VICTORIA UNIVERSITY OF WELLINGTON
TE HERENGA WAKA
NEW ZEALAND

Demo 3a - Transformer & Linear Power Supply Simulation in LTspice

XMUT204 Electronic Design

Overview

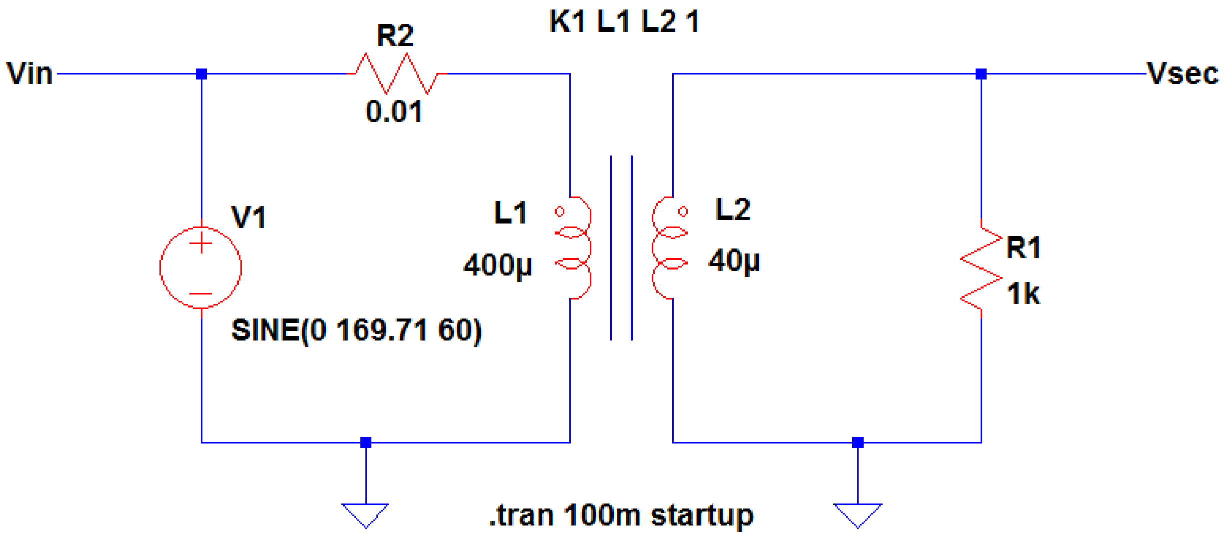
1. Transformer simulations.
    a. Primary & secondary windings
    b. Multiple secondary windings.
2. Power supply simulations.
    a. Transformer.
    b. Bridge full-wave rectifier.
    c. Linear voltage regulator IC.

(Primary and Secondary Windings) Transformer Simulation

- 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 (+ also heat losses).
- 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

(Primary and Secondary Windings) Transformer Simulation

- 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.



(Primary and Secondary Windings) Transformer Simulation

- 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:

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

(Primary and Secondary Windings) Transformer Simulation

- 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:

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

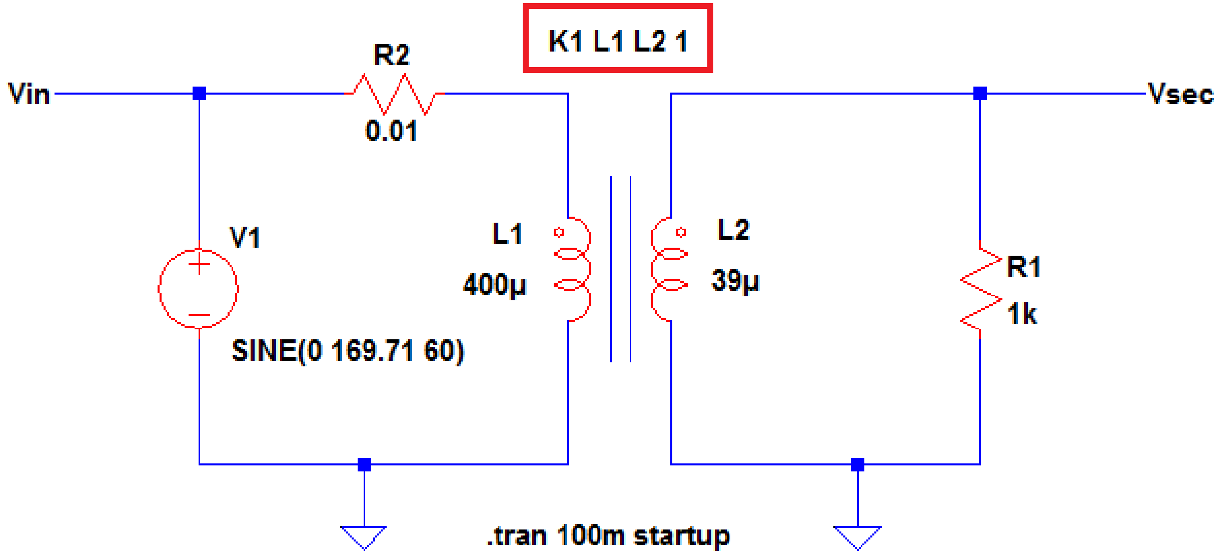
- For a 400 micro Henry primary inductance, the secondary inductance would be

