

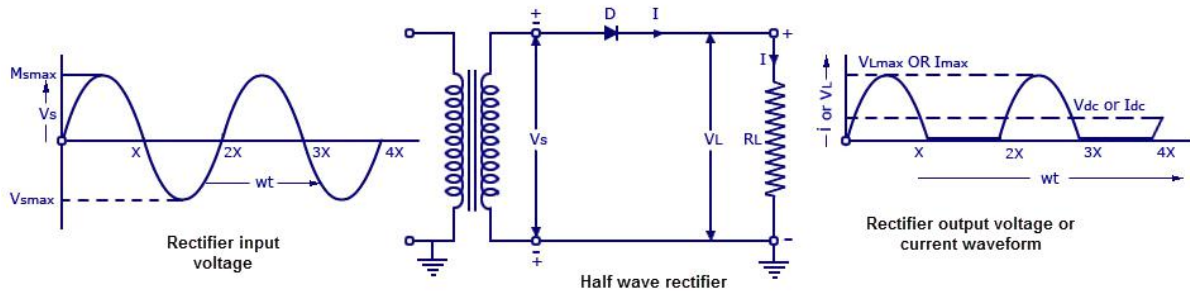
**Course code:** ETE 203/EEE 202

**Instructor:** Dr. Mirza Rasheduzzaman

**Course title:** Electronic Devices and Circuits 1 Lab

### Step by step guide to use LTspice for carrying any circuit simulation

Lets say, we want to construct a **Half Wave Rectifier** Circuit using LTspice following the arrangement below.

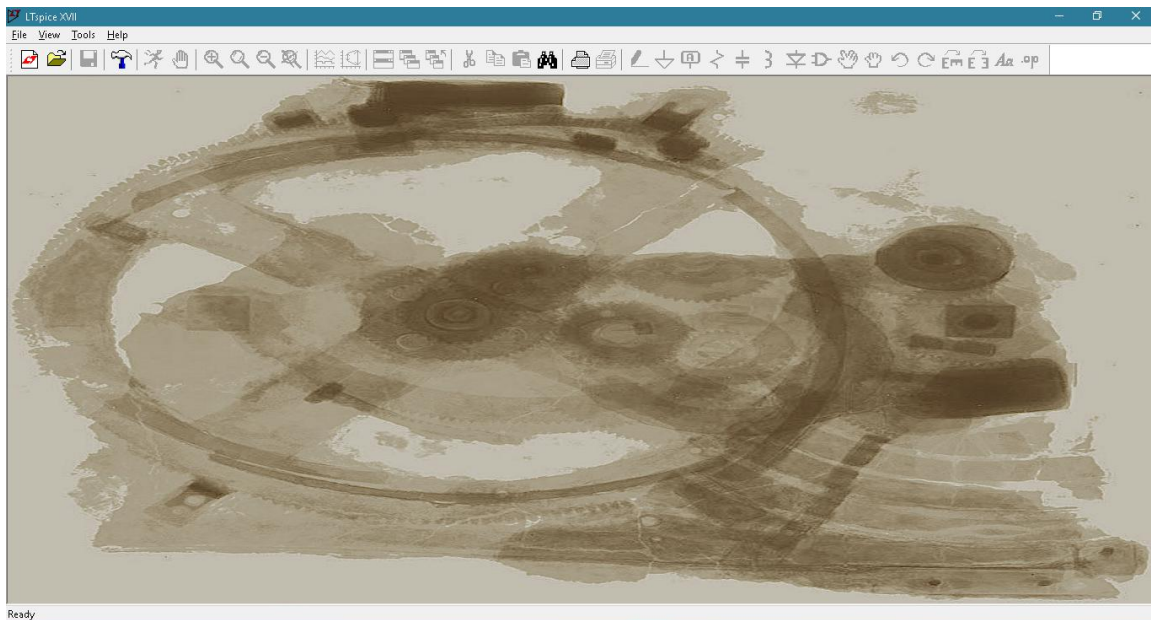


**Fig:** Half wave rectifier circuit

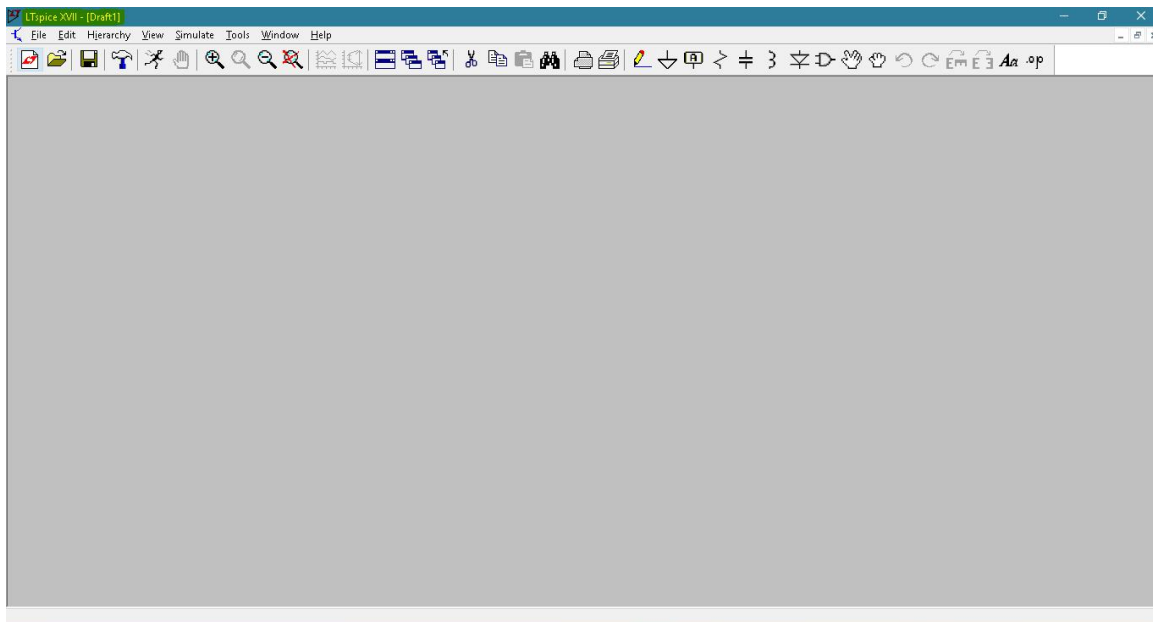
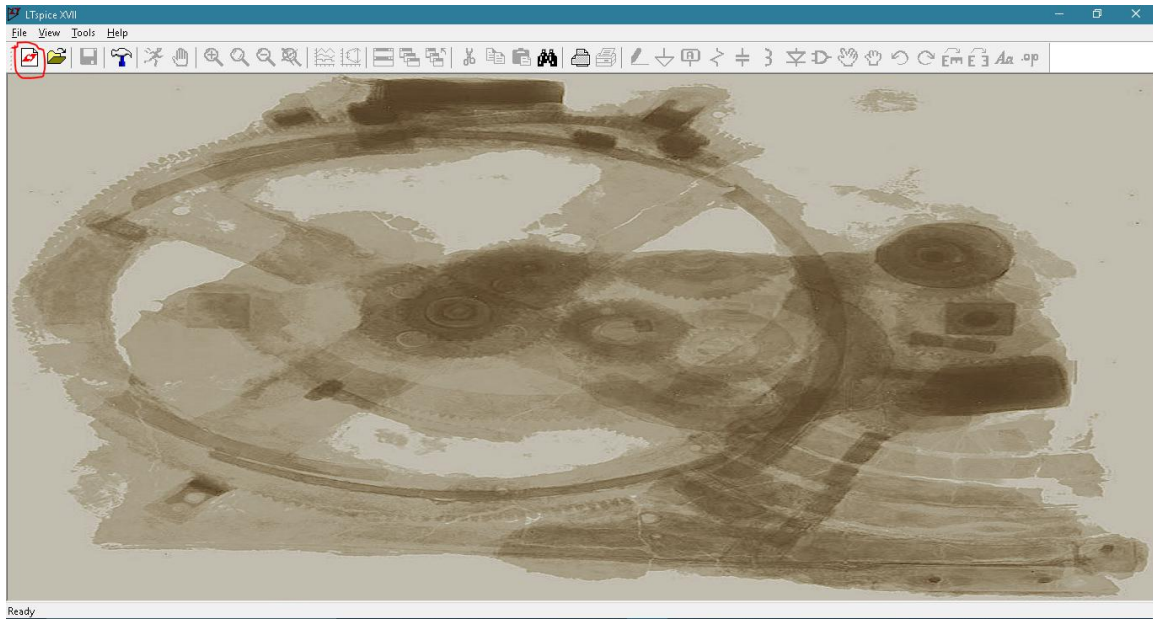
In this case, we have to follow the given steps:

This demonstration is being shown from LTspice XVII.

