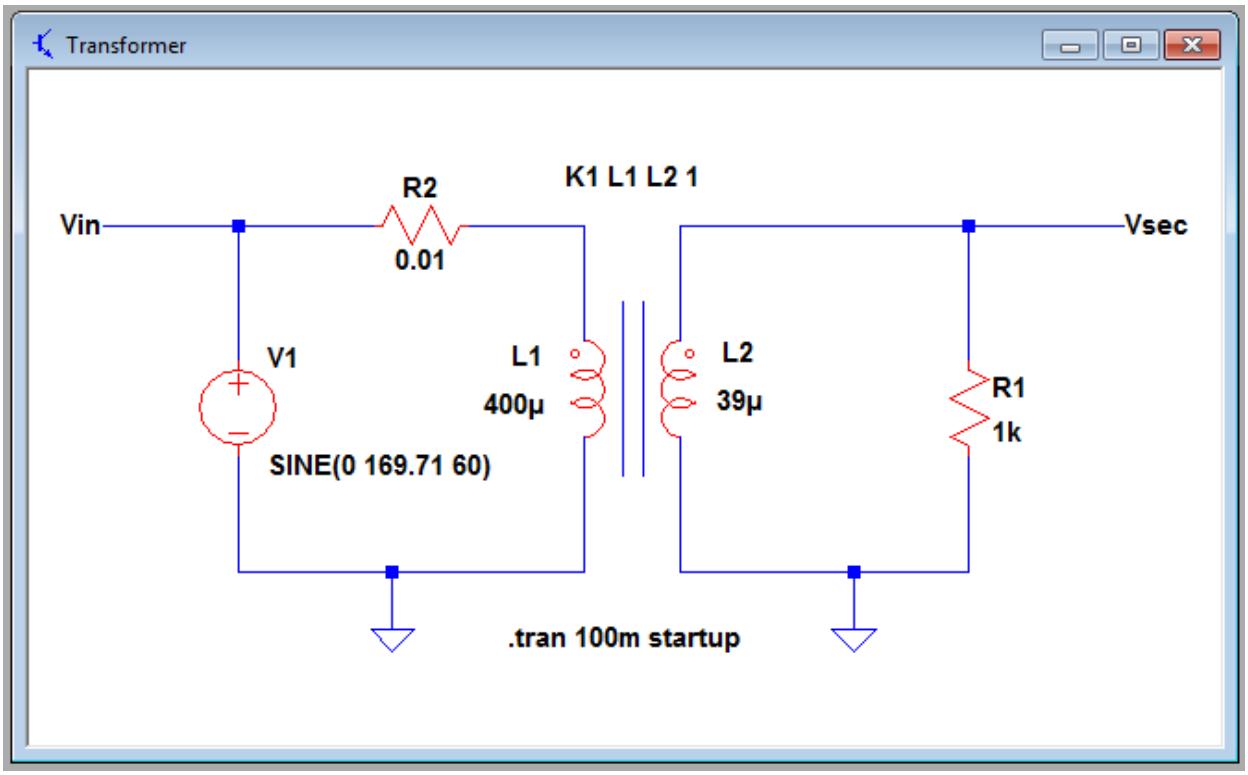
$$\frac{\text{PrimaryInductance}}{\text{SecondaryInductance}} = \text{TurnsRatio}^2 = \left(\frac{\text{Vprimary}}{\text{Vsecondary}}\right)^2 = \left(\frac{169.71\text{V}}{53\text{V}}\right)^2 = 10.25$$

For a 400 micro Henry primary inductance, the secondary inductance would be:

