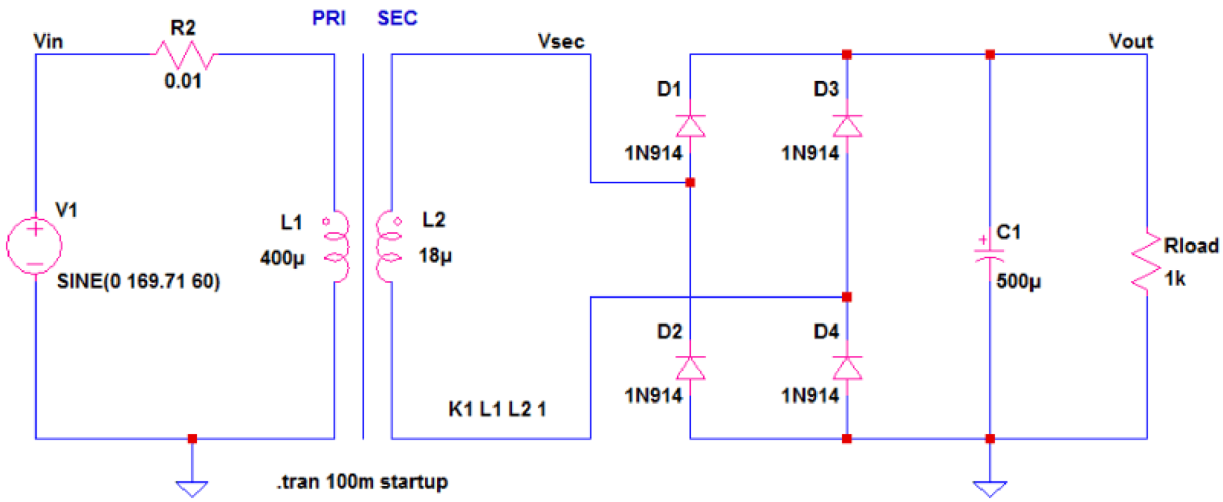
TE WHARE WĀNANGA O TE ŪPOKO O TE IKA A MĀUI
VICTORIA
UNIVERSITY OF WELLINGTON

XMUT204 Electronic Design

Demo 3aii - Linear Power Supply Simulation in LTspice

This practical exercise shows the steps to carry out simulation of linear power supply in LTspice.



1. V1 – AC supply - How to Set the Voltage Source: Sine, Amplitude = 169.71 V (=120 x V2), Frequency = 60 Hz.

2. 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.

3. 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.

$$\text{TurnsRatio} = \frac{\text{TurnsPrimary}}{\text{TurnsSecondary}} = \frac{\text{VoltagePrimary}}{\text{VoltageSecondary}}$$

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

For example:

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

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

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

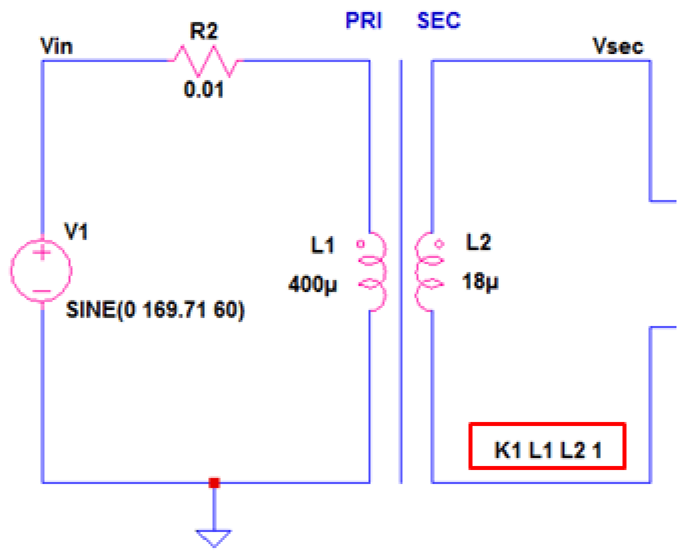
$$\text{InductanceSecondary} = \frac{400\text{uH}}{4.71^2} = \mathbf{18uH}$$

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

4. Transformer Declaration K1 L1 L2 1

   This is the way LTspice understand a transformer. K1 means a declaration number 1, L1 and L2 and the primary and secondary inductances while the constant 1 is the coupling coefficient.

