ECEN 204

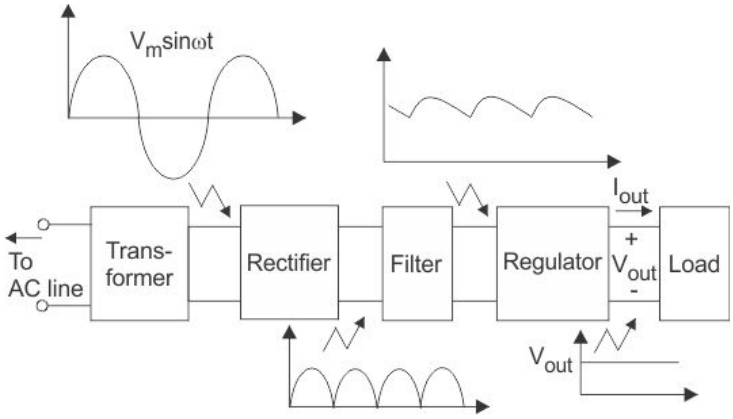
Design Report 1 : 5V Power Supply

Daniel Eisen : 300447549

September 16, 2019

1. Introduction and Background

The basis of this design of power supply is to accept an input AC voltage, in this case step down transformed to a lower voltage, and output a set regulated DC voltage (5V)



Components of typical linear power supply

Figure 1.

1.1 Design Blocks

As seen in figure 1, the supply is broken into separate blocks. First using a transformer high voltage AC (say mains) is stepped down to ~14V rms.

The next block is rectification, in this supply's case this is 'full wave. This take the oscillating AC input and 'flip' the negative voltages, outputting a pulse train of half sines. This is shown in figure 1, and practical measurements later in the report.

(note half bridge wave just throws away the negative pulses).

Next is filtering or smoothing. This smooths out the pulses, up towards the average DC value, achieved with a capacitor that contributed to a high RC value to increase discharge time to longer than pulse period.

Finally the smoothed input is input to a voltage regulator that ideally delivers a constant 5V from varied input voltages to varied loads.

1.2 Diode Rectification

Diodes basically only allow current flow in one direction, ie at an applied voltage polarity. To achieve half wave AC rectification simply with a single diode, as current is blocked on a negative voltage. To achieve full wave rectification a 4 diode configuration to provide a path of both positive and negative current while preserving a constant polarity at the output nodes, see fig 2.

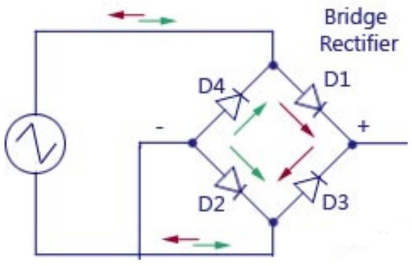


Figure 2.

1.3 Voltage Regulation

There are options available for regulation the output voltage to 5V.

A simply zener setup in reverse controlled breakdown but this very restrictive in the variance of input voltages and output loads.

So the second is the use of a pre-existing discrete IC package that works for larger load variance, though note it has a minimum required operating voltage input.

1.4 Design Requirements

- Supply is required to accept 12V 400mA AC from a plug pack transformer
- Rectify input with general purpose diodes
- Smoothing to constant DC
- DC regulated to a stable 5V

2. Design Description

For both the bread board construction and the pcb circuit an initial design was used and then after construction and testing revised and finalised. Therefore the designs in this section do not necessarily reflect the final construction of either.

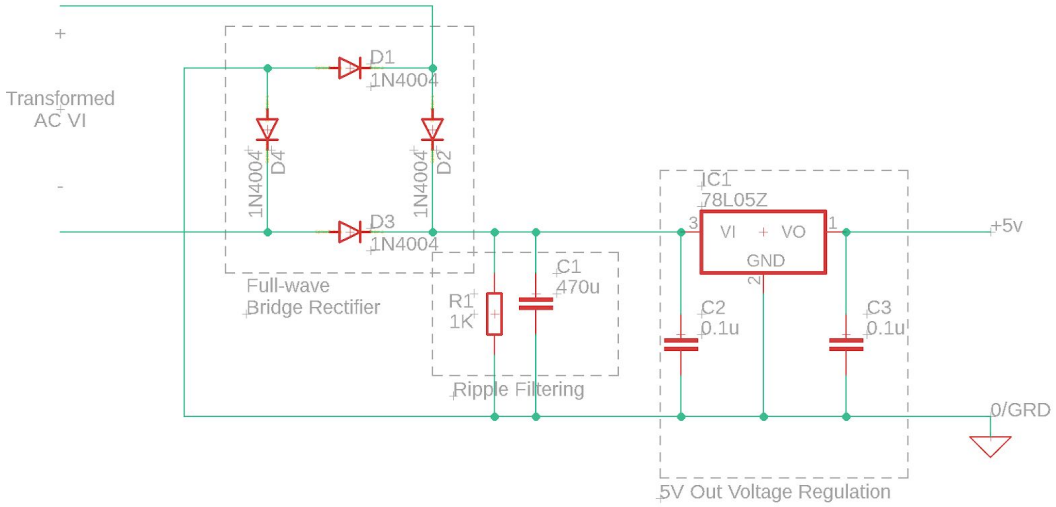


Figure 3.