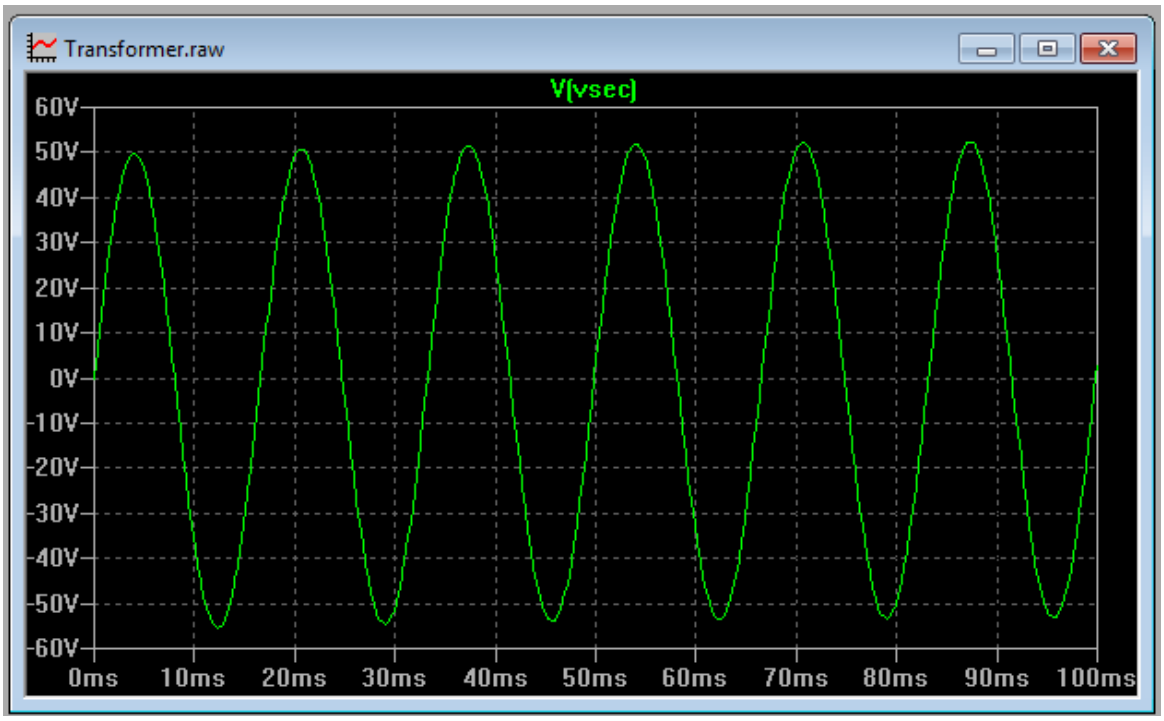
$$\frac{\text{PrimaryInductance}}{\text{SecondaryInductance}} = 10.25$$

$$\text{SecondaryInductance} = \frac{\text{PrimaryInductance}}{10.25} = \frac{400\text{uH}}{10.25} = 39\text{uH}$$

The resulting model will be looking like the figure given below.