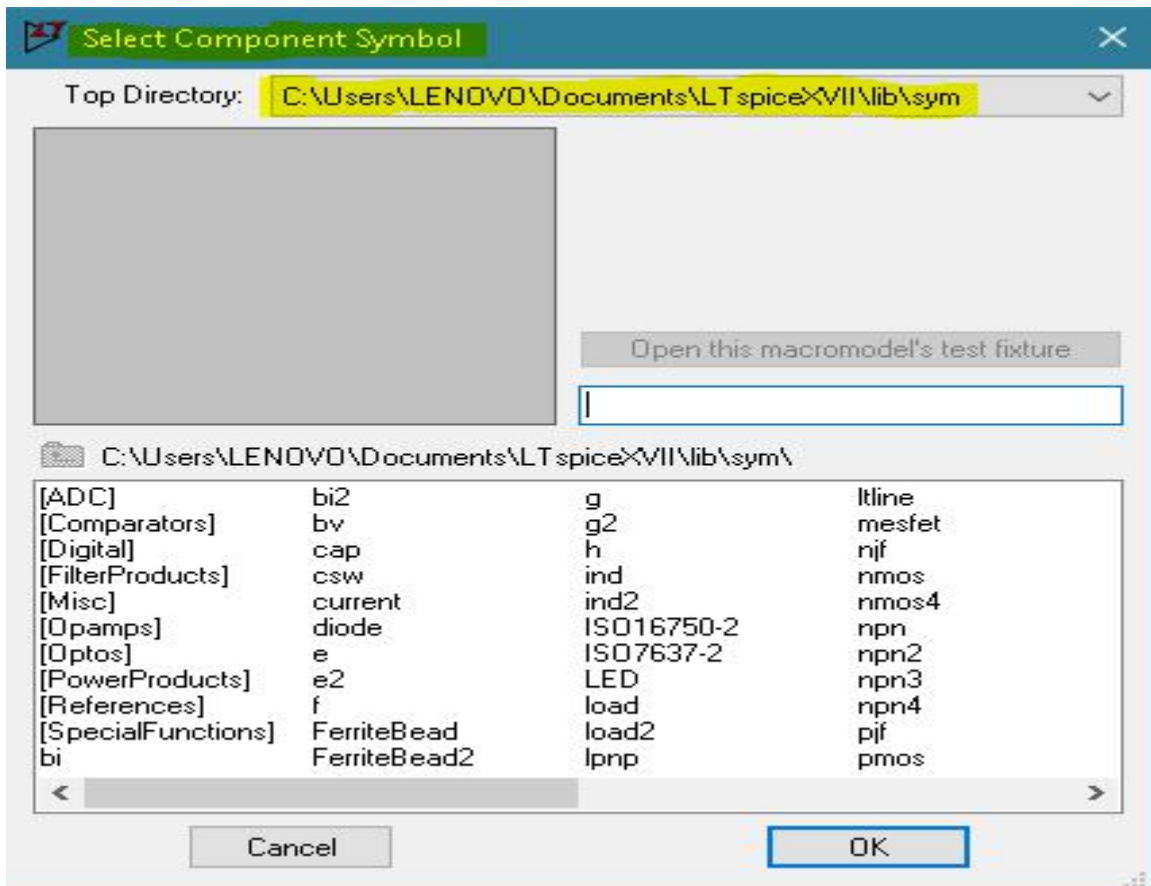
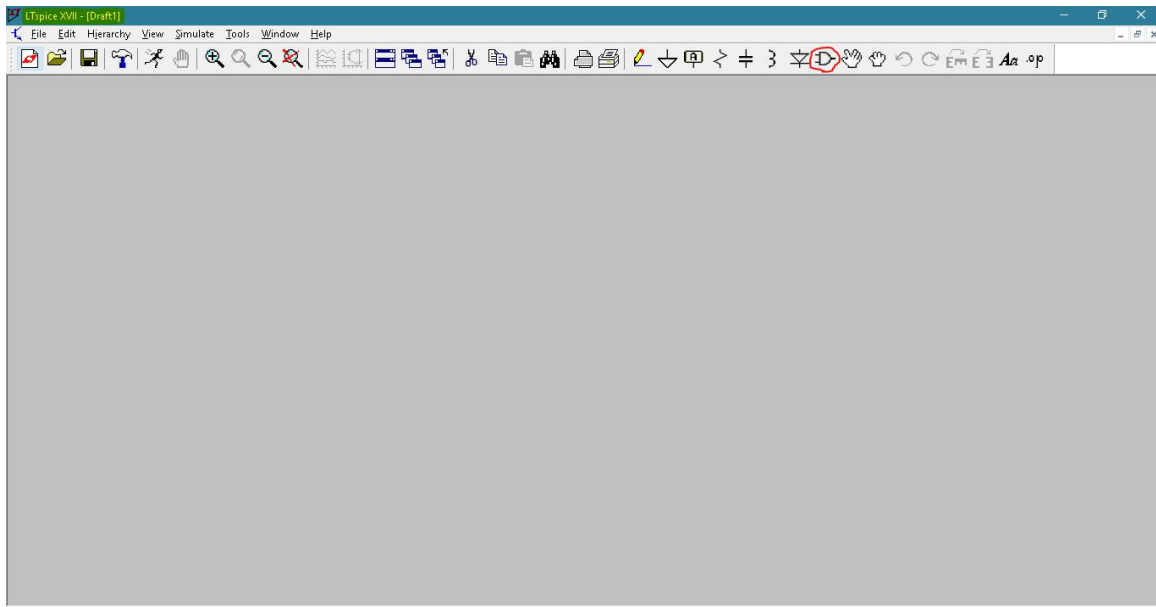
**Step 1:** Open LTspice XVII and you will see a Homepage.



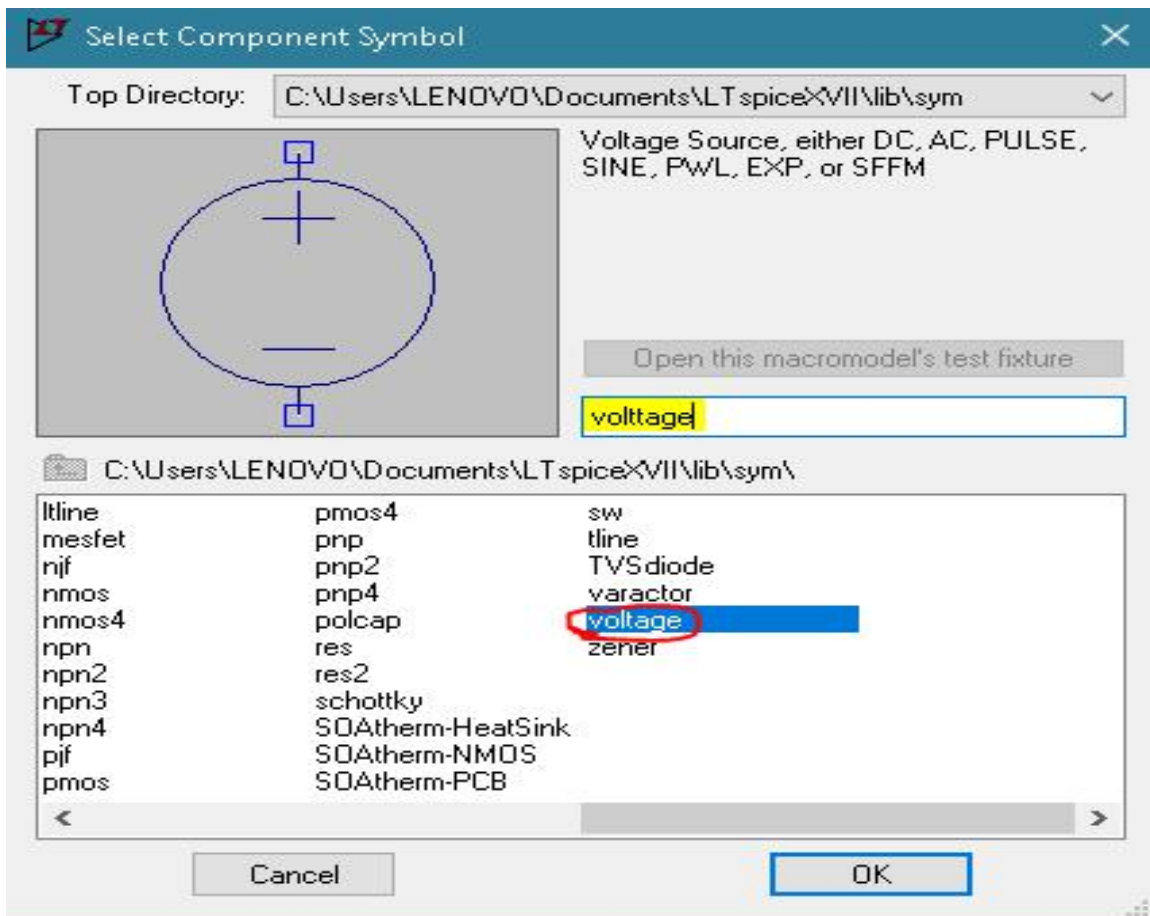
**Step 2:** Click on New Schematic as marked and you will get to see a Draft..