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

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

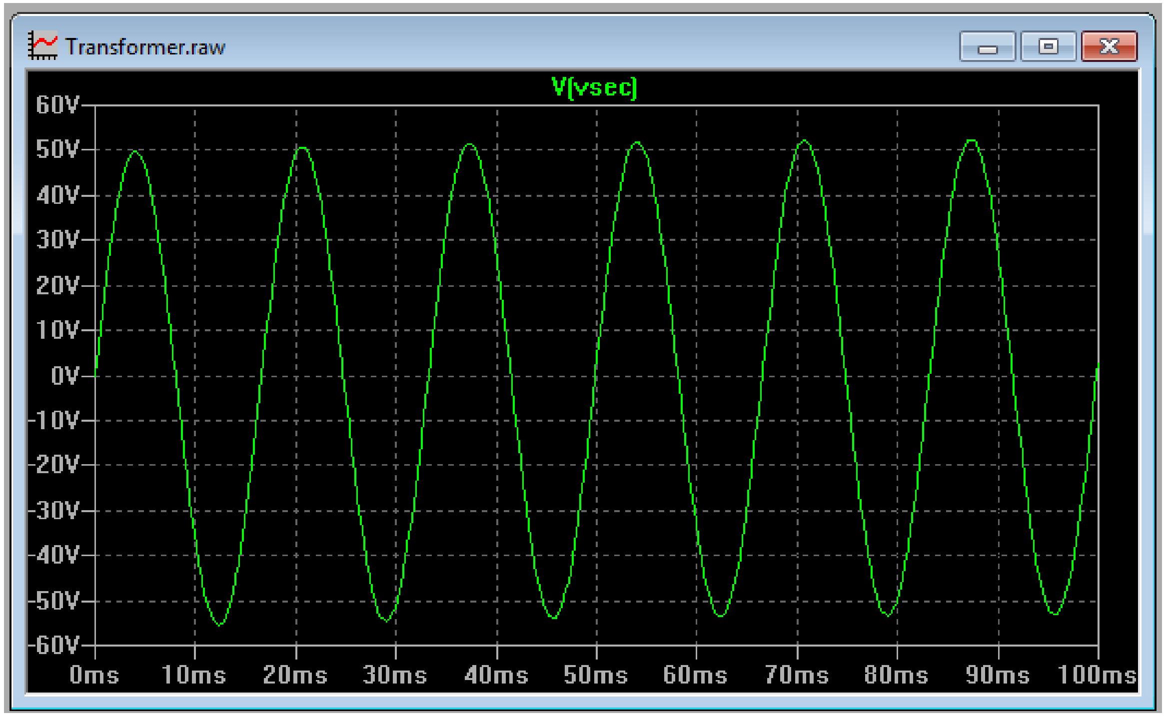
(Primary and Secondary Windings) Transformer Simulation

- “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.



(Primary and Secondary Windings) Transformer Simulation

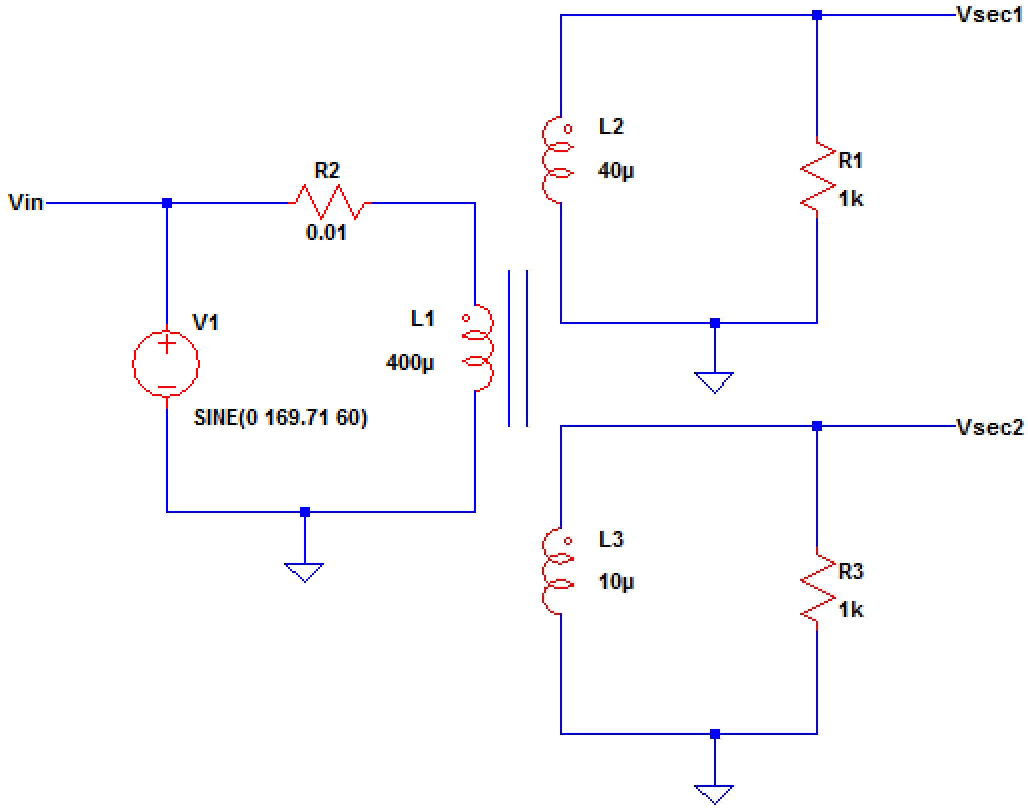
- Voltage at the secondary (Vsec)