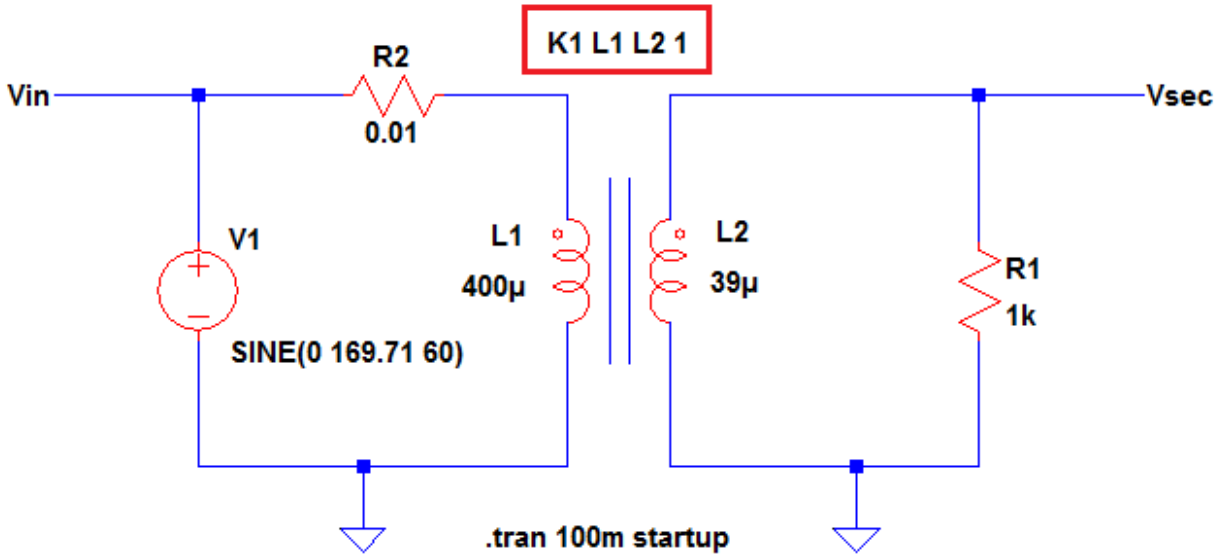


If run the simulation, the secondary voltage waveform is:



3. Declaration to Make a Schematic Model in LTspice for a Transformer

"K1 L1 L2 1" is the command needed to declare a coupled inductor to act as a transformer. K1 is simply a notation that you want L1 and L2 to be inductively coupled to each other. The constant "1" means a perfect coupling. It can be less than one to simulate a non-ideal coupling.



To Setup K1 L1 L2 1, follow below step-by-step guide.

- Press Letter "T" on the keyboard.
- Tick SPICE directive.
- Type "K1 L1 L2 1" in the pop up window.
- Click OK once done typing.
- Click anywhere in the schematic to drop the command.

4. Run Transient Command

To verify the result, a transient command is can be used. To configure a transient command, follow the instructions below.

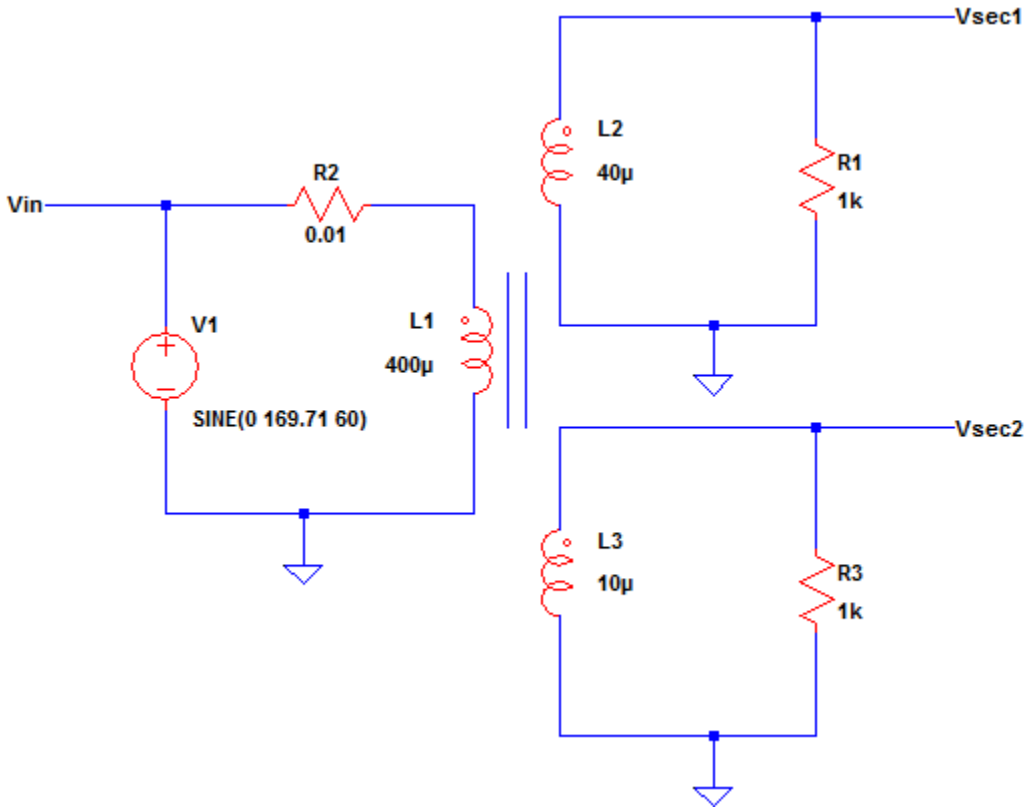
- Navigate to the Transient folder.
- Type in the Stop Time: 100m
- Check on Start external DC supply voltages at 0 V.
- Click OK button once finished.