**Step 3:** Click on Component as marked and you will see a Select Component Symbol wizard, which contains the library folder in the Top Directory, where all the component symbols are saved as I have chosen them to be stored in that particular folder during installation.



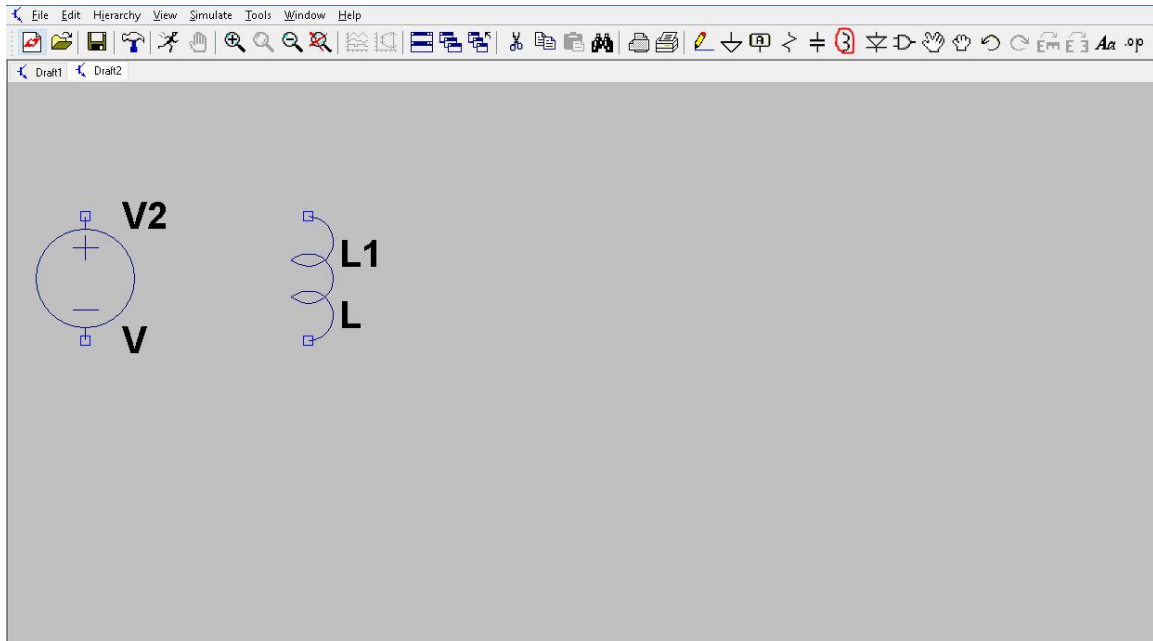
**Step 4:** Initially, we need an AC voltage source. Hence, type voltage in the search filter and you will find it and then click OK.



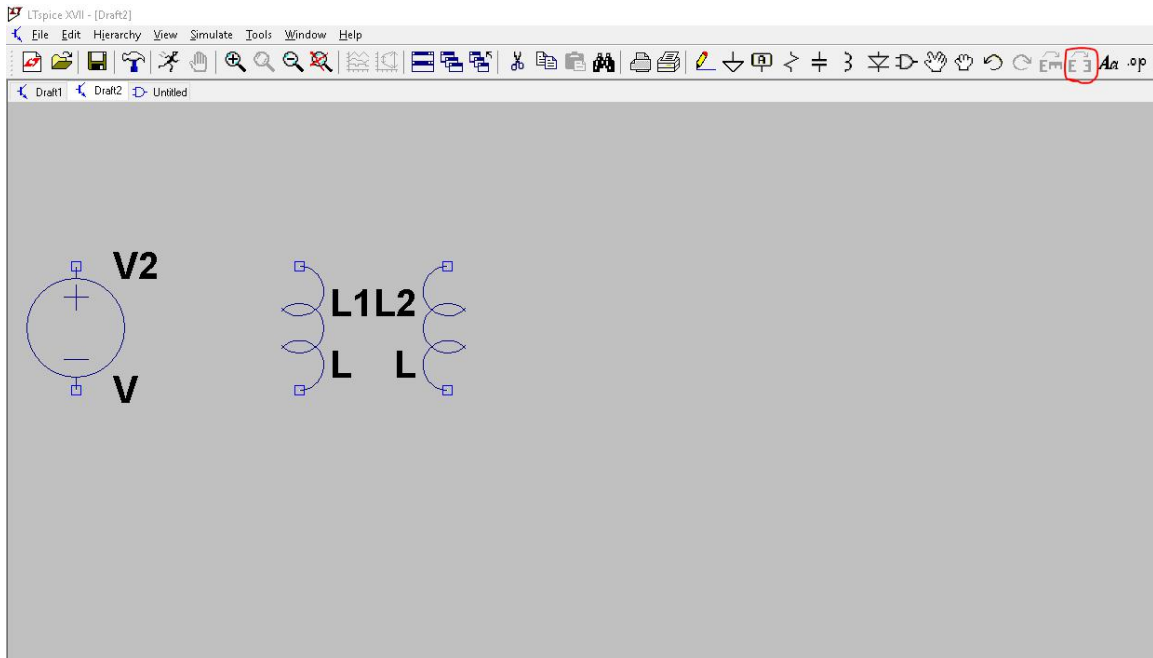
**Step 5:** Place the AC supply in an appropriate position and type escape (Esc).



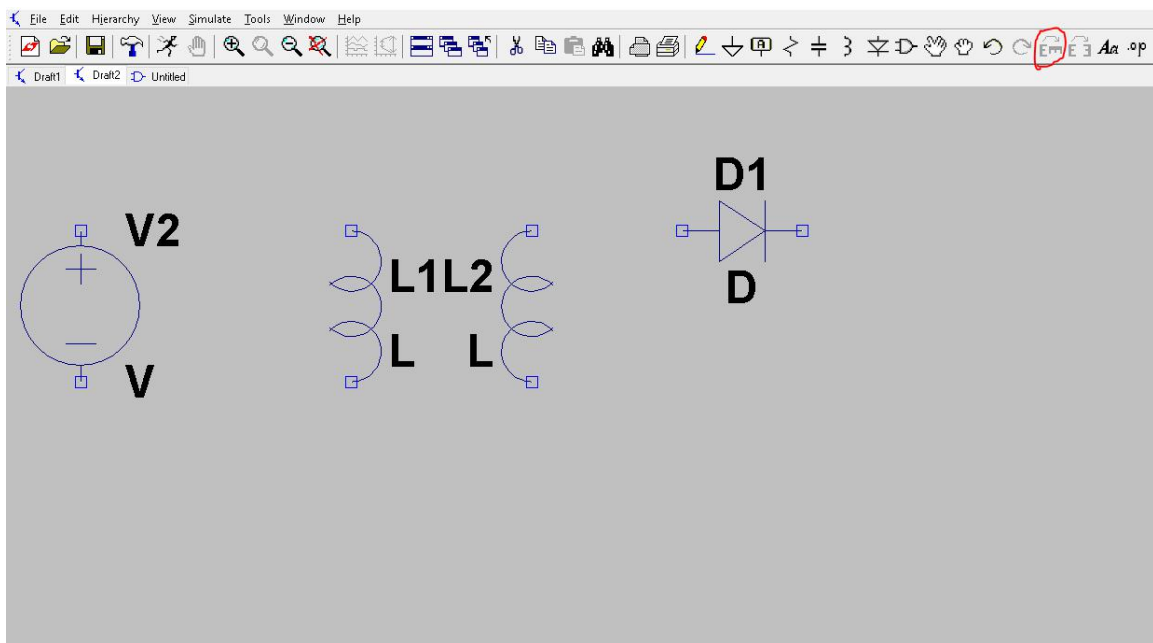
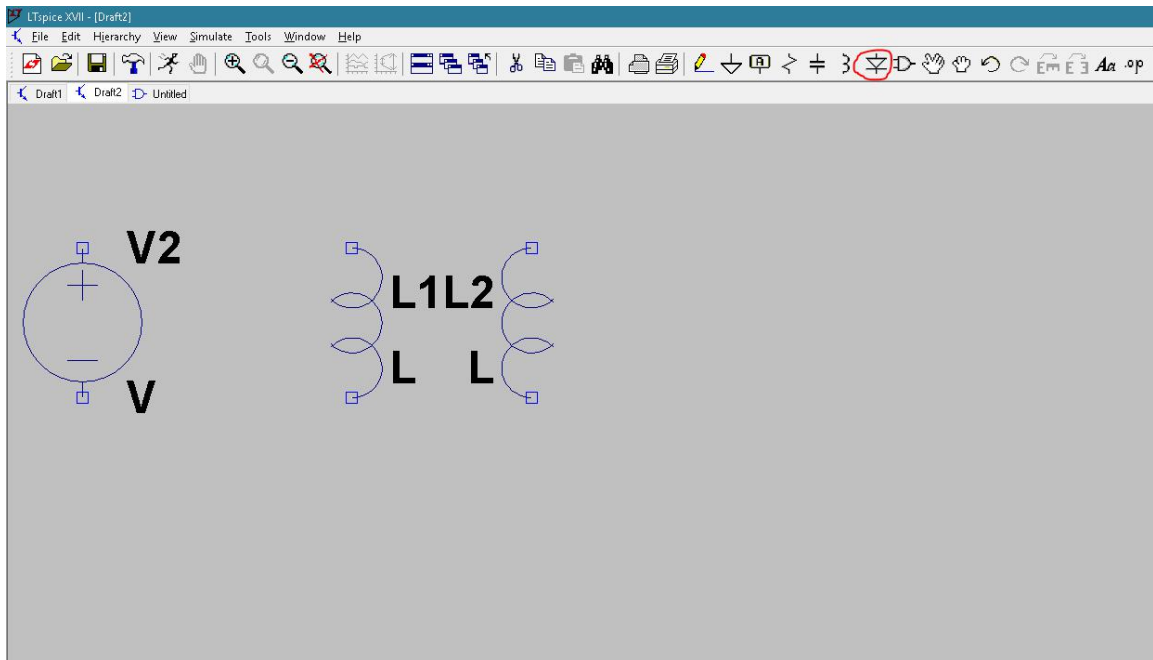
**Step 6:** Now, we need to make a transformer with two inductors placed in mirror position to each other. Click on Inductor as marked and adjust it in parallel to the AC supply.