(Multiple Secondary Windings) Transformer Simulation

- A multiple secondary transformer is can be done by making use of the spice directive window.

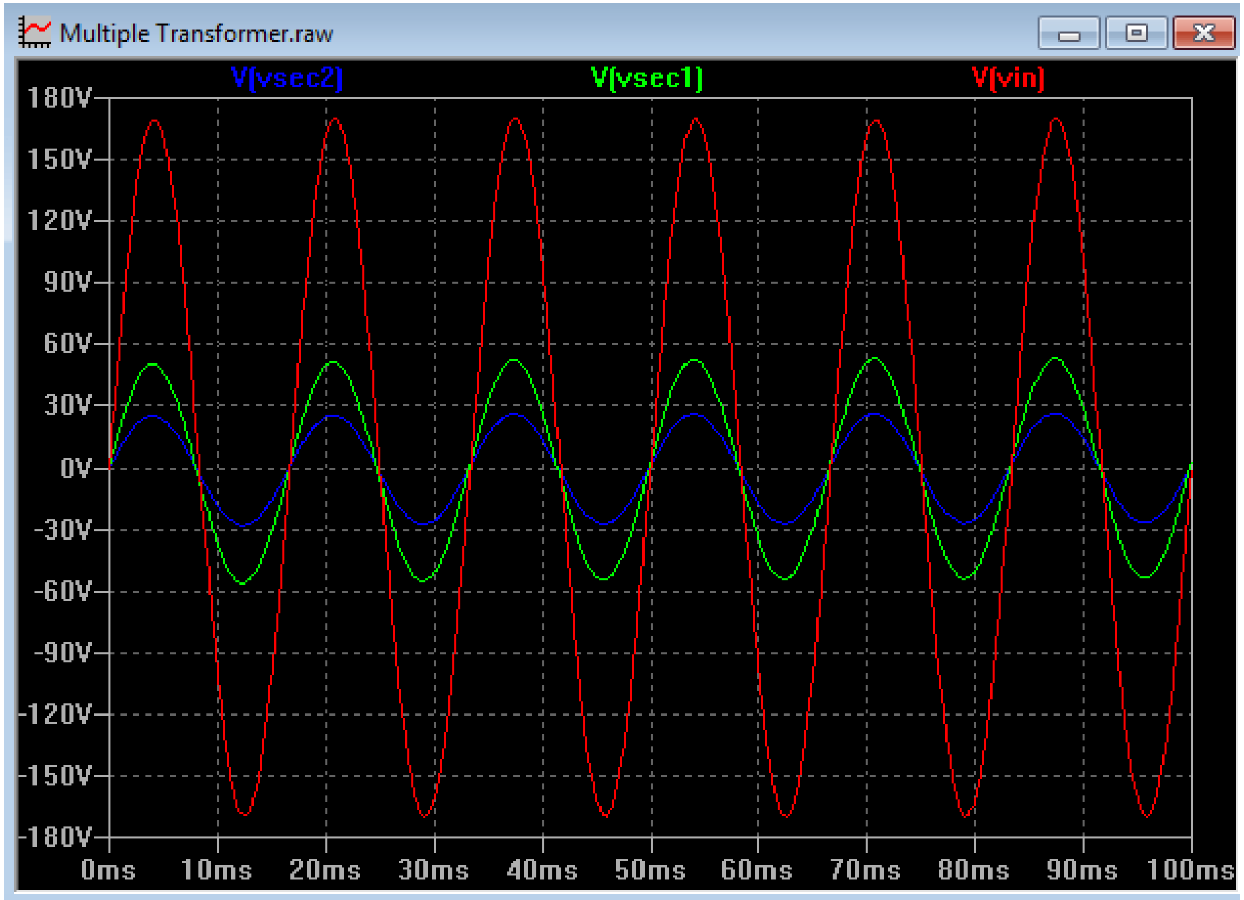
- L1 is the primary inductance of the primary winding while L2 and L3 are the inductances of the secondary windings.