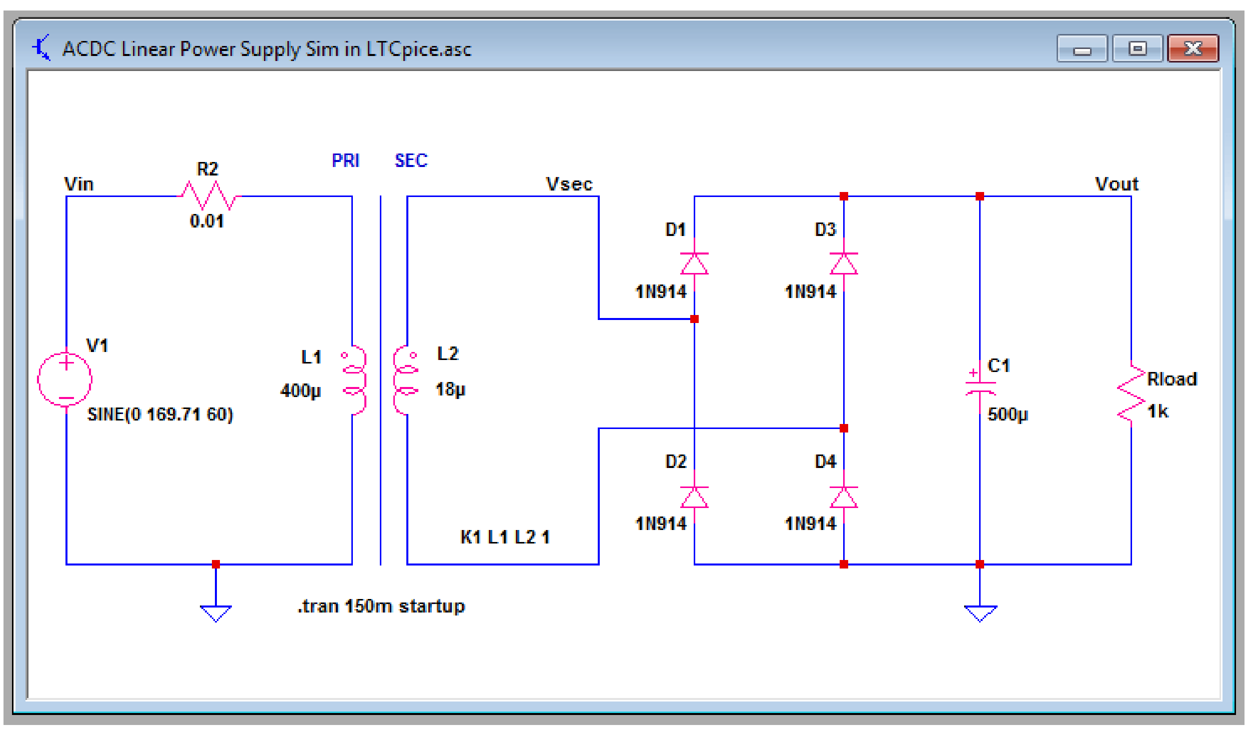
   Unity means a perfect coupling. Non-ideal coupling is always less than unity.



5. Simulation Command .tran 100m startup

   This is a transient simulation command. To set, follow below guides in Edit Simulation Cmd in the Simulate menu bar.

   Stop Time = 10m, Start External DC supply voltage at 0 V = check.



