The goal of this lab. is to design a power supply using Spice.

Pre-Lab

1. Make sure you know how to use and model Zener diodes (pp.24-28 of slide set "diodes' circuits").

2. Make sure you know how to build and analyze a regulated power supply (pp.51-71 of slide set "diodes' circuits").

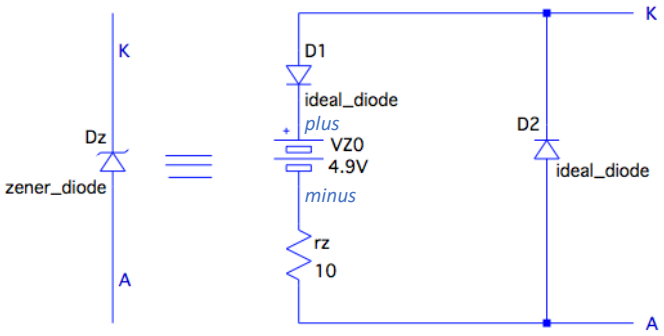
3. Before coming to Lab. complete the design of the power supply using hand calculation. The quiz assumes you have completed the design before lab).

4. Make sure you know how to define diodes and their associated models in LTSPICE

5. Make sure you know how to define and use sub circuits in LTSPICE. The easiest way to define a sub circuit is to type it in using the "Spice directive" option.

6. Unfortunately, the diode models provided do not adequately describe the operation of the diode in breakdown region. This is a serious problem to satisfactory model Zener diodes.
   Fortunately, you can employ the model shown in Figure 1 for the Zener, by defining the Zener as a sub circuit. The diodes D1 and D2 can be modeled as ideal diodes. An ideal diode can be implemented in SPICE by using a very small value of n (say n = 0.01).

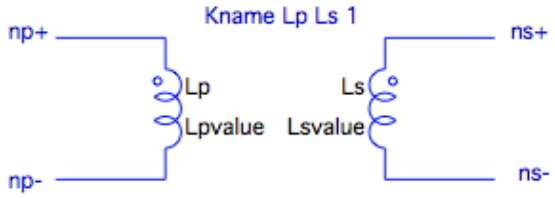
   ```
   .model ideal_diode D Is=100p n=0.01
   ```



7. Make sure you understand how to associate a sub circuit to a given LTSPICE schematic symbol.
   After instantiating a symbol, to associate it to a sub circuit right click on the symbol and a) change the prefix attribute to X, and b) use the name of the subcircuit as the SpiceModel attribute.

Attribute	Value	Vis.
Prefix	X	
InstName	Dz	X
SpiceModel	zener_diode	X
Value	zener_diode	
Value2		
SpiceLine		
SpiceLine2		

8. Make sure you know how to define and model transformers (mutual inductances) in LTSPICE



SPICE syntax for an ideal transformer:

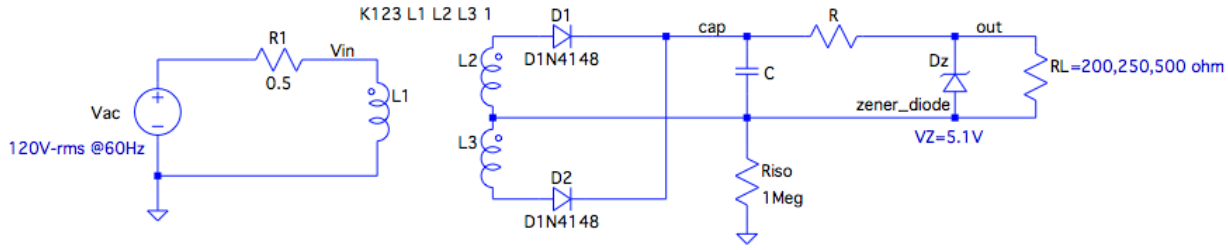
```
Lp np+ np- Lpvalue
Ls ns+ ns- Lsvalue
Kname Lp Ls 1
```

The turn ratio between the primary and secondary of an ideal transformer is given by:

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

Lab.

Design a regulated power supply using the circuit in Figure. The 1 MΩ resistor Riso is included to provide DC continuity and therefore "keep SPICE Happy": it has little effect on circuit operation.



It is required that the power supply provide a DC voltage of 5V nominal and can supply a load current as large as 25 mA (i.e. RL can be as low as 200Ω). Assume the availability of 5.1 V Zener diodes having rz=10Ω at Iz=20 mA (in other words VZ0=4.9V) and that the minimum current through the Zener should be at least 5mA.

Provide the following information:

1. Your "hand" calculations. Show in detail all your work.
2. Explain the reason behind any design choice.
    a. How did you select the rectifier diodes?
        i. Compute "analytically" the PIV and the max current through the rectifying diodes.
        ii. Compare your computations against SPICE.
    b. How did you select the design ratio between primary and secondary of the transformer?
    c. How did you select the value of the bias resistor R?
3. Use SPICE to verify the correctness of your design choices:
    a. Attach your LTSPICE deck
    b. Attach a plot of the output waveform for RL = 200, 250, 500 Ω
4. If the load gets shorted will the system get damaged? _____
   Explain why _____