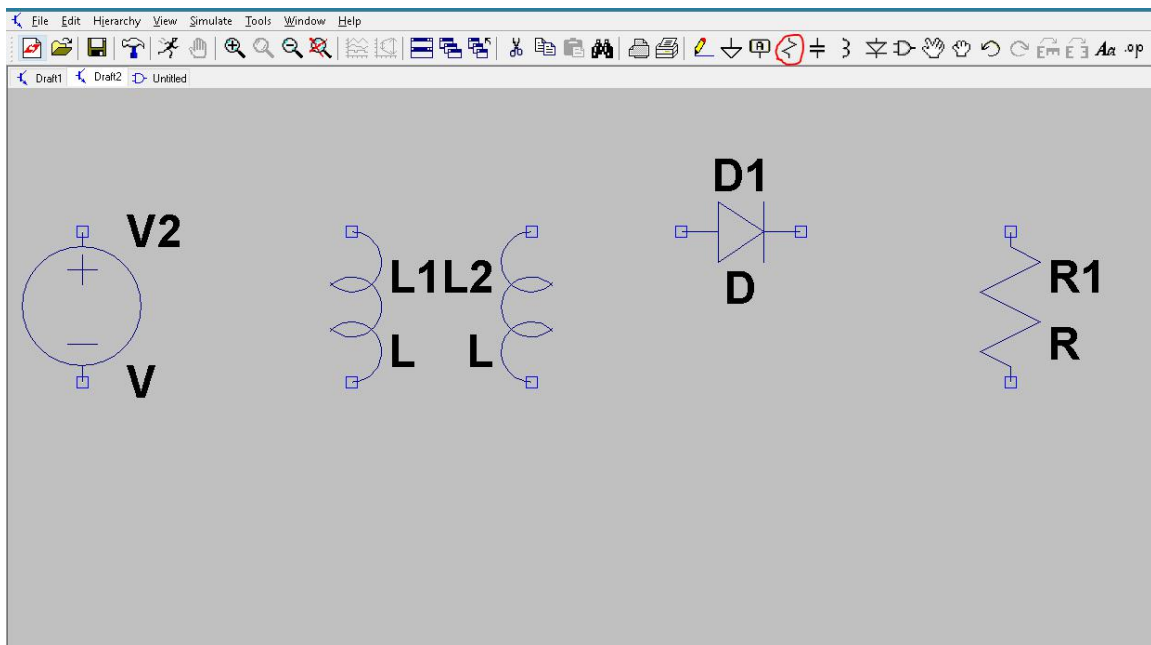
**Step 7:** Select another inductor and click on Mirror as marked and locate it in opposite direction to the first inductor.



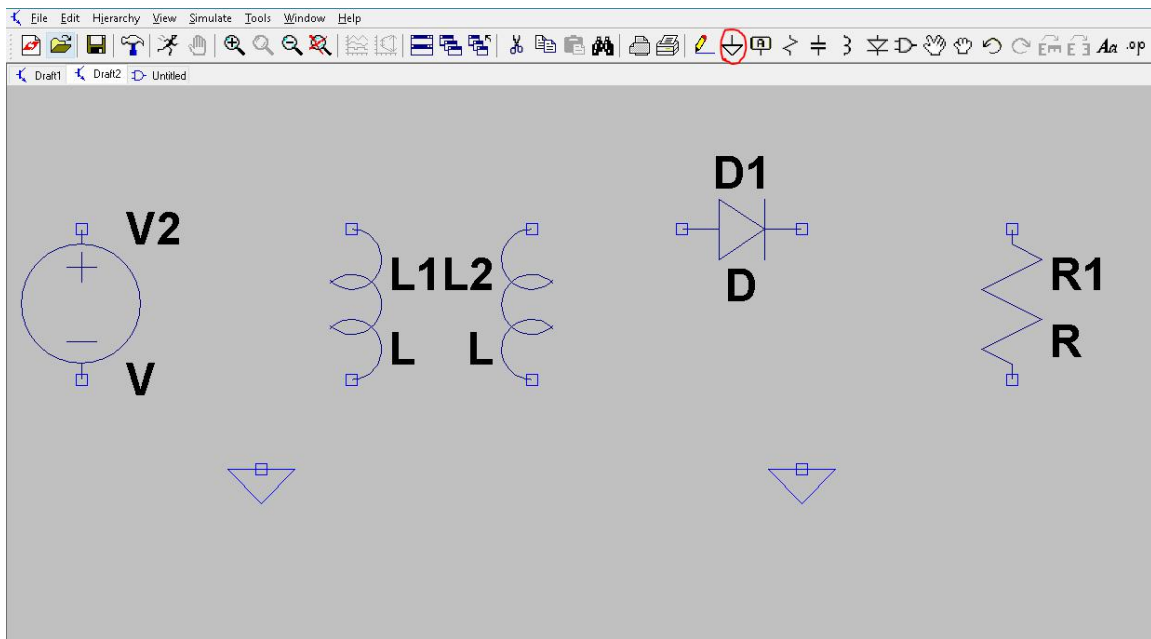
**Step 8:** Click on Diode as marked and select Rotate thrice as marked to set the diode in forward bias to the transformer.



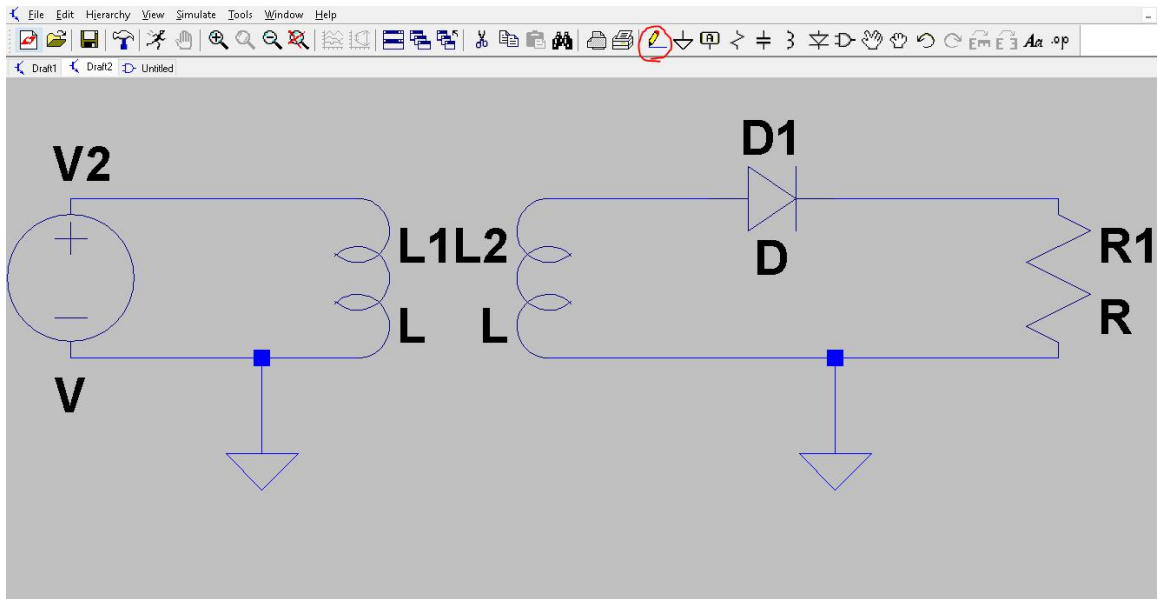
**Step 9:** Click on Resistor as marked and place it in series to the diode.