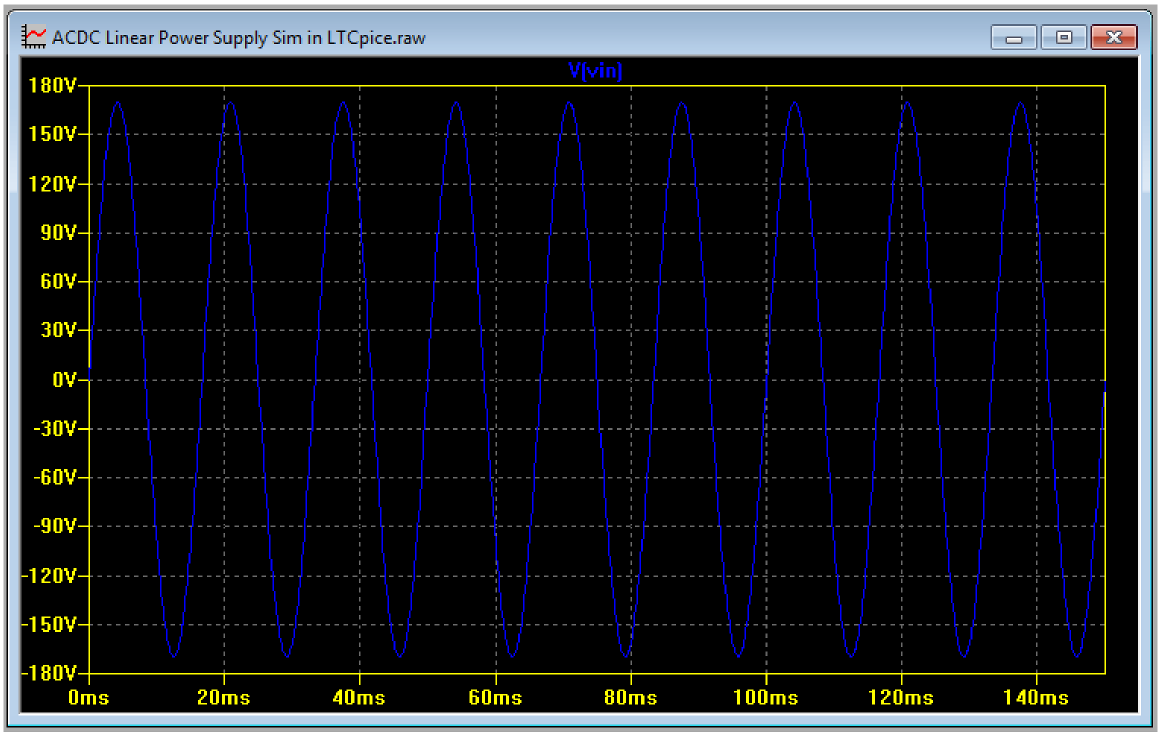
6. D1-D4 comprises the bridge rectifier.

   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.

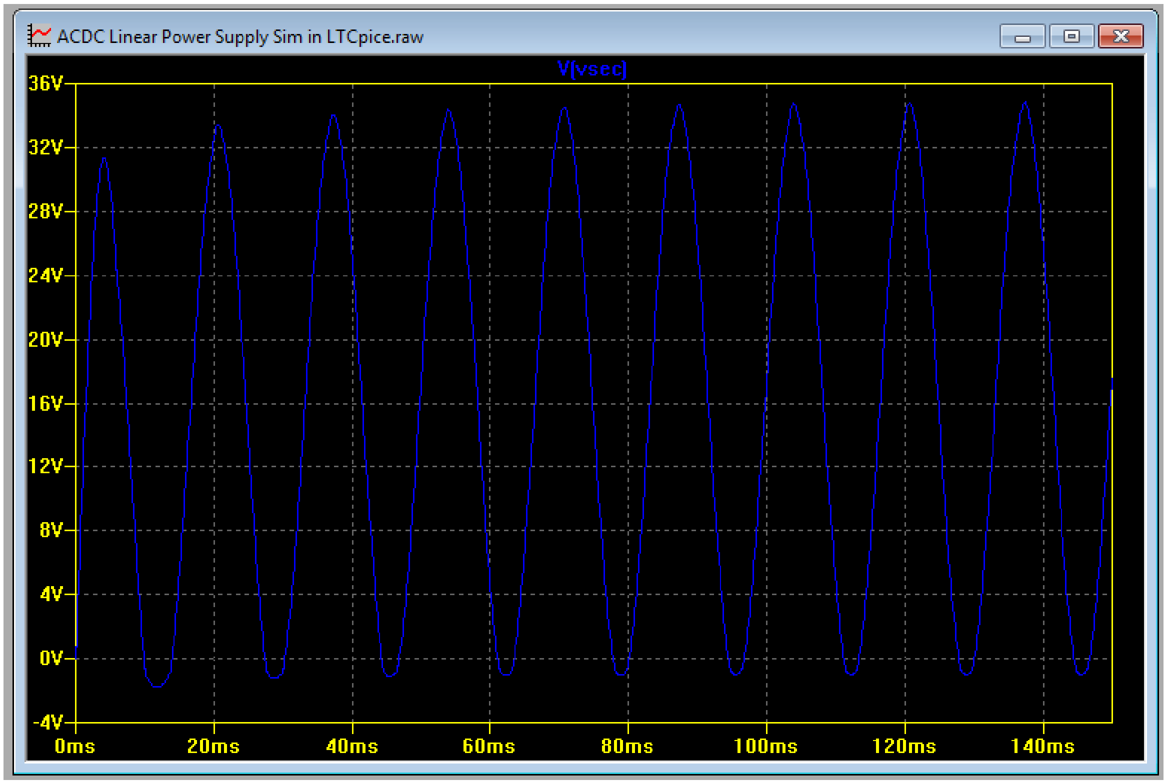
7. C1 is the filter capacitor that makes the pulsating DC to almost pure DC. The critical parameter to take note is ripple current and voltage rating.