5. How to Make a Transformer Model in LTspice with Multiple Secondary Windings

Follow above instructions on how to compute for the inductances. A multiple secondary transformer is can be done by making use of the spice directive window.

To demonstrate well, let us consider below circuit. L1 is the primary inductance of the primary winding while L2 and L3 are the inductances of the secondary windings.

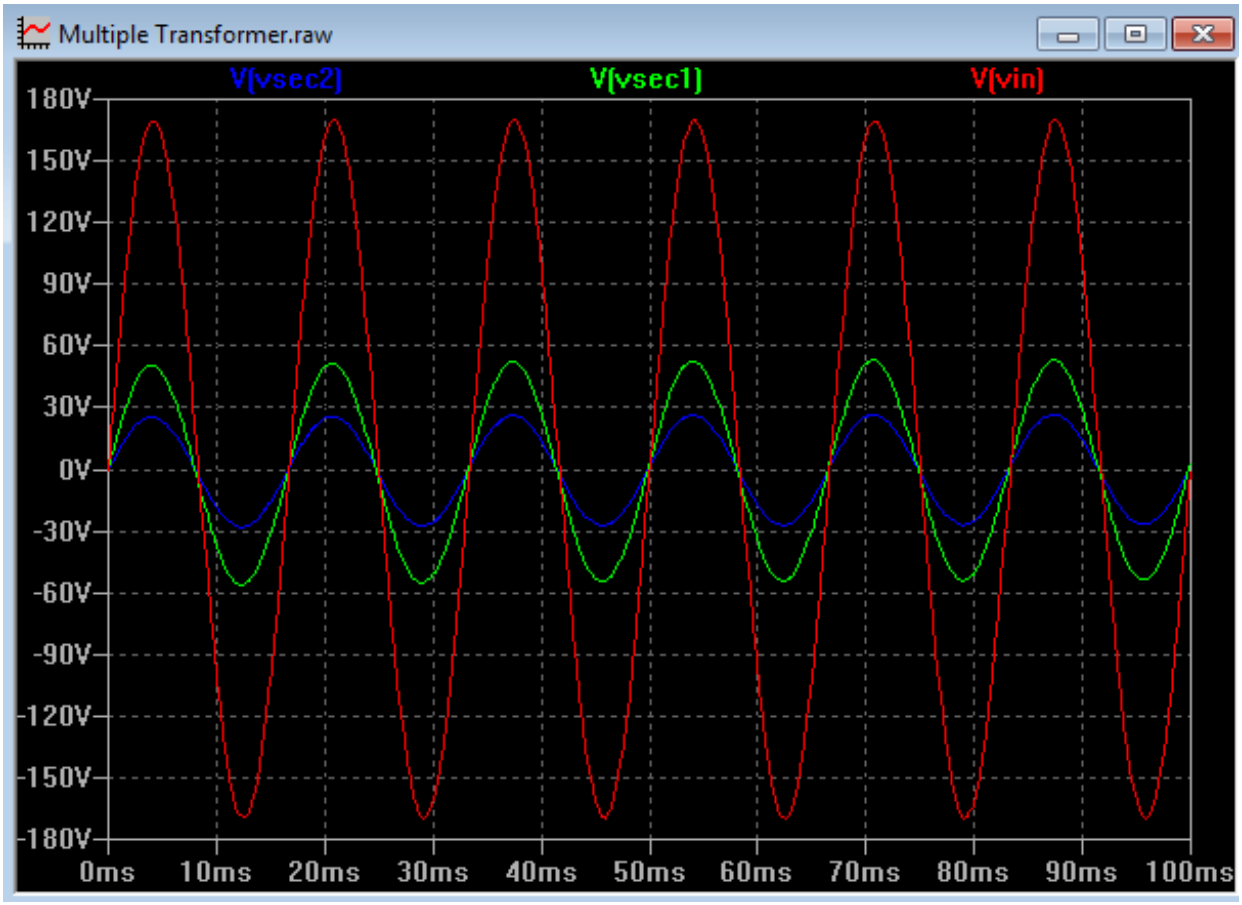
Be mindful of the dots in the secondary inductors; they must be in the same orientation with the dot of the primary inductor to have an in-phase output.