- 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.

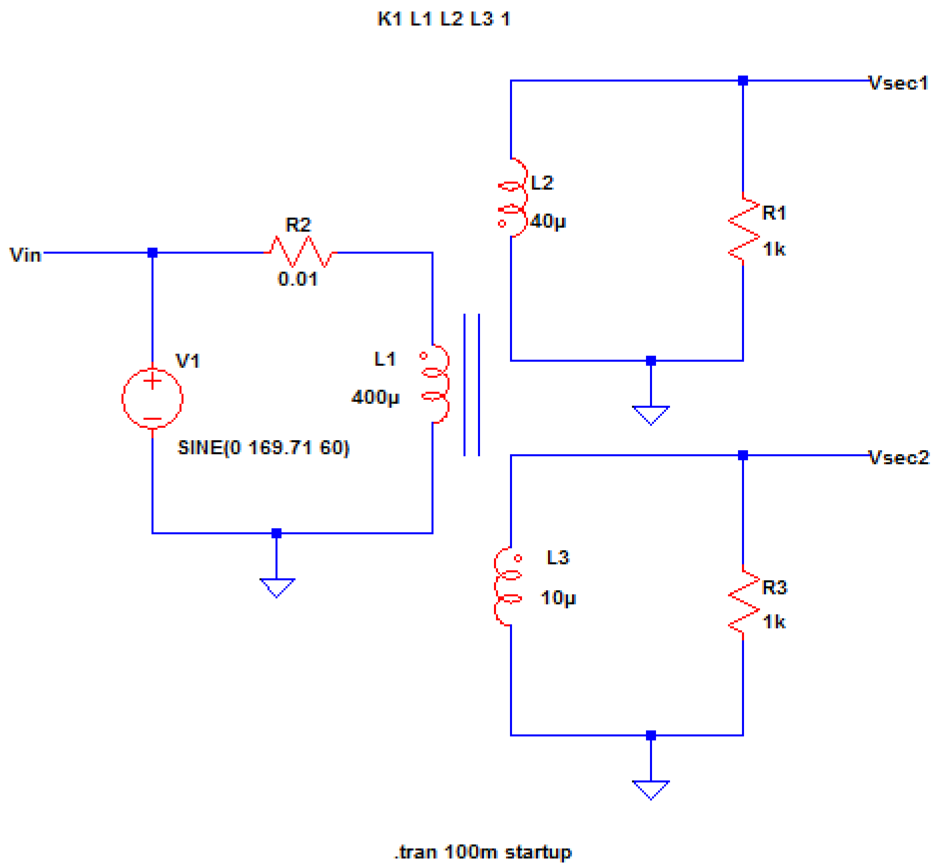
(Multiple Secondary Windings) Transformer Simulation

- Voltage waveforms in primary and secondary windings:

