EEE210: Electronic Circuits and Devices

Lab #4: Simple Power Supply

Experimental Work: All oscilloscope readings are done with DC coupling of the channels unless otherwise noted.

1. Start *Multisim* and build the circuit of figure 1. Adjust the potentiometer to $R_L = 1$ kΩ (`R2`) and set its increment property to 2%. Adjust the `TMAX` simulation parameter to 0.3 ms using:

   `Simulate > Simulation Settings > Interactive Simulation Settings`

2. Observe `vL(t)`, `vi(t)` on the oscilloscope (still with $R_L = 1$ kΩ) with and without capacitor `C1`.

3. Measure $v_2(t)$, $v_o(t)$ on the oscilloscope with $R_L = 0\ \Omega$. Using the cursors, measure the minimum and maximum values of $v_2(t)$. You may now close the oscilloscope window as it will no longer be required.

4. While varying R_L from 1 kΩ to 0, measure:

   - average of $v_o(t)$ as indicated by the DC voltmeter,
   - average of $I_{470}(t)$ as indicated by the DC ammeter,
   - average of $I_Z(t)$ as indicated by the DC ammeter,
   - average of $I_{R_L}(t)$ as indicated by the DC ammeter.

   You may also observe that the current in the 470 Ω resistor remains constant as long as the Zener diode is in reverse conduction (this is the maximum available current in normal operation of the power supply); R_L takes the current that it needs and the *balance* is shunted through the Zener.
   Suggestion: Record the values in a spreadsheet.

Report:

1. Sketch a graph of the average of $v_o(t)$ versus the average of $I_{R_L}(t)$ and estimate from the graph the *nominal value* (smallest value) of the load resistor for this simple power supply.

2. Compare the nominal value of the load resistor to what is predicted by the theory (i.e. in course notes).

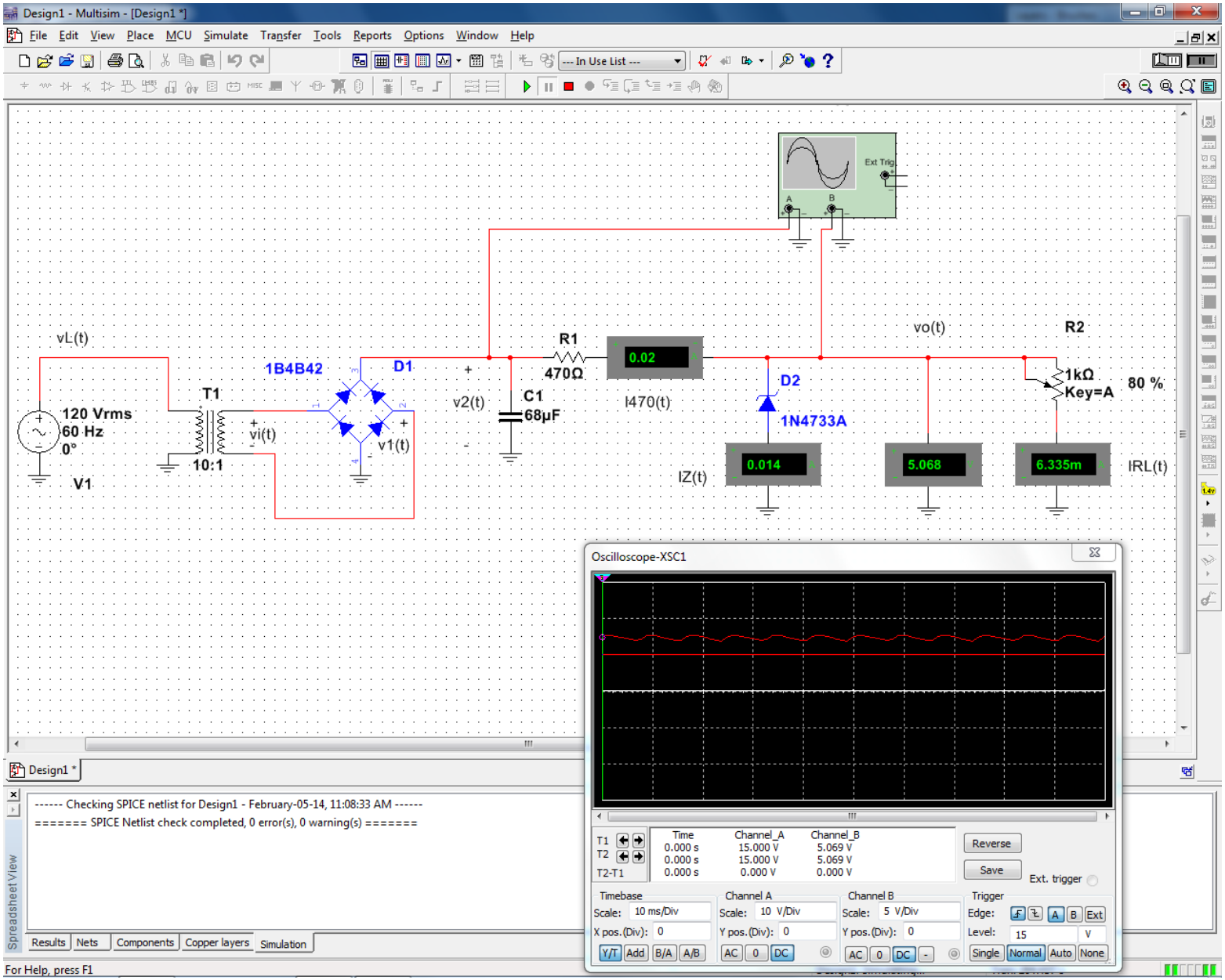
Design1 - Multisim - [Design1 *]
File Edit View Place MCU Simulate Transfer Tools Reports Options Window Help
vL(t)
120 Vrms
60 Hz
0°
V1
T1
10:1
vi(t)
1B4B42
D1
v1(t)
v2(t)
C1
68µF
R1
470Ω
0.02
I470(t)
D2
1N4733A
IZ(t)
0.014
vo(t)
5.068
R2
1kΩ
Key=A
80 %
6.335m
IRL(t)
Ext Trig
A
B
Oscilloscope-XSC1
Time Channel_A Channel_B
T1 0.000 s 15.000 V 5.069 V
T2 0.000 s 15.000 V 5.069 V
T2-T1 0.000 s 0.000 V 0.000 V
Reverse
Save
Ext. trigger
Timebase
Scale: 10 ms/Div
X pos.(Div): 0
Y/T Add B/A A/B
Channel A
Scale: 10 V/Div
Y pos.(Div): 0
AC 0 DC
Channel B
Scale: 5 V/Div
Y pos.(Div): 0
AC 0 DC
Trigger
Edge:
Level: 15 V
Single Normal Auto None
Design1 *
------ Checking SPICE netlist for Design1 - February-05-14, 11:08:33 AM ------
======= SPICE Netlist check completed, 0 error(s), 0 warning(s) =======
Spreadsheet View
Results Nets Components Copper layers Simulation
For Help, press F1

Figure 1: