Design of a Supply using LTSPICE – Pre-Lab

1. Read the handout "Notes on Using Zener diodes"
2. Before coming to lab. design the power supply using hand calculation (the Quiz assumes you have completed the design before lab.)
3. Read the handout "Using Transformers in LTSPICE"
4. Make sure you know how to define diodes and their associated models in LTSPICE.
5. Make sure you know how to define subcircuits in LTSPICE (read LTSPICE user manual pp. 95-96)
6. Make sure you understand how to associate a subcircuit to a given LTSPICE schematic symbol (lookup the link "LTSPICE tutorial at Laurier University" especially Method 3)
7. KEEN Related Activities

   * In your team report Illustrate all technical tradeoffs adopted (multidimentional problem solving) and the economic viability of your solution compared to available commercial solutions

Lab – Design of a Power Supply

It is required to design a regulated power supply using the circuit in Fig 3.52. The 100 MΩ resistor is included to provide DC continuity and thus “keep SPICE happy”; it has little effect on circuit operation.

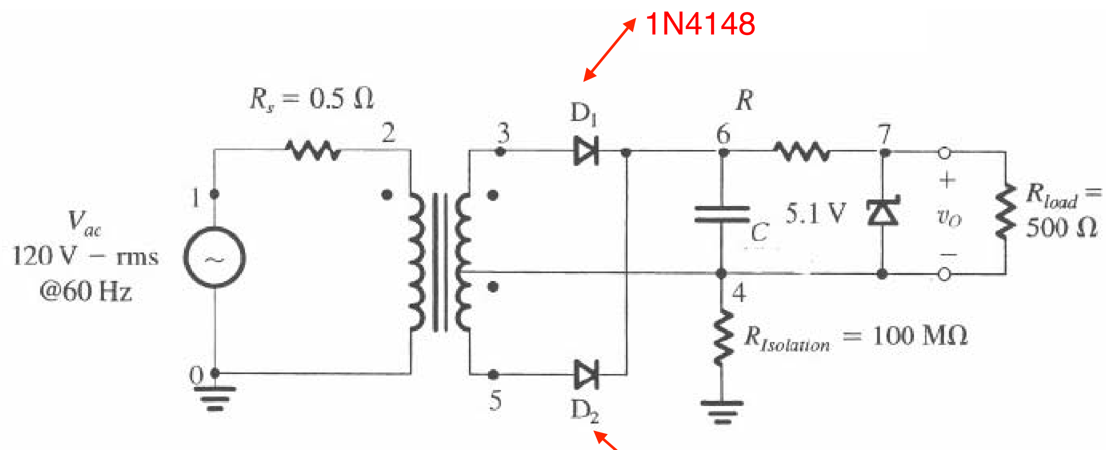


Fig. 3.52 A 5-V regulated power supply.

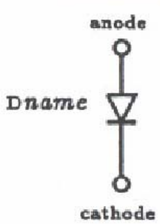
Let it be required that the power supply provide a dc voltage of nominally 5 V and be able to supply a load current as large as 25 mA (i.e., R_L can be as low as 200 Ω). Assume the availability of 5.1-V zener diodes having $r_z = 10\ \Omega$ at $I_Z = 20$ mA (and thus $V_{Z0} = 4.9$ V), and that the minimum current through the zener should be at least 5 mA.

Diodes

PN-Junction Diode

Element

Spice Description



D*name* anode cathode *diode_model_name* [*#_diodes_in_parallel*]

.MODEL *diode_model_name* D (*parameter_name* = *value* .....)

Figure Spice element description for the *pn* junction diode and the general form of the associated diode model statement.