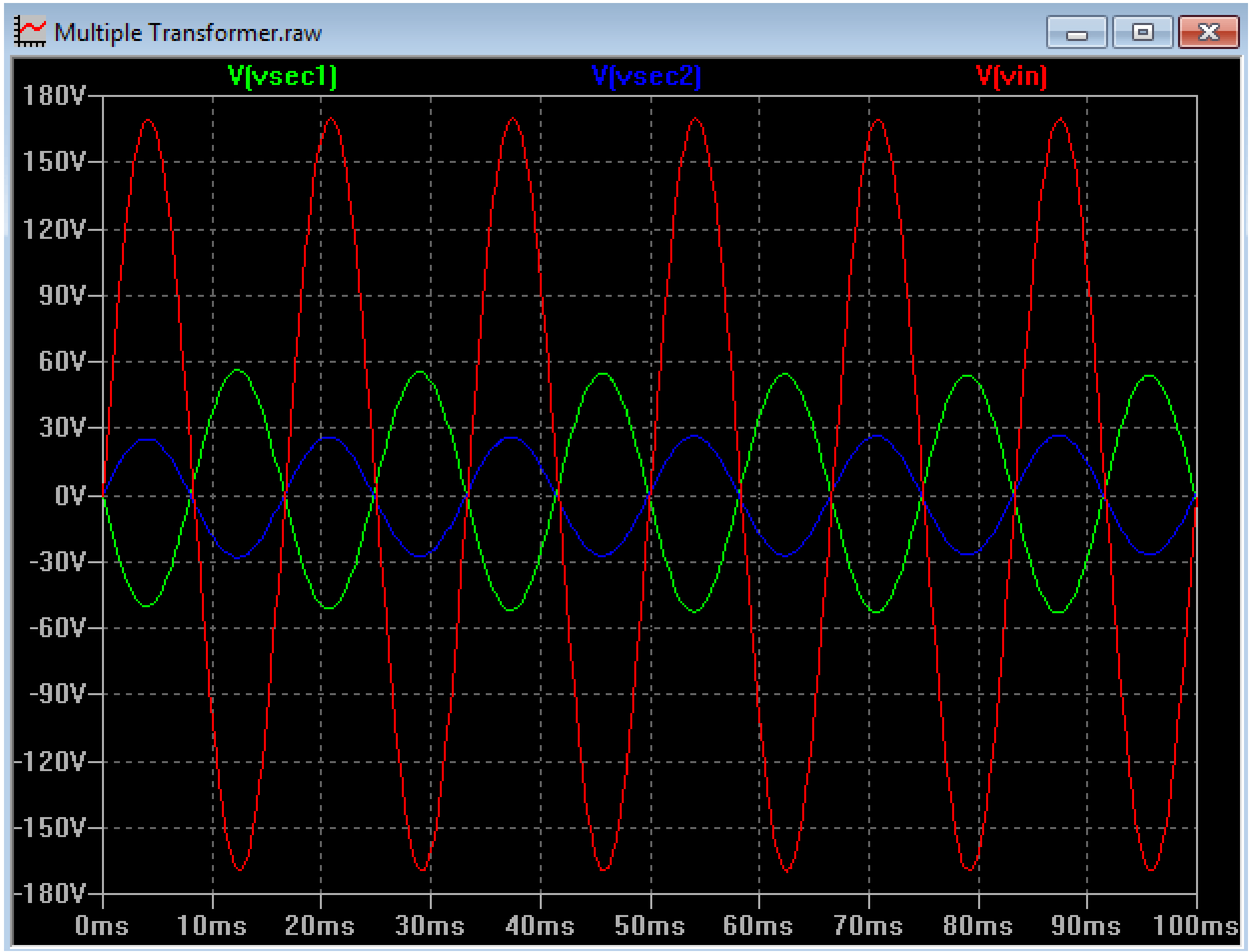


(Multiple Secondary Windings) Transformer Simulation

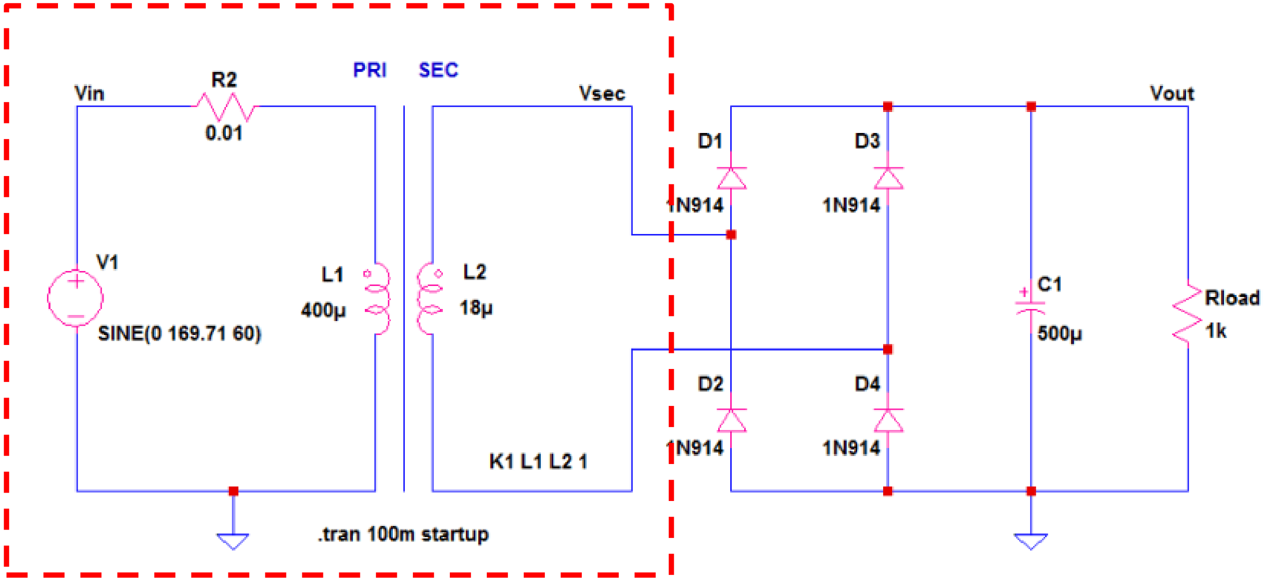
- Exchanging the polarities of the dots in the secondary inductors



(Multiple Secondary Windings) Transformer Simulation



Linear Power Supply Simulation (Transformer Simulation)



- V1 – AC supply - Set the Voltage Source: Sine, Amplitude = 169.71 V (=120 x V2), Frequency = 60 Hz.
- R2 – This is the primary winding resistance. In LTspice, a small resistance in series with the primary winding is needed for the simulation to proceed.
- L1 and L2 are primary and secondary inductances of a transformer. This is how LTspice models a transformer. LTspice does not have an option to enter a turn ratio like other simulators. Instead, the primary and secondary is related through the inductances.