To declare L1, L2 and L3 as one transformer with two secondary windings, follow below instructions.

- Press letter “T” on the keyboard.
- Tick SPICE directive.
- Type “K1 L1 L2 L3 1” in the window.
- Click OK button once done typing
- Click anywhere in the schematic to drop the command.

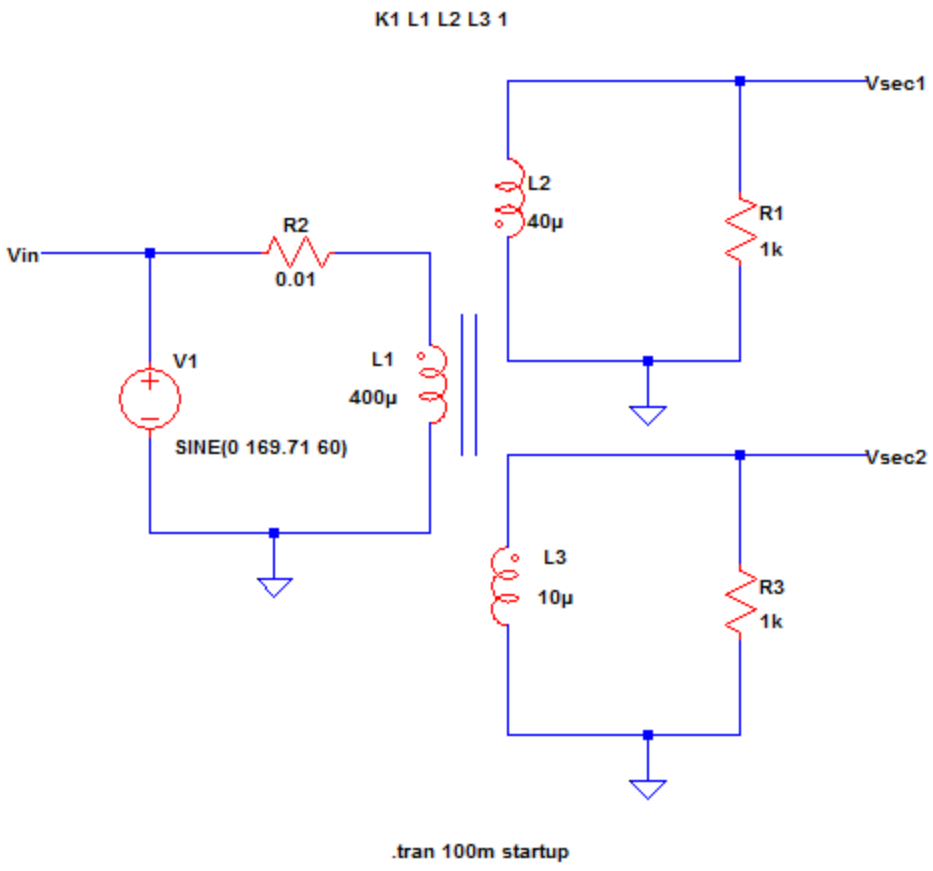
Then setup a transient command in step number 6 to view the result. Below are the waveforms.



Notice also that all waveforms are in phase with each others.

If you want an inverted secondary voltage, do not follow the dot orientation of the primary inductor.