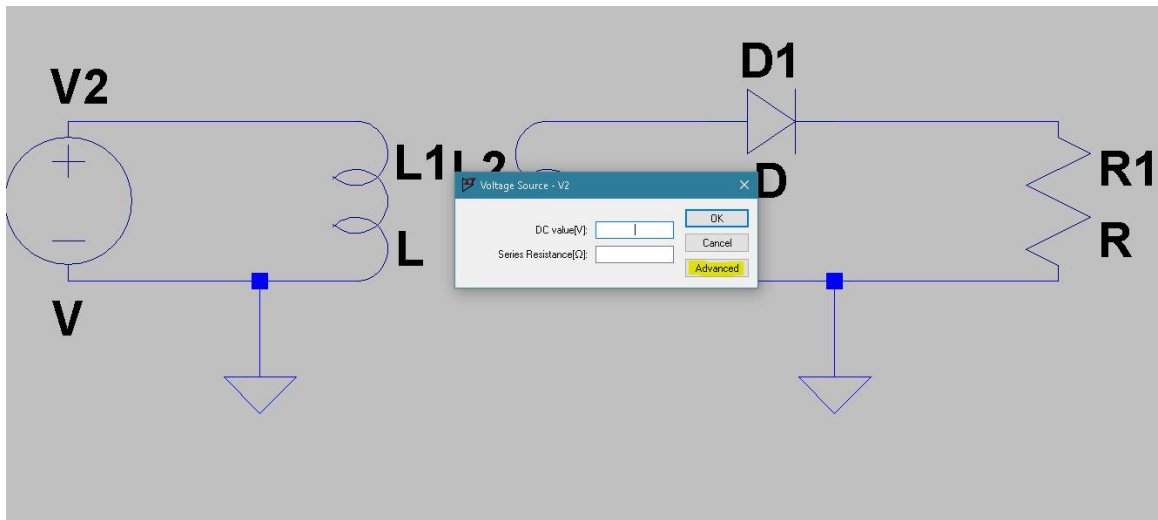
**Step 10:** Click on Ground as marked and put two of them across the AC supply and resistor in the following way.