Linear Power Supply Simulation (Transformer Simulation)

- Vin(RMS) = 120 V, Vsec(Peak) = 36 V and Inductance Primary = 400 uH, the secondary inductance must be:

$$\text{TurnsRatio} = \frac{\text{TurnsPrimary}}{\text{TurnsSecondary}} = \frac{\text{VoltagePrimary}}{\text{VoltageSecondary}} = \frac{120 \times \sqrt{2}}{36} = 4.71$$

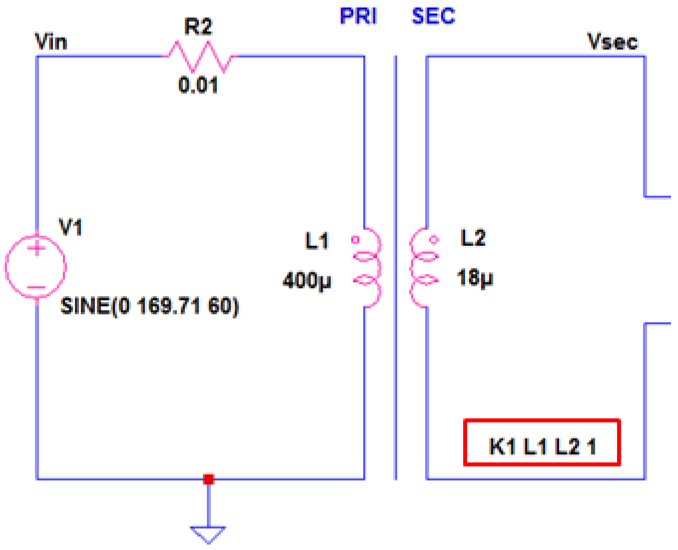
$$\text{TurnsRatio}^2 = \frac{\text{InductancePrimary}}{\text{InductanceSecondary}} = 4.71^2 = \frac{400\text{uH}}{\text{InductanceSecondary}}$$

$$\text{InductanceSecondary} = \frac{400\text{uH}}{4.71^2} = \mathbf{18uH}$$

- In buying a transformer, be sure it can support the desired frequency, voltage and current rating.

Linear Power Supply Simulation (Transformer Simulation)

- K1 means a declaration number 1, L1 and L2 and the primary and secondary inductances while the constant 1 is the coupling coefficient.
- Unity means a perfect coupling. Non-ideal coupling is always less than unity.



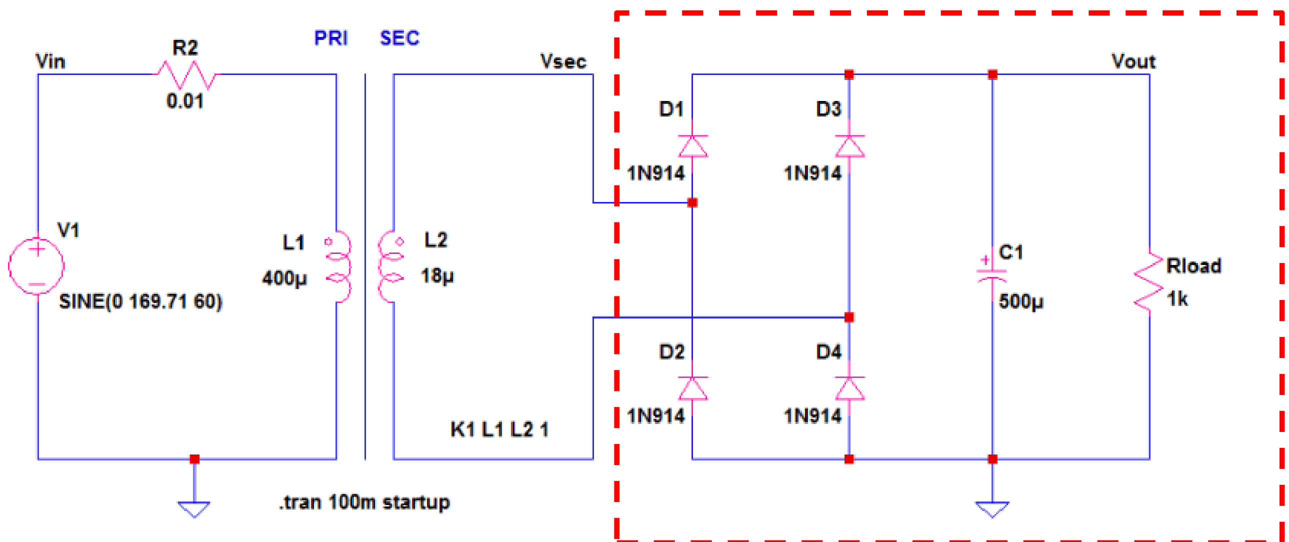
Linear Power Supply Simulation (Bridge FW Rectifier Simulation)

- D1-D4 comprises the bridge rectifier.