2.1 Bread Board

This circuit covers the 3 central design blocks as follows implemented and outlined in the schematic figure 3:

2.1.1 Rectifier

Standard, general purpose rectifier diodes from the 1N400x range were used in this construction. They could supply ample rectification at varying temperatures and our expected inputs were well within the datasheets listed limits.

2.1.2 Filtering

In order to smooth the rectified voltage signal, a standard 1K resistor in parallel with a 470uF capacitor were included in the design. But in order to minimise ripple to input to the voltage regulator this block must undergo testing and tuning to determine an optimal RC combination, discussed later in *Construction and Testing*.

2.1.3 Voltage Regulation

Previously it was discussed there were 2 options, zener and IC. A fulling working zener construction able to be stable and varied loads and input was determined to be too high an

added complexity and the L7805 IC unit was used. This eliminated the concern for a constant output (V) to varied loads but note will require minimum input and it's real efficiency will be later discussed

2.2 PCB

The PCB design was put into a schematic (figure 4a) in altium using a standard, provided design. Then arranged into a PCB board design, fig 4b.

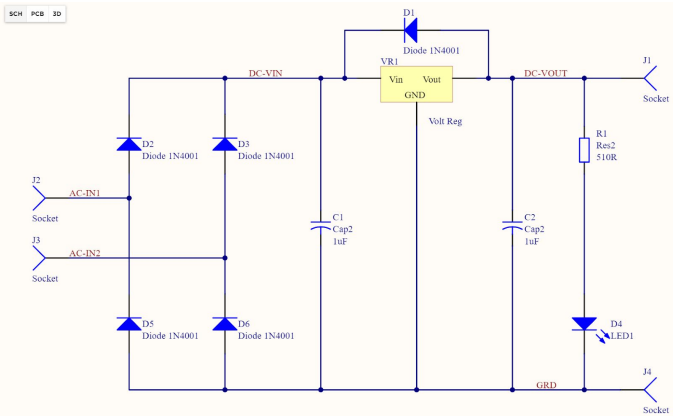


Figure 4a

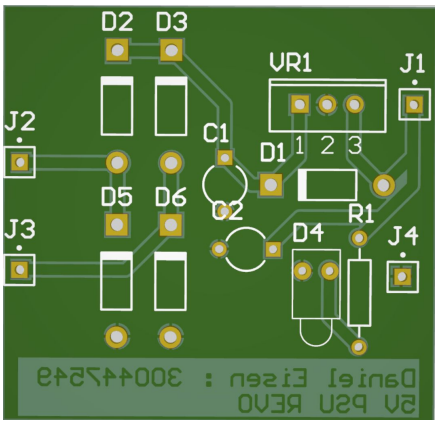


Figure 4b

It is worth noting the above design lack a large capacitor, forgoing the 470u for a 1u, on the V-reg input and the results/consequences and solution to that is discussed in the sections following.

Another addition the the added diode across the IP/OP of the V-reg. This is just added short protection.

Other than these addition/omissions this design closely follows the the breadboard design

3. Construction and Testing

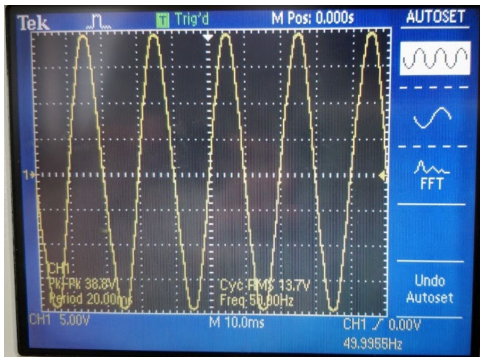
3.1 Bread Board (See appendix for Final Image)

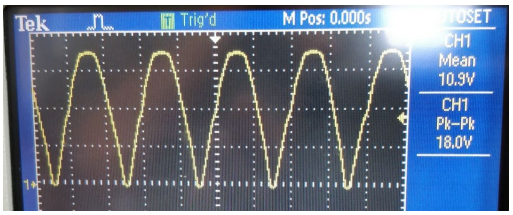
The prototype of the PSU is constructed on a breadboard to allow for individual block construction and ease of testing.

3.1.1 Rectifier

Step one is the rectifier construction. The 4 diodes are arranged to form the bridge circuit seen in fig 2. With the transformer on the input. The output (across a resistor) is probed to confirm the rectification. This circuit did not require any tuning, with results shown below,
input AC left, rectified voltage right.