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

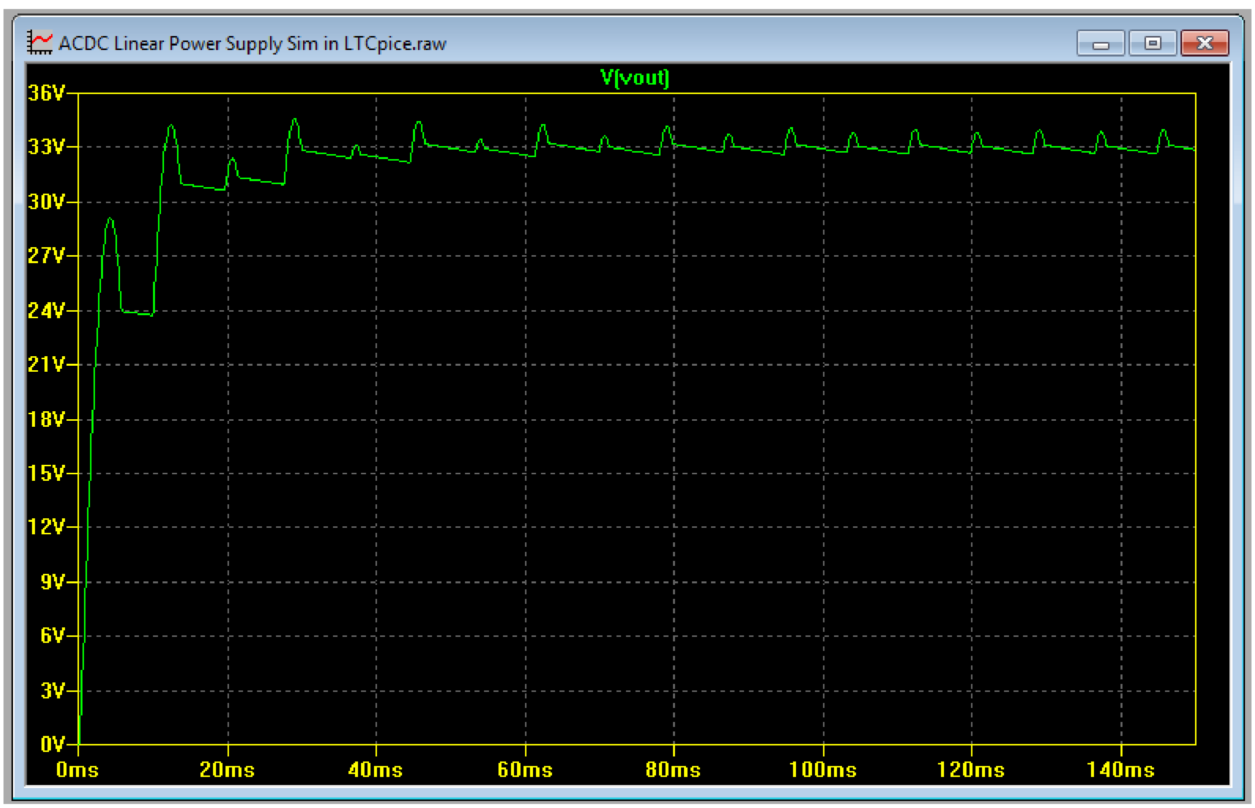
   Voltage at the input (Vin)