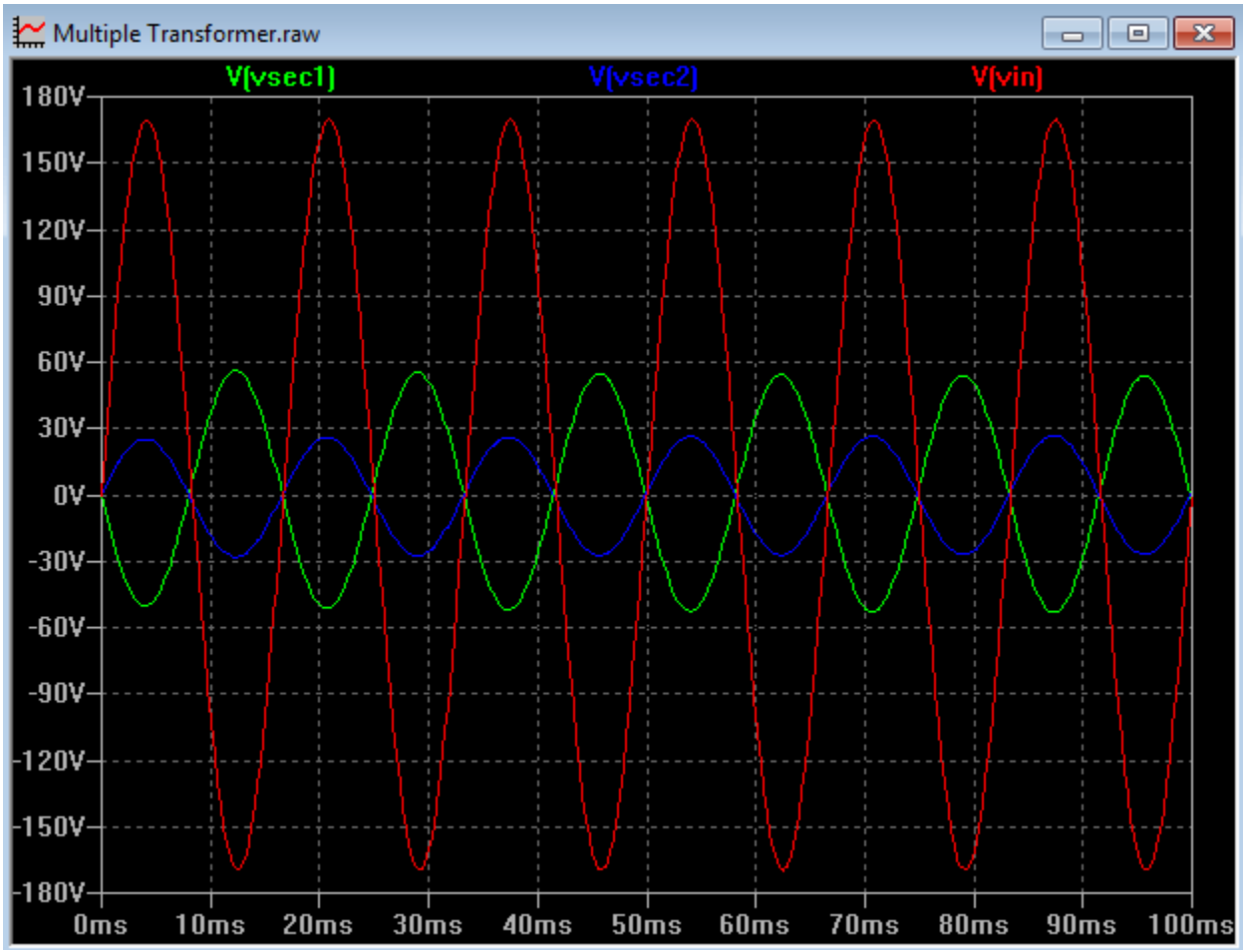
For instance in below circuit, L2 is oriented not the same as L1 and it has a secondary voltage (Vsec1) out of phase of the primary voltage Vin.



The resulting waveforms of the multiple secondary windings as given above are shown in the figure given below.

Notice the waveforms of the voltages at the primary winding (vin), the first secondary winding (vsec1) and the second secondary winding (vsec2).

Notice also the polarities of the waveforms due to relative phase arrangements of the windings. The first secondary voltage waveform is out of phase with regards to voltage waveforms at the primary and the second secondary windings.


Multiple Transformer.raw
V(vsec1)
V(vsec2)
V(vin)
180V
150V
120V
90V
60V
30V
0V
-30V
-60V
-90V
-120V
-150V
-180V
0ms
10ms
20ms
30ms
40ms
50ms
60ms
70ms
80ms
90ms
100ms