Figure 5a,b





3.1.2 Filtering

Step two is building and tuning the filtering section to reduce ripple percentage to a minimum. Ripple percentage is defined as $\%R = \frac{V_{op-p}}{<V>} x100$. Ie the ratio between the voltage variance of the ripple and the DC offset voltage.

This is important as the voltage regulator IC must be constantly supplied with minimum input voltage, >= 6V [see appendix data]. This was tuned by setting the constant 1K resistor value and varying the capacitance of C_1 from 1uF to 1000uF and measured %R, see figure 5 below. Note for full data see appendix.

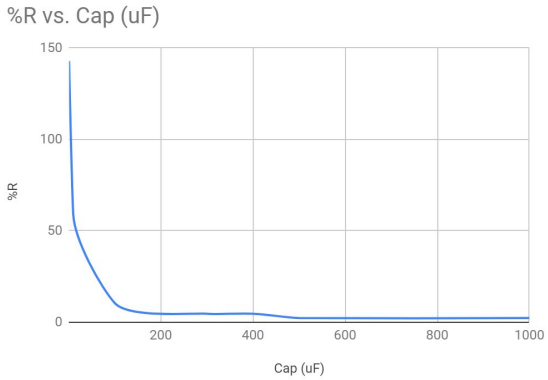


Figure 6

From the data collected, you can safely determine that capacitor values over 450uF produces a small enough to be perfectly usable, %R < 4%. In the constructed circuit uses a 1000uF cap but as low 470uF would work.

3.1.3 Regulation Performance

Two performance metrics were measured for; output current to differing loads, figure 6 and Power Out / Power In, figure 7.

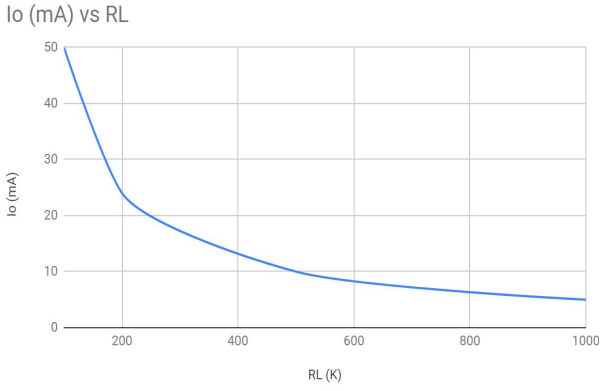


Figure 7

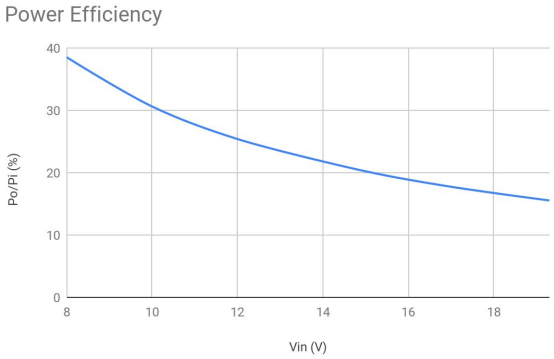


Figure 8

From our measurements we see that although can supply dead steady 5V from varied load and Vin it does not hold a steady current, actually significant drops and as Vin is risen the power efficiency drops.

3.2 PCB (See appendix for Final Image)

After the PCB was soldered, connected to the transformed AC input and a load connected at output. Vout is initially very jagged and non constant (fig 8), this due the non optimal filtering on the regulators input and should be replaced with a larger capacitance, at least 10uF. This produces a smooth output 5V (fig 5).

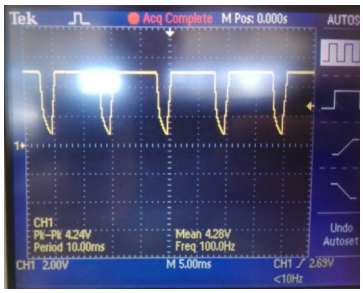


Figure 9

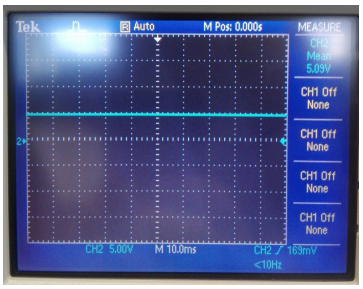


Figure 10

Power efficiency of the OCB is the same as presented in the breadboard design and the output current to a varied load is shown below.