**Step 11:** Click on Wire as marked and make the connections as per the circuit diagram.

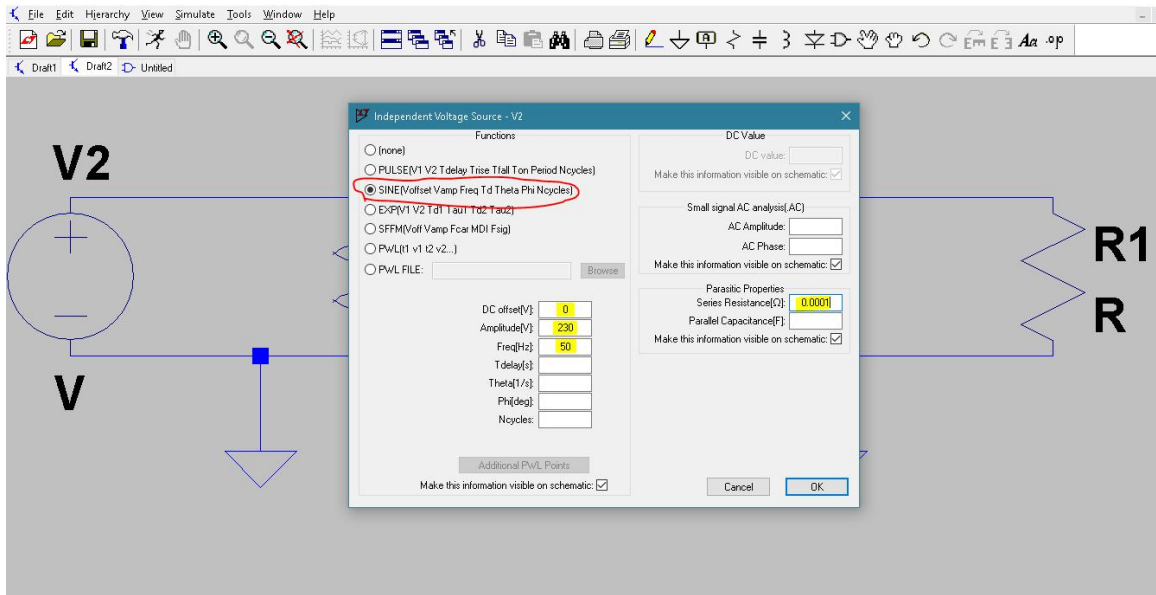


**Step 12:** Right click on the AC source and you will see a parameter tab. Click on Advanced.

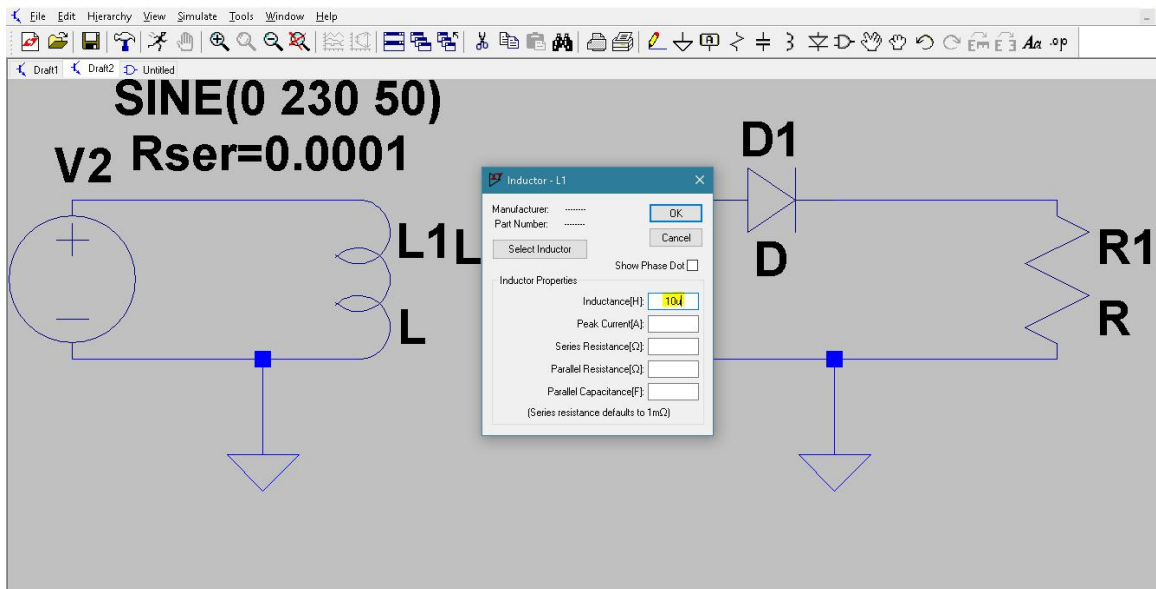


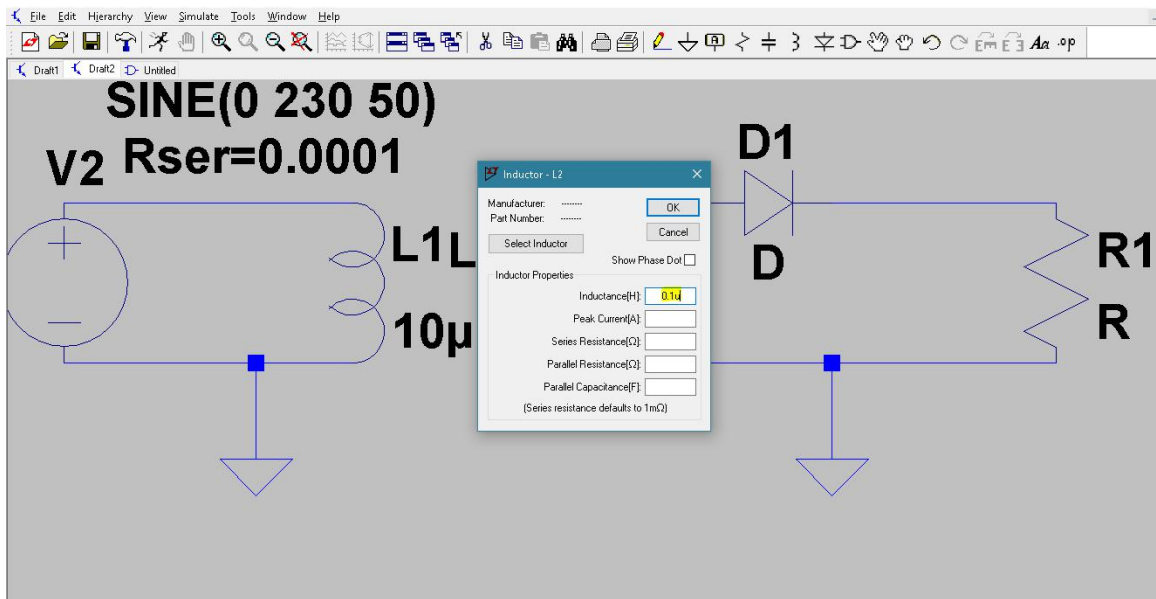
**Step 13:** Select SINE as marked and set the default parameters.



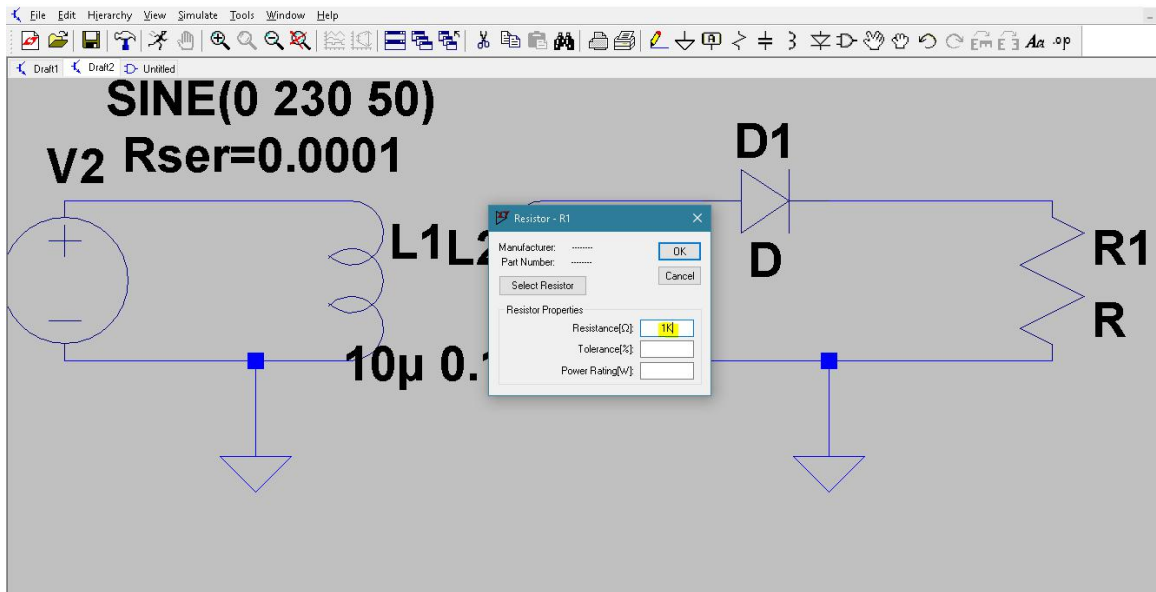


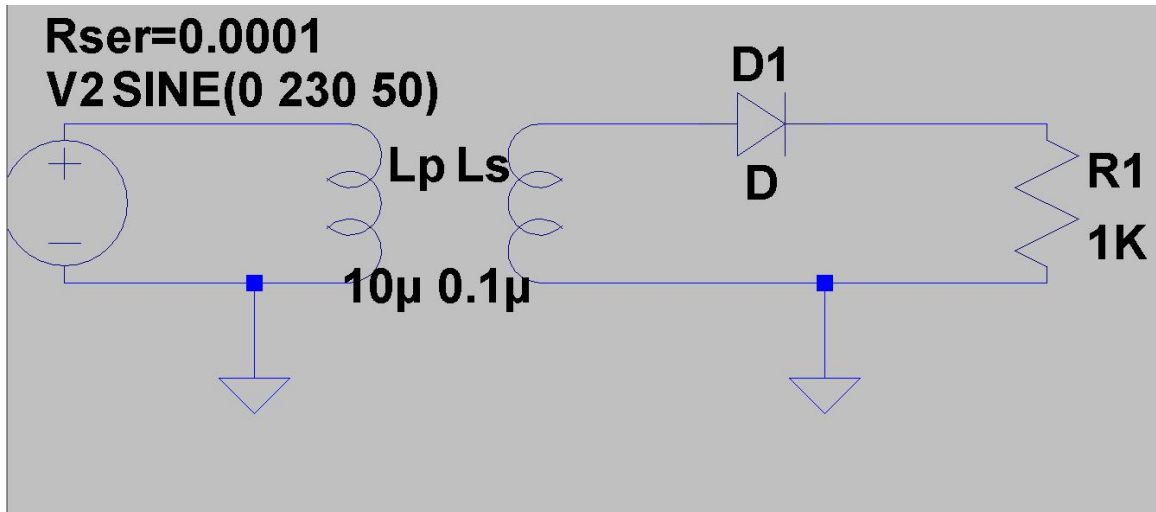
**Step 14:** Click OK and right click on each of the inductors and simultaneously set their default parameters.





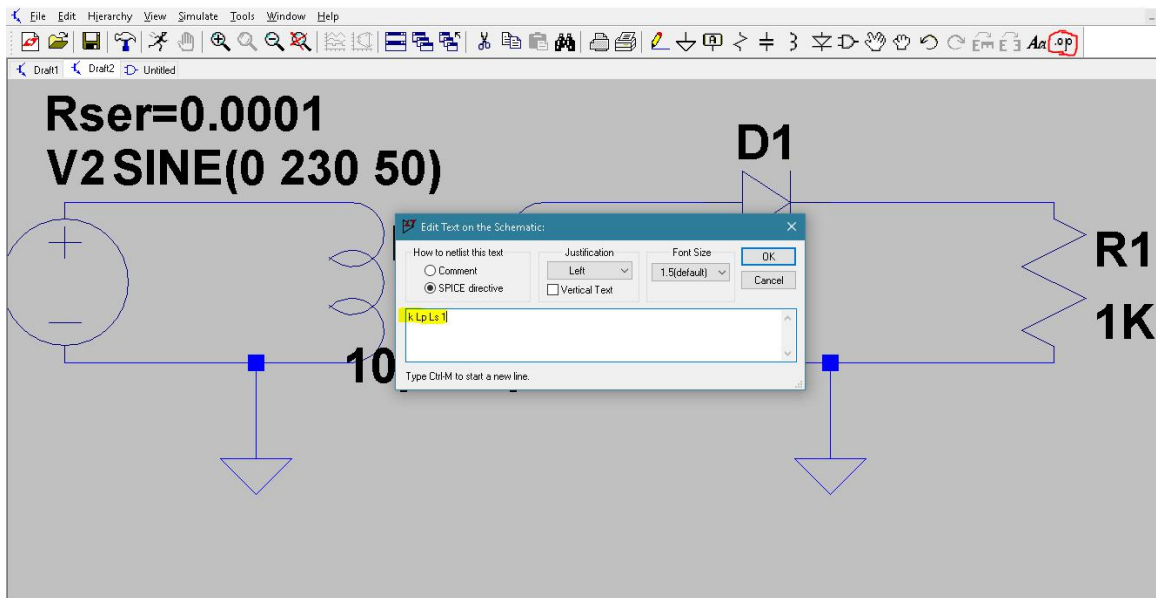
**Step 15:** Click OK and similarly, right click on resistor and set its default value and click OK.