  A bridge rectifier is preferred because it can give higher RMS voltage and able to use smaller filter capacitor. The critical parameters to take note are current and peak reverse voltage.

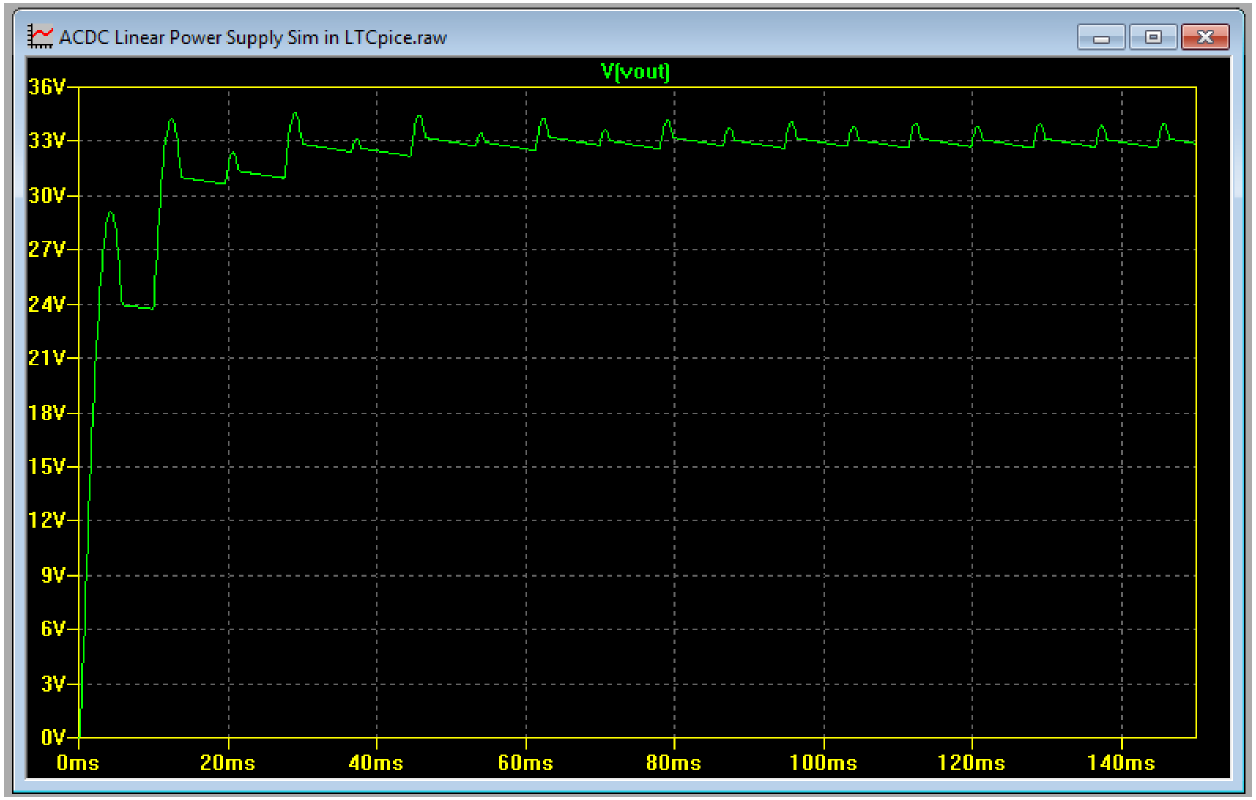
- C1 is the filter capacitor that makes the pulsating DC to almost pure DC. The critical parameter to take note is ripple current and voltage rating.

- Waveforms on the circuit model given above are as follows.



