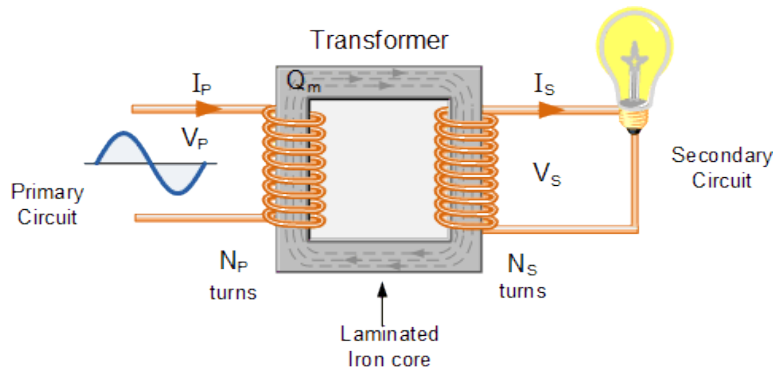


How to create a transformer using LTSpice

Transformers that are used here simply steps up or down AC voltages.



Primary Power = Secondary Power

$$V_p I_p \cos(\theta) = V_s I_s \cos(\theta)$$

$$\frac{V_p}{V_s} = \frac{N_p}{N_s} = \frac{I_s}{I_p}$$

The relation between turn ratio and inductor ratio can be written as;

$$\frac{V_p}{V_s} = \frac{N_p}{N_s} = \frac{I_s}{I_p} = \sqrt{\left(\frac{L_p}{L_s}\right)}$$

Example: Find the turn ration of a transformer if primary inductance is 100H and secondary inductance is 1 H.

Using the equation $\frac{N_p}{N_s} = \sqrt{\left(\frac{L_p}{L_s}\right)} = \sqrt{\left(\frac{100}{1}\right)} = 10$.This is a step down transformer.

Step down Voltage Transformer

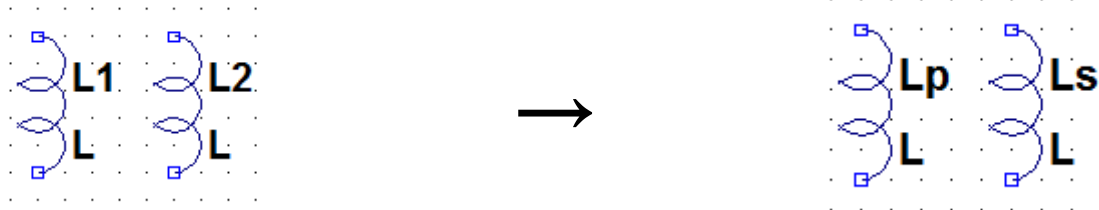
- Number of primary turns > Number of secondary turns
- Primary voltage > Secondary voltage

Step up Voltage Transformer

- Number of primary turns < Number of secondary turns
- Primary voltage < Secondary voltage

Exercise: We will design a step down transformer using LTSpice with 5:1 turn ratio. We can use $V_p=170V$, $f_p=60Hz$.

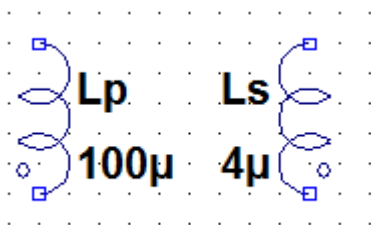
Step 1) You need two inductors. Change inductor names as Lp and Ls. Right click on L1 and change it to Lp and repeat same process for L2.



Step 2) Right click on Lp and L2s to open inductor properties window.