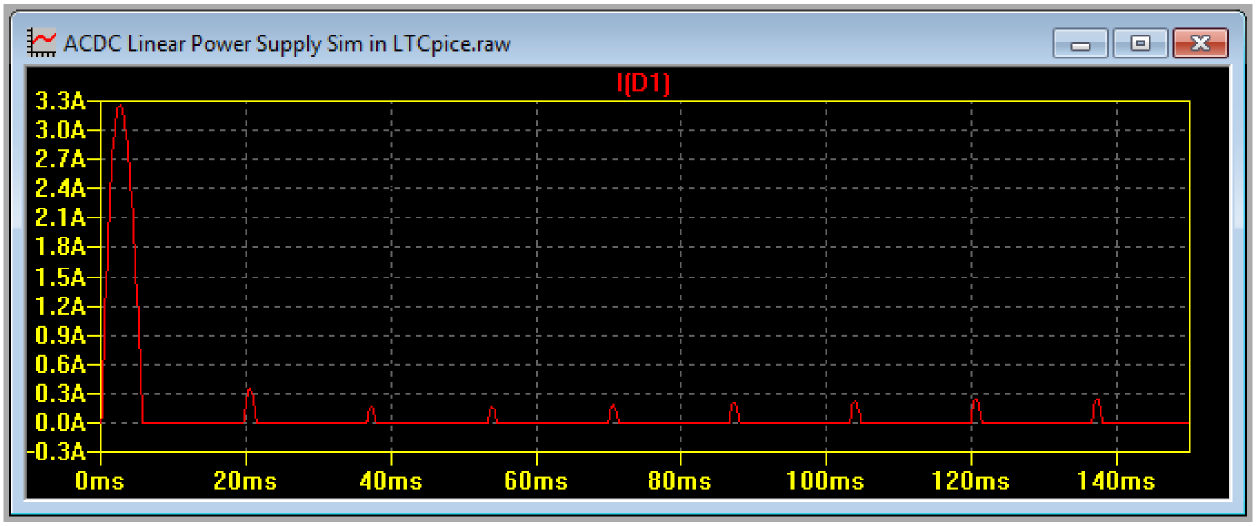
   Voltage at the secondary (Vsec)



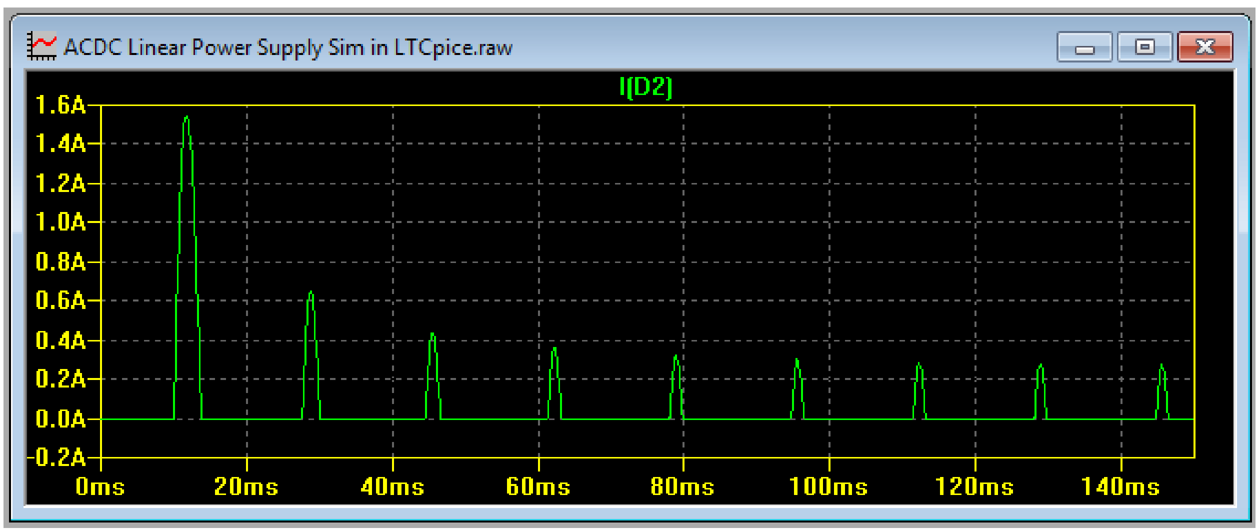
Voltage at the output (Vout)