Linear Power Supply Simulation (Bridge FW Rectifier Simulation)

- Voltage at the output (Vout)

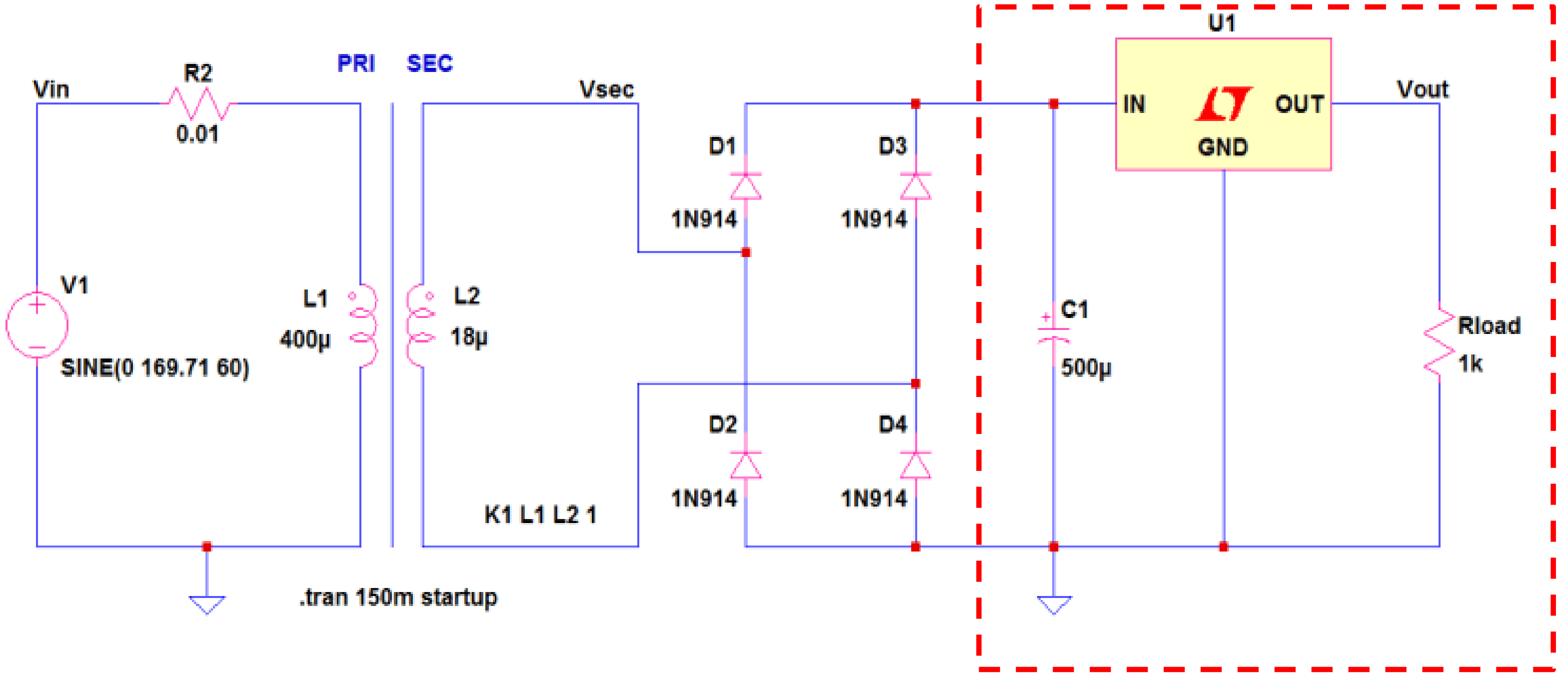


Linear Power Supply Simulation (Linear Regulator Simulation)

- A voltage regulator is needed on the output for more precise and critical applications.
- The most important parameter to consider is the power dissipation of the regulator. The power dissipation of the regulator is just the difference between input and output voltage of the regulator times the load or output current.
- In below circuit, U1 is a linear regulator (in LTspice it is LT1085-3.3 for 3.3 V, LT1085-3.6 for 3.6 V, LT1085-5 for 5 V, and LT1085-12 for 12 V).
- The difference between the voltage on C1 and the output voltage should be not too big to minimize power loss on the regulator.
- However, do not make the difference very low also because linear regulators have minimum dropout voltage requirement for proper regulation.

Linear Power Supply Simulation (Linear Regulator Simulation)

- Power supply design with linear voltage regulator is added



Linear Power Supply Simulation (Linear Regulator Simulation)

- Using above circuit, if the desired output is 24 V and the dropout voltage requirement of the regulator U1 is 3 V, set the voltage on C1 to be around 28 V.
- If the voltage on C1 node is having big ripple, 28 V is not be enough. Insure a low ripple on C1 node. The power dissipation of the above linear regulator will be:

$$\mathrm{Pdiss} = (28\mathrm{V} - 24\mathrm{V}) \times \frac{24\mathrm{V}}{1\mathrm{k}\Omega} = 96\mathrm{mW}$$

- Supposing the load is 10 ohms, the new power dissipation will be

$$\mathrm{Pdiss} = (28\mathrm{V} - 24\mathrm{V}) \times \frac{24\mathrm{V}}{10\Omega} = 9.6\mathrm{W}$$

- For higher regulator dissipations, use heat sink to cool it off. An external heat sink is preferred. For SMD regulators, make the PCB pad as big and thick as possible.