Diode model parameters

Model parameters*	Description	Unit	Default
AF	flicker noise exponent		1.0
BV	reverse breakdown knee voltage	volt	infinite
CJO	zero-bias p-n capacitance	farad	0.0
EG	bandgap voltage (barrier height)	eV	1.11
FC	forward-bias depletion capacitance coefficient		0.5
IBVL	low-level reverse breakdown knee current	amp	0.0
IBV	reverse breakdown knee current	amp	1E-10
IKF	high-injection knee current	amp	infinite
IS	saturation current	amp	1E-14
ISR	recombination current parameter	amp	0.0
KF	flicker noise coefficient		0.0
M	p-n grading coefficient		0.5
N	emission coefficient		1.0
NBV	reverse breakdown ideality factor		1.0
NBVL	low-level reverse breakdown ideality factor		1.0
NR	emission coefficient for isr		2.0
RS	parasitic resistance	ohm	0.0
TBV1	bv temperature coefficient (linear)	$°C^{-1}$	0.0
TBV2	bv temperature coefficient (quadratic)	$°C^{-2}$	0.0
TIKF	ikf temperature coefficient (linear)	$°C^{-1}$	0.0
TRS1	rs temperature coefficient (linear)	$°C^{-1}$	0.0
TRS2	rs temperature coefficient (quadratic)	$°C^{-2}$	0.0
TT	transit time	sec	0.0
T_ABS	absolute temperature	°C	
T_MEASURED	measured temperature	°C	
T_REL_GLOBAL	relative to current temperature	°C	
T_REL_LOCAL	Relative to AKO model temperature	°C	
VJ	*p-n* potential	volt	1.0
XTI	IS temperature exponent		3.0

Unfortunately the diode model does not adequately describe the operation of the diode in the breakdown region; that is, it does not provide a satisfactory model for Zener diodes.

However, the SPICE user can employ the model shown in Fig. 3.51 for the zener by defining the zener as a subcircuit Here D1 is an ideal diode that can be implemented in SPICE by using a very small value for n (say $n = 0.01$), and D_2 is a regular diode model for the forward direction of the zener (for most applications the parameters of D_2 are of little consequence).

Hint: use an ideal diode for both D1 and D2

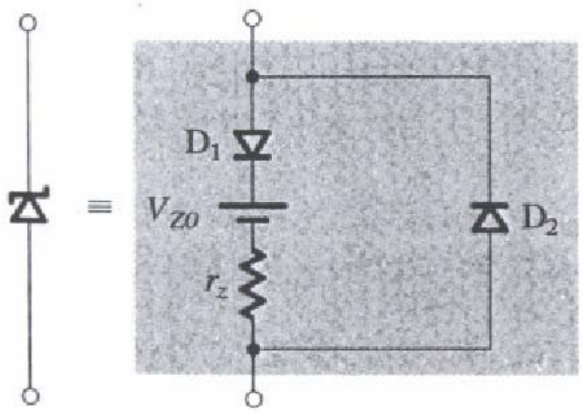
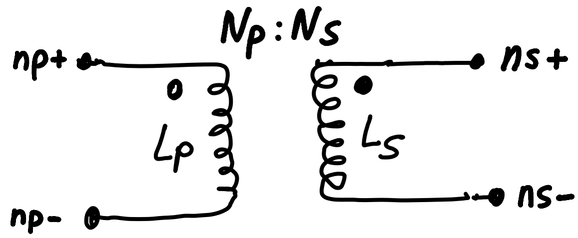


Fig. 3.51 Model for the zener diode. This model can be used in SPICE by defining the zener as a subcircuit. Diode D_1 is ideal and can be approximated in SPICE by using $n = 0.01$.

In the report: ← Worksheet

1. First design the power supply by hand (show in detail all your work) and then use SPICE to verify the correctness of your design choices.
2. As part of the design make sure to compute analytically the PIV and the max current through the full-wave rectifying diodes (verify the computed values against SPICE).
3. Explain the reasons behind any design choice,
4. Explain how did you select the rectifiers diodes, and the source of the SPICE model
5. Attach your spice deck,
6. Plot the output waveform.
7. If the load get shorted will the system get damaged? Explain why.

General Syntax of the SPICE statements used to describe a non ideal transformer:



$$\frac{N_p}{N_s} = \sqrt{\frac{L_p}{L_s}}$$

SPICE statements:

```
Lp  np+  np-  Lpvalue
Ls  ns+  ns-  Lsvalue
Kname  Lp  Ls  0.999
```