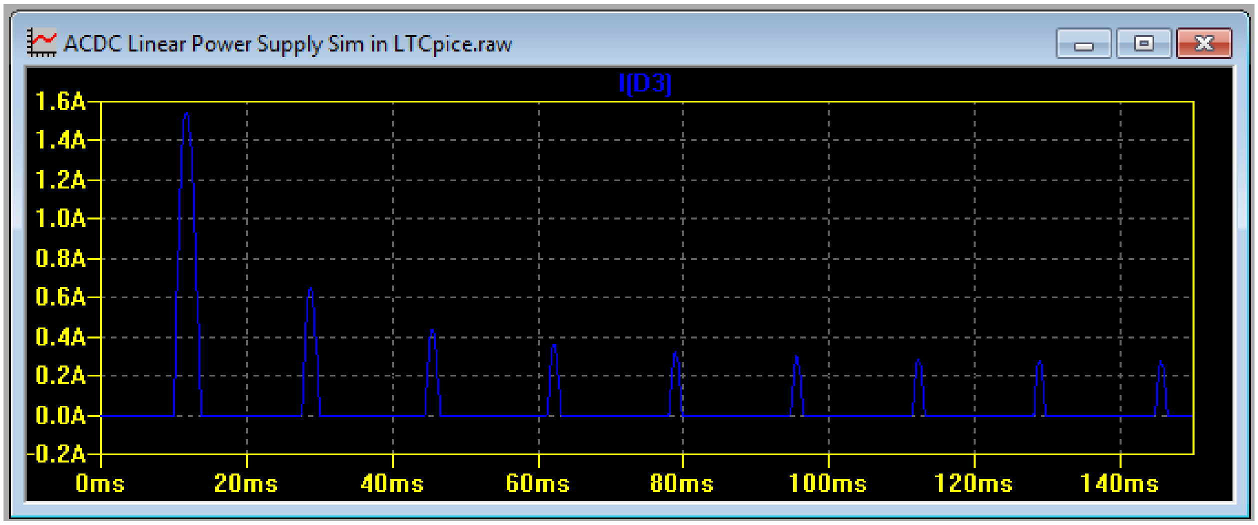
Current in D1 (ID1)



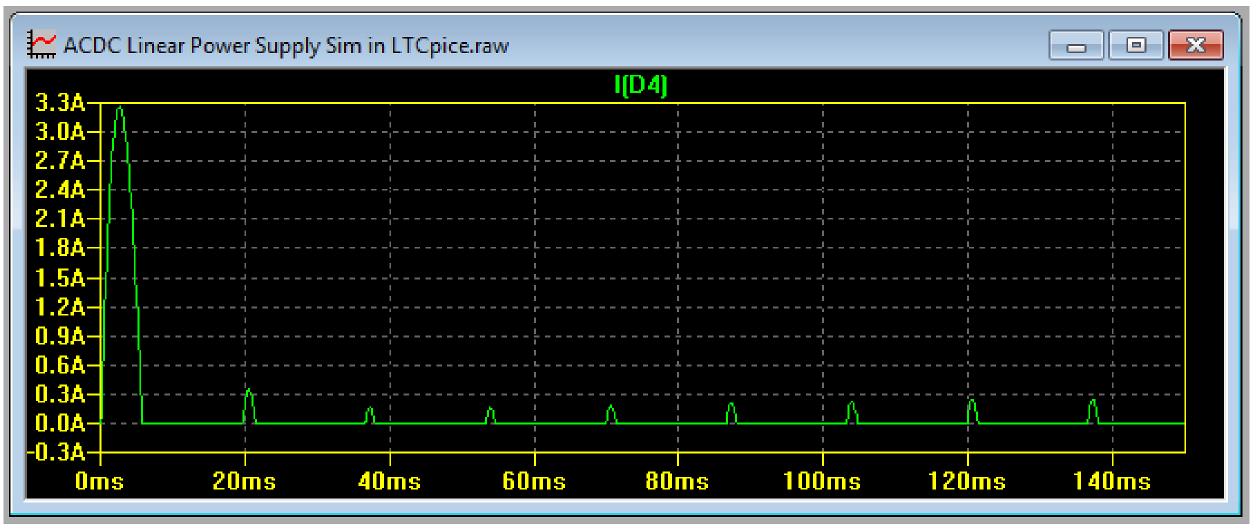
Current in D2 (ID2)