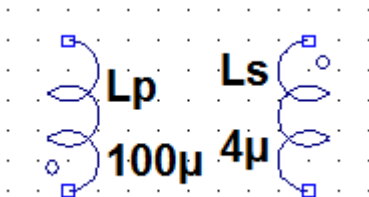
Two screenshots of the 'Inductor Properties' dialog boxes. The left window is for 'Inductor - Lp' and shows 'Inductance[H]: 100μ'. The right window is for 'Inductor - Ls' and shows 'Inductance[H]: 4μ'. Both windows have fields for 'Peak Current[A]', 'Series Resistance[Ω]', 'Parallel Resistance[Ω]', and 'Parallel Capacitance[F]'. A 'Show Phase Dot' checkbox is checked in both. To the right of the windows is a diagram of the two inductors, Lp (100μ) and Ls (4μ), with phase dots on the top terminal of each.

Step 3) This is an optional step. We will rotate Ls to get a better look for our transformer.



To get this you need to select Ls and rotate 90⁰ and then mirror it and then rotate it 90⁰ again.

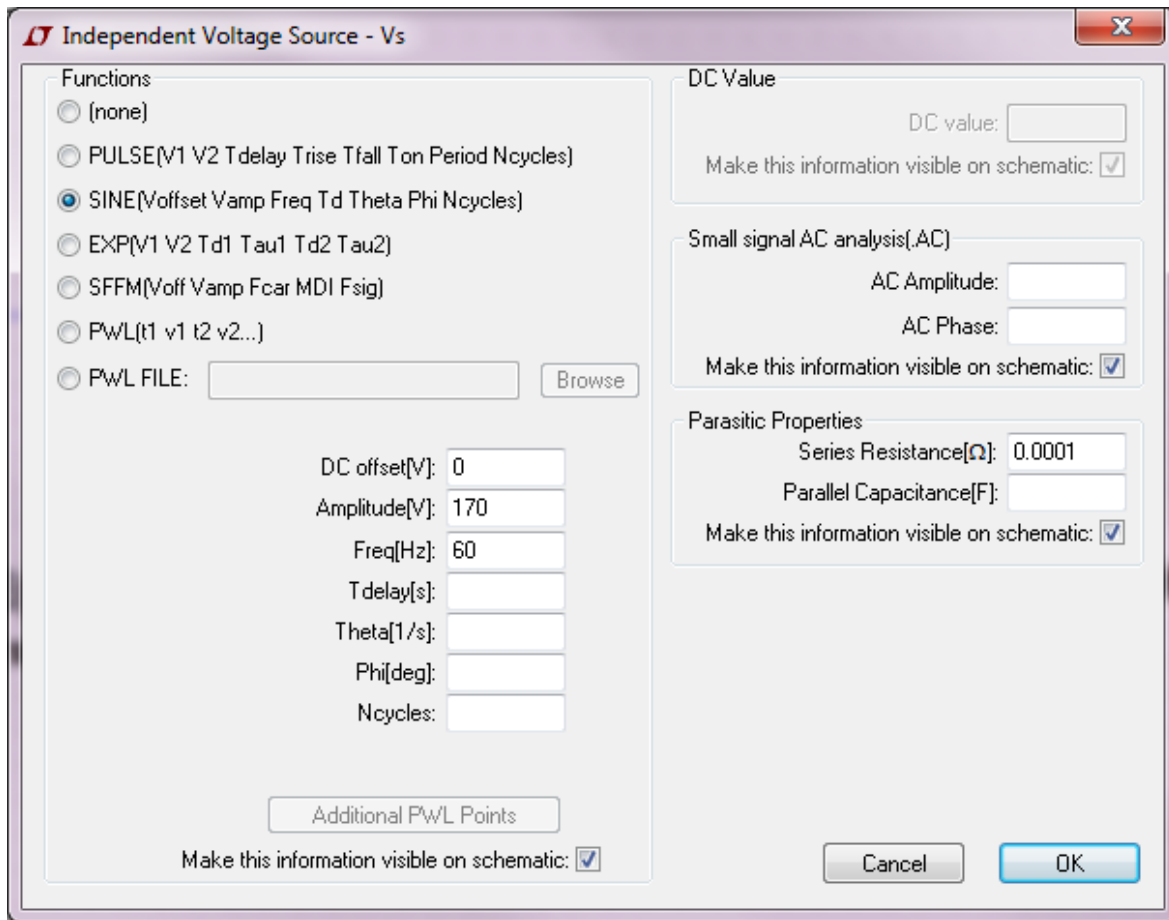
This will make sure input and output have same phase. Phase dots are on the same side for both Lp and Ls.