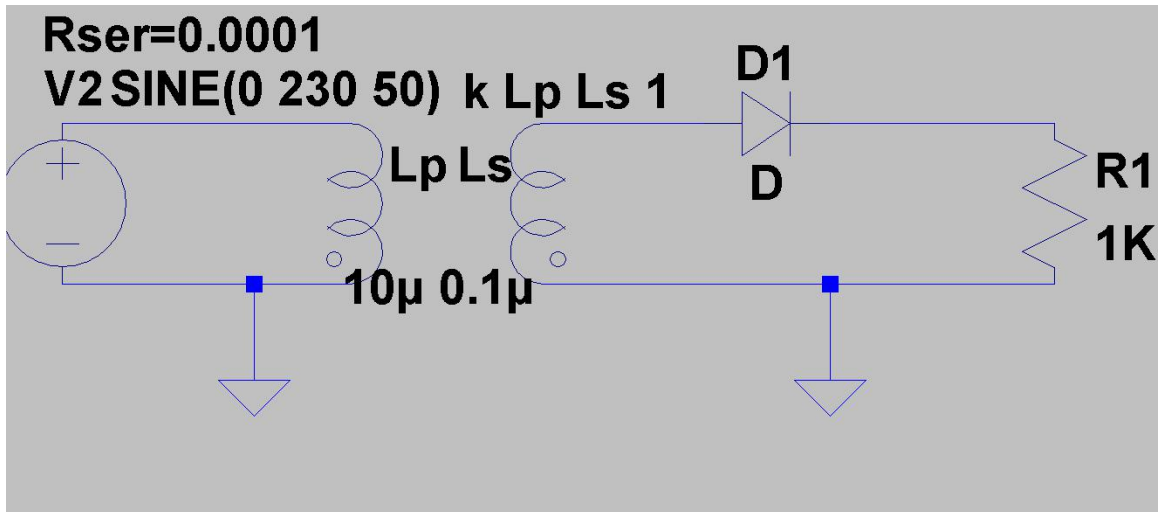




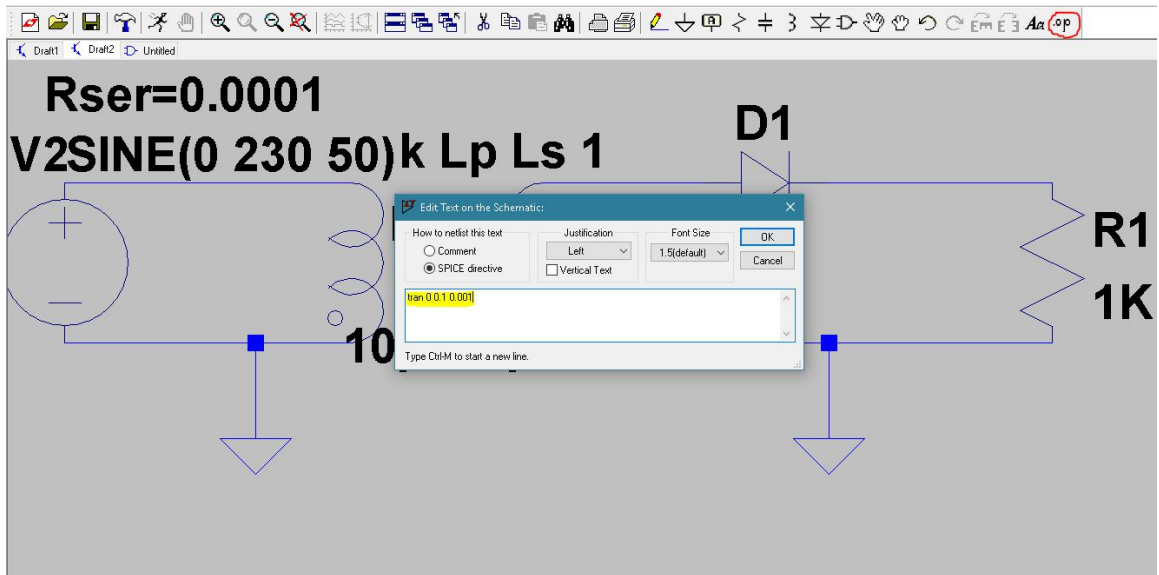
Now, the circuit is complete, but before carrying out the simulation, we need to focus on one thing, i.e. we have made the transformer with two inductors and LTspice does not consider it as a transformer rather considers it two pairs of inductors connected with two different circuits. Thus, we need to define the mutual inductance between them. This can be done by mentioning the coupling coefficient ( $k$ ) of transformer in the schematic using Spice directive in LTspice.

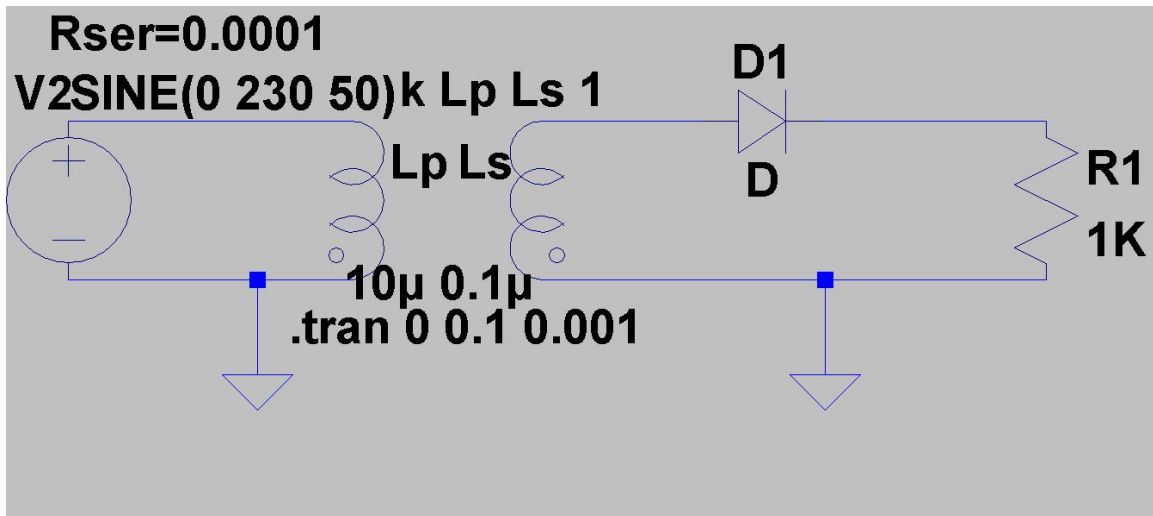
**Step 16:** We know,  $k$  ranges from 0 to 1. 0 means loosely coupled and 1 means tightly coupled. So, it's better to use  $k=1$  as tightly coupled transformer has no leakage. Hence to define it as such we need to click on SPICE Directive as marked and define the coupling coefficient as follows and click OK.