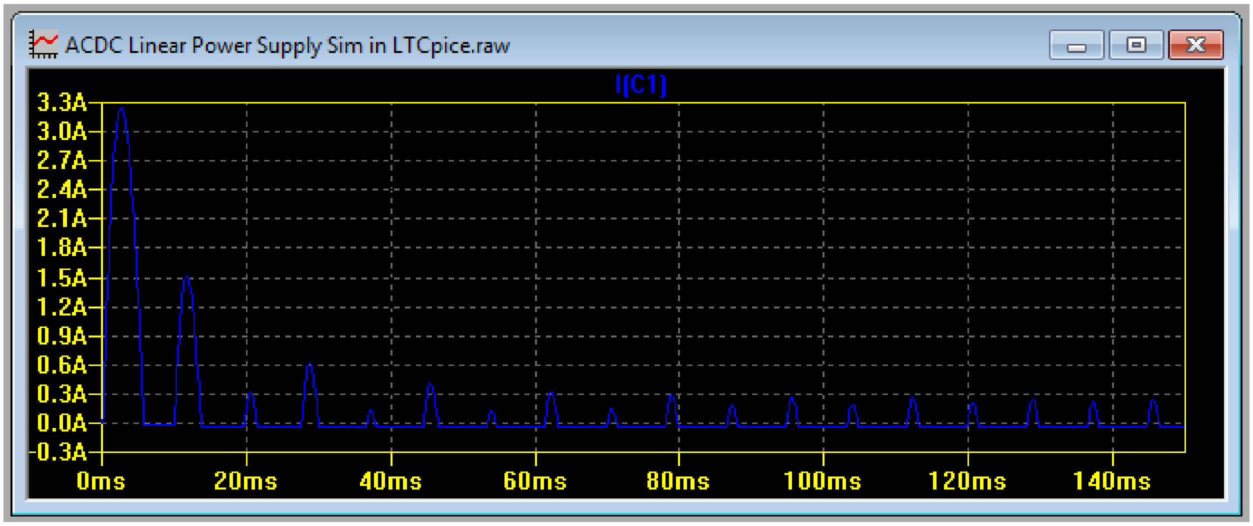
Current in D3 (ID3)



Current in D4 (ID4)



Current in C1 (IC1)

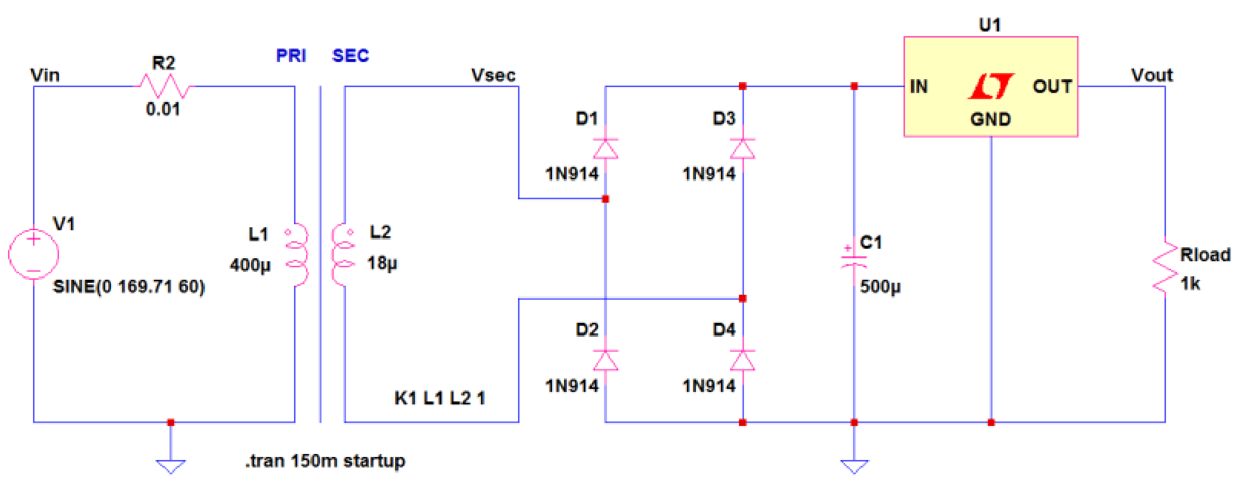


9. Adding a Linear Voltage Regulator on the output

   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.



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.