RL (K)	Vo (V)	IL (mA)
1	4.97	4.97
2	4.98	2.5
3	4.98	1.68
5	4.98	1.01
7	4.98	0.73
9	4.98	0.57

4. Discussion and Conclusions

The breadboard construction provided little problems in implementing the circuit block individually and then integrated. Though with use of large capacitors in the filtering circuit it was easy to have the wrong polarity for long enough to produce confusing results without noticing the burning smell.
In integration testing it must be ensured the voltage regulator is supplied with a constant minimum voltage else confusing and irregular output readings will be seen. This was almost always due to non-optimal filtering, so adjusting the RC fixed this.

PCB was the direct opposite; other than the filter capacitor replacement. The soldering of the board itself presented the greatest issue. This is due almost entirely to the exposed ground plane and over soldering leading to it being very easy to create short. Could be solved with a shield, or in this case persistence.

As shown in the testing sections, although this PSU is successful in providing a constant 5V, it lacks current preservation and maybe more importantly has a low initial power efficiency that only gets worse with increased input voltage. Future designs could make use a better regulator IC, perhaps a switching voltage regulator.

Personally I found this a very fulfilling and interesting, as it used my gained knowledge to form a tangible and ‘usable’ device. I was my first larger soldering project and first PCB and that was quite exciting.

5. Additional Questions

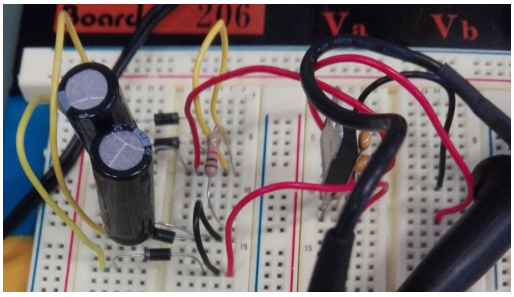
a. DC would pass through one path of the rectifier bridge without issue and the input voltage (- 1.4V over the diode) would output. The smoothing capacitor will charge and stay charged further reducing the voltage by V_c.
Therefore for the DC input to be successfully regulated, V_{in} - 1.4V - V_c >= 6V.

b. The final design uses 2 main capacitances. One between the rectifier output and V-reg input (the larger), to smooth ripple and to filter out higher frequency noise on the regulator input. Another smaller cap on the V-reg output to also filter noise. The minimum and recommended value are listed on the data sheet.

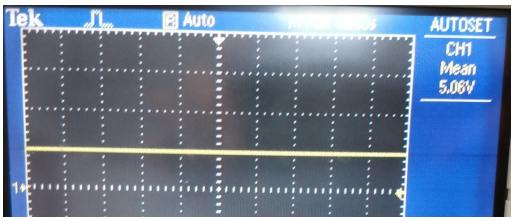
c. i. The rectifier as is current uses general purpose diodes with a 2 diode pass through per path. This has a minimum v-drop of 1.4V. So the max input will not make it through the bridge.

   ii. A single diode rectifier would allow a non zero output, a lower minimum required v-regulator and maybe a controlled voltage amplifier.

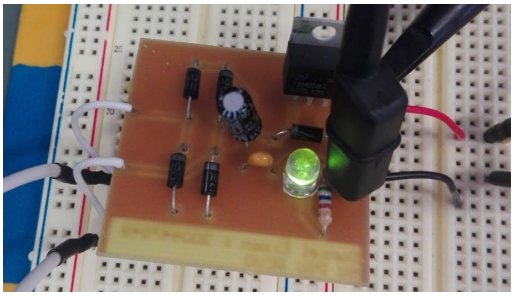
Appendix



Fully constructed breadboard circuit



Steady V out of breadboard



Fully constructed PCB

Ripple Measurements Breadboard

Cap (uF)	Vpp	<V>	%R
1	16	11.2	142.8571429
10	8.4	14.2	59.15492958
100	1.8	17.1	10.52631579
300	0.8	17.1	4.678362573
400	0.8	17.1	4.678362573
500	0.4	17.4	2.298850575
1000	0.4	17.4	2.298850575
R	Vpp	<V>	%R
10000	7.6	14.7	51.70068027
1000	16.4	12.4	132.2580645
500	17.2	10.9	157.7981651
250	17.4	10.7	162.6168224
100	17.4	10.5	165.7142857

V in vs V regulated Out

VI	VRO
4	2.534
5	4.165
6	5.024
7	5.031
8	5.031
9	5.031
10	5.031
11	5.03
12	5.03
13	5.03
14	5.03
15	5.029

Voltage Regulator Performance Metrics

Vi	Vo	Io (mA)	R	
8	5.031	5	1000	
8	5.029	10	500	
8	5.024	24	200	
8	5.018	50	100	
Vi	Ii (mA)	Vo	Io (mA)	Po/Pi %
8	8.16	5.031	5	38.53400735
10	8.21	5.029	5	30.6272838
12	8.25	5.029	5	25.3989899
15	8.29	5.028	5	20.21712907
17	*8.335*	*5.027*	*5*	*17.73880518*
19.3	*8.378*	*5.0261*	*5*	*15.54187689*