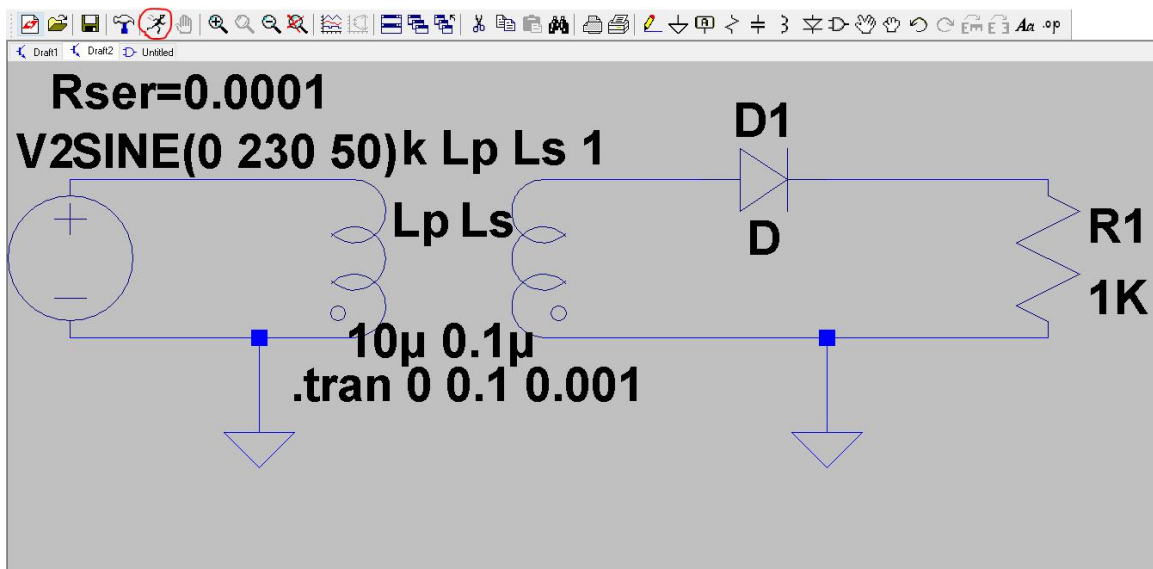


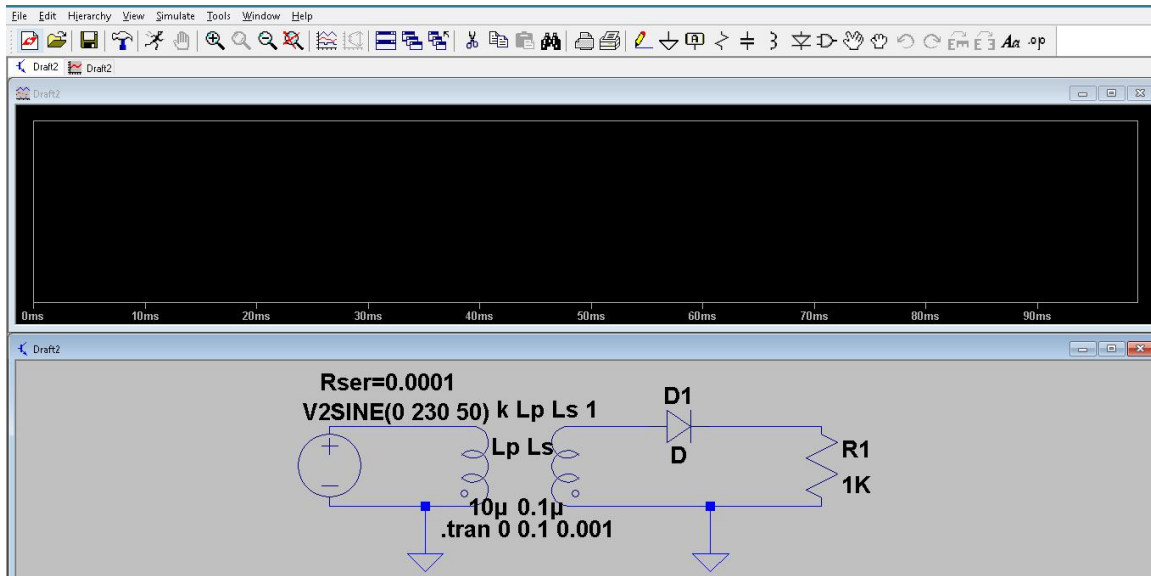
**Step 17:** Again go to the SPICE directive, set the mutual inductance in unit Henry (H) as follows and click OK.



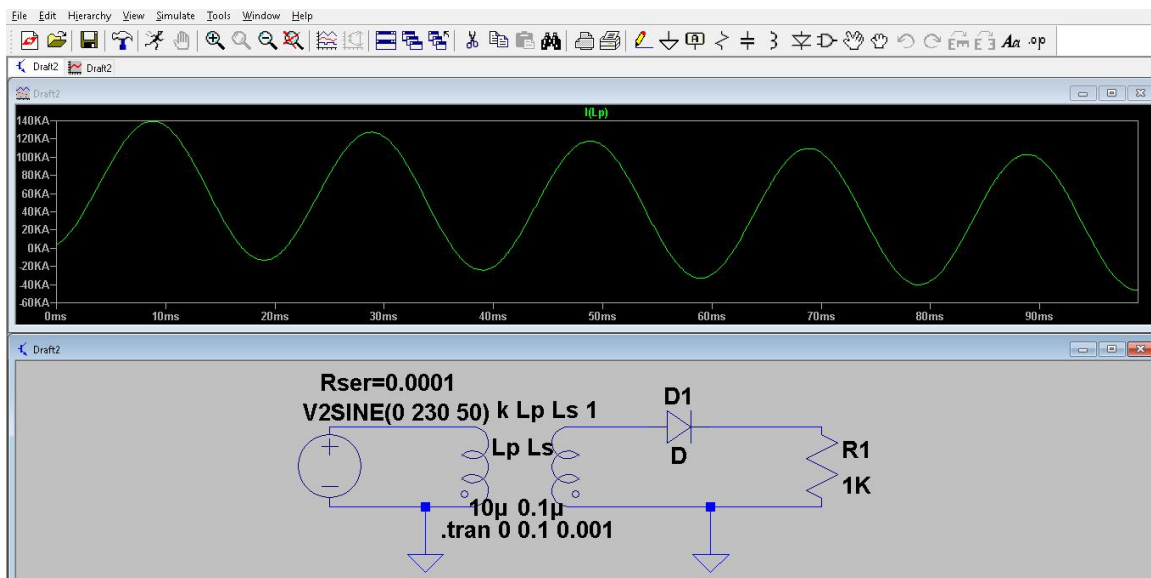


**Step 18:** Click on Run as marked and the output window appears.

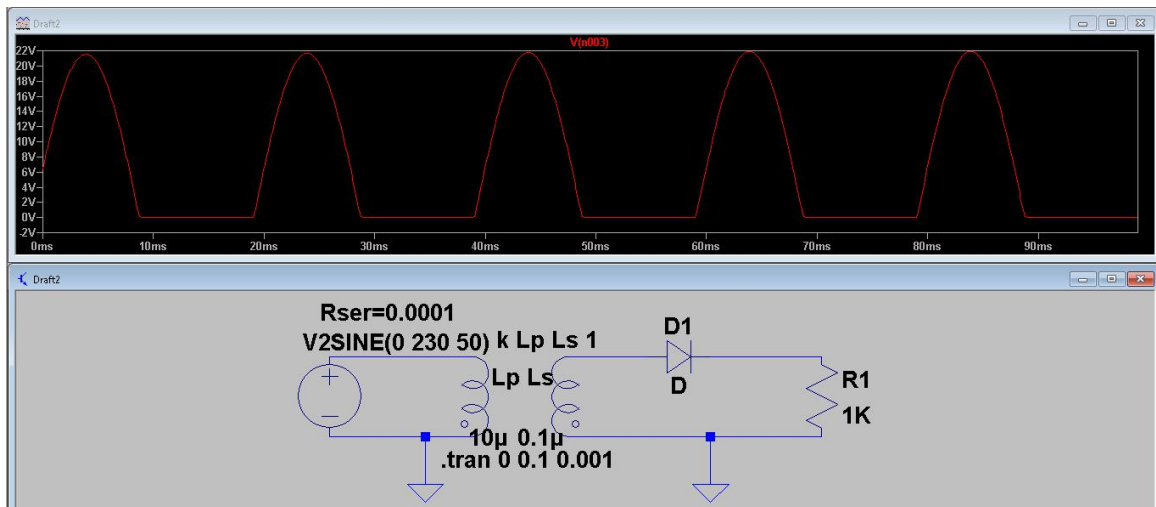
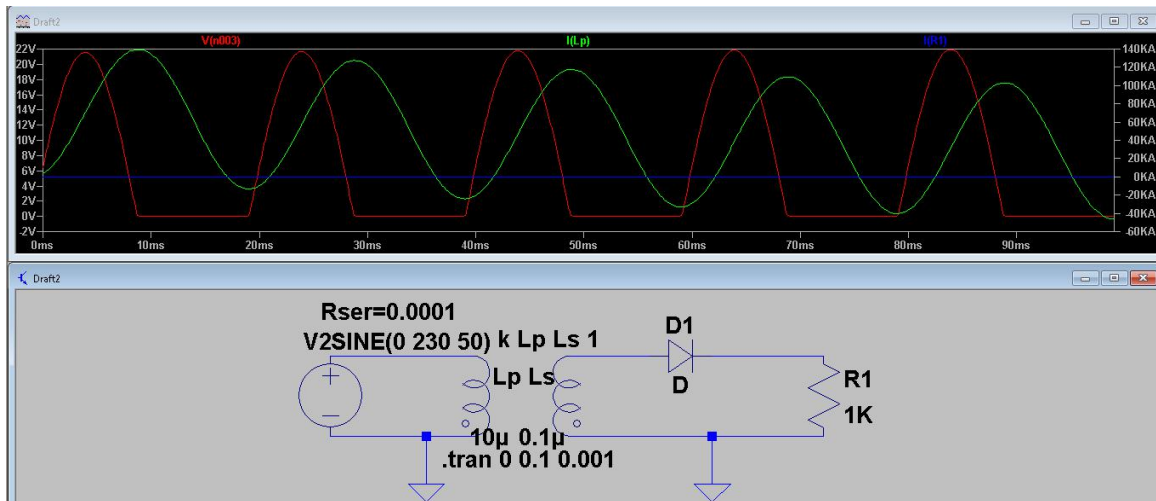




**Step 19:** Click on the transformer and you will see the input waveform.



**Step 20:** Click across the resistor and you will see the half rectified output waveform.



That's all!

Keep in mind that this is only an example and while carrying out other simulations, you might not have to follow those special cases like defining the coupling coefficient etc as transformer might not be required in other circuits. You can change the labels as you want by right clicking on them and adjust their position by clicking on Move.

Good Luck!

Prepared by:

*Ramisha Rabeya*  
*Teaching Assistant*  
*Electrical and Electronics Engineering*  
*University of Liberal Arts Bangladesh (ULAB)*