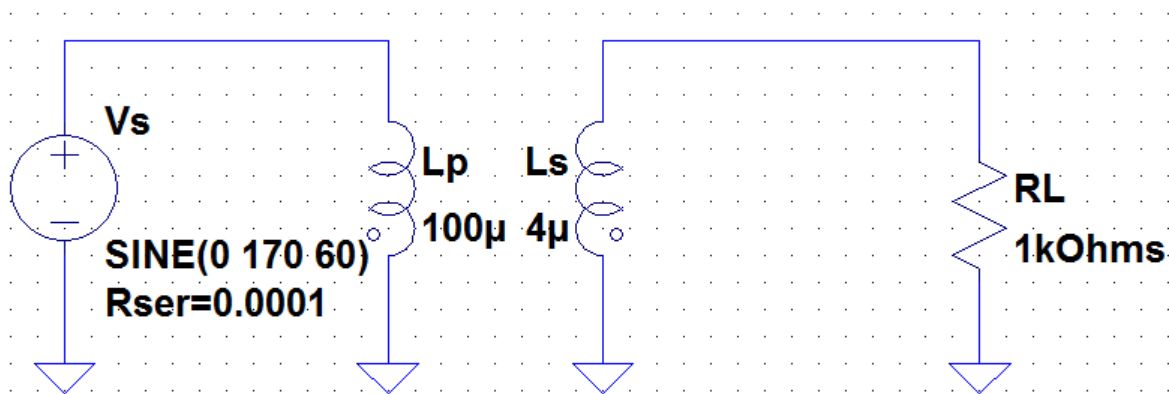
To get this you need to select Ls and rotate 180⁰.

This will make sure input and output have 180⁰ phase difference. Phase dots are not on the same side.

Step 3) Now we can connect Vs and RL to our circuit. Make sure Vs has following properties.

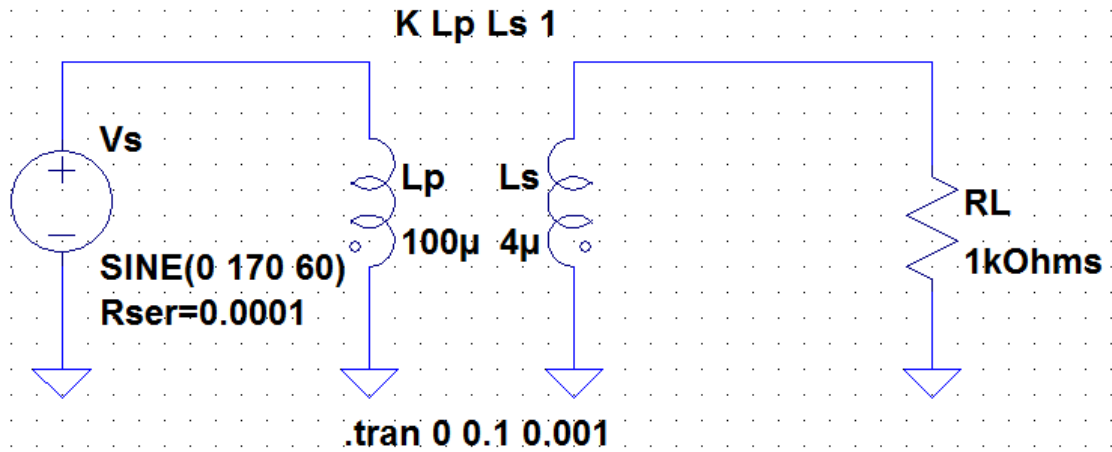


Make sure you add really small series resistance to source ($R_s=0.0001 \Omega$). After adding $R_L=1k\Omega$ load resistance you should have the following circuit.

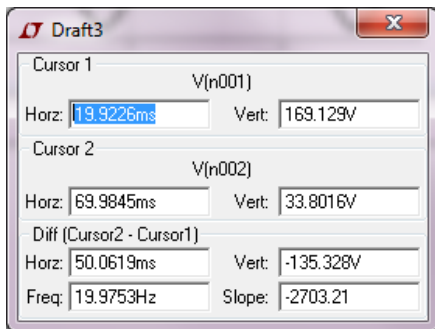
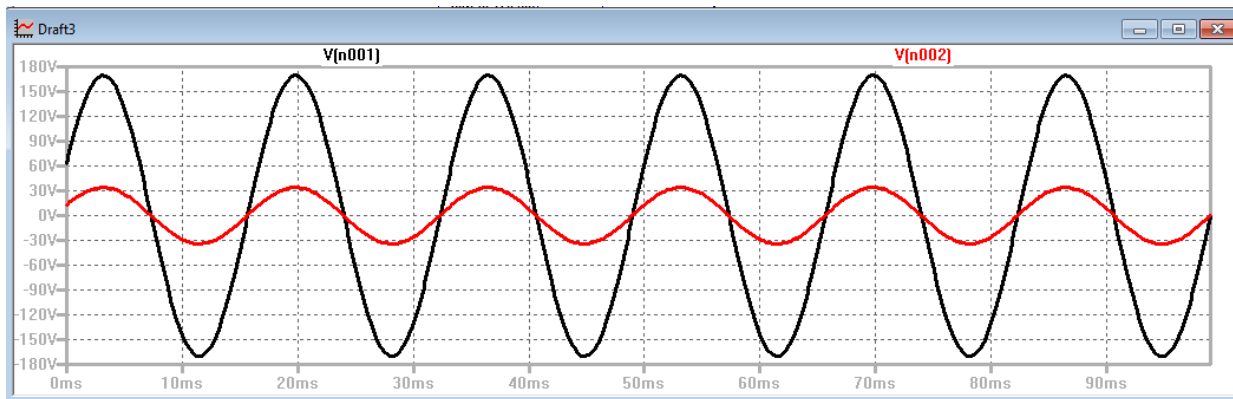


This transformer won't work properly because LTSpice does not know this is a transformer. This looks like two inductors are in the circuit. We need to tell LTSpice these are transformer. We will use a Spice directive to add a K-Statement ("K Lp Ls 1 ") to this circuit. Click on ^{op} and add "K Lp Ls 1 ". This will tell LTSpice that Lp is primary and Ls is secondary of the transformer.

The last number is called mutual coupling coefficient and can be between 0 and 1. 1 means perfect coupling (no leakage inductance) between the inductors. During simulations we will use 1 for this coefficient. We also need to add transient analysis to see the voltage levels at primary and secondary. The frequency is 60Hz and that means 1 cycle time (period) will be $1/60$. If we want to see 6 cycles we need to have $6/60 = 0.1$ second time for simulation. We will add Spice directive to our circuit.



After this we can simulate and see the Vp and Vs of the transformer.



After adding two cursors we can see $V_p = 169.123V$ and $V_s = 33.8016V$. This shows that turn ratio is 5:1.

Exercise: We will design a step down center tapped transformer using LTSpice with 20:1 turn ratio. We can use $V_p=170V$, $f_p=60Hz$. The center tapped transformer secondary will be divided into two (V_{s1} and V_{s2}) with same amplitude and 180° phase difference.

This time we will use 3 inductances. The steps are the same. Here is the circuit. Please pay attention to phase dots. This is a full wave rectifier circuit.

