# Paul's DIY electronics blog

Arduino (7) PicAXE (1) Raspberry Pi (26) T&M (31)

**Monday, May 18, 2015**

## Tuning a 0..30V DC 0..3A PSU DIY kit

*Before you read on, you should know that after building two versions of the DIY kit, I was not very impressed about the stability, noise, but above all, the complete lack of any form of protection. Think about that before you connect something valuable to the supply.*
*Subsequently, I designed a more modern power supply that you can follow here:*
*http://www.paulvdiyblogs.net/2017/07/my-new-power-supply.html*
*After building two of them as well, and learning a lot and have a lot of fun experimenting and building, I still decided to purchase a lab quality professional power supply.  That should hopefully tell you a few things.*
*There are many things you can learn about the kit in this post, and also select possible improvements, but please understand that I can't really help you anymore. It's been too long ago.*


Due to a move, I recently sold my large lab PSU, and needed a substitute. I wanted to be a little more flexible, and did not want a huge and heavy supply on my bench anymore.

Searching the web, I came across an inexpensive DIY kit that implemented a very popular design for a power supply. I seldom if ever need more than 1A, so I used the kit to tune it to my liking, and also added the latest modifications for the original design.

I added an LCD display and one addition to the original design is a current setting mechanism, using the display, so you can set the current limiting or constant current mode before connecting the DUT.

I have built two supplies and can connect them in parallel to get more current, or in series to get a true dual +0..30V *zero* -0..30V supply or a 0..60V supply. One is designed for 3A and one for 1A max.

After some fiddling, I also designed a simple dual tracking system when the two supplies are used in series, so one supply controls the other.
Enjoy!

**Search This Blog**

**About Me**

**paulv**
Netherlands
View my complete profile

**Blog Archive**



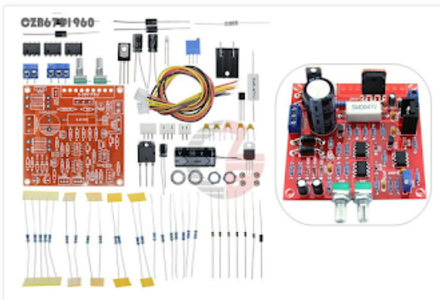
**Alphabetical List**

paulv

## Tuning a 0..30V 0..3A PSU kit

The very popular PSU schematic for a lab power supply that can supply 0..30V and 0..3A (http://www.electronics-lab.com/projects/power/001/index.html) has created a lot of interest. So much so, that several Chinese suppliers have created a kit with just about all parts including a PCB for a very attractive price. I paid $12.64 and that is with free shipping. On top of that, it took less than 2 weeks for the kit to arrive.

I purchased something this one:



Note that this link may no longer work over time, just search for “AC 24V 0..30V 0..3A DIY kit” and you shall find…

If you buy the parts from a usual on-line supplier, you probably spend more on the shipping cost alone. So where’s the catch?

The kit is based on the original article and has some issues that need to be addressed. There are several postings on the electronics-lab forum that go into great details about the original design:
http://www.electronics-lab.com/forum/index.php?board=2.0

However, if you adhere to a few simple requirements, you can use this kit without too many changes and create a fine supply for your bench that will probably be adequate for 90+% of your power supply needs.

Here is the constraint: If you stay below a maximum current of 1.5A, the kit will work perfectly with a couple of changes in the components and we will discuss that here. The kit can also be extended with more functionality and that is also discussed here.

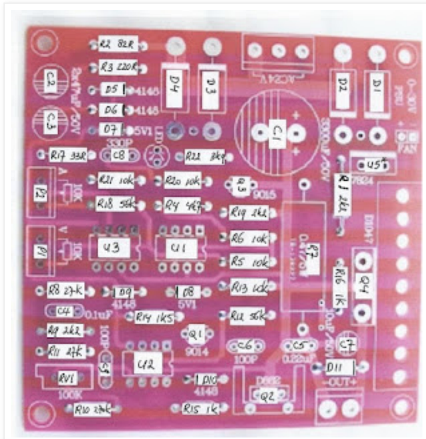
To start off, several components supplied with the kit (the list is at the very end of this post) are different from the original design, so let’s go over those:

D7 and D8 are 1N4733A 5V1 zener diodes, and they require 49mA for a bias. This deviates from the original design that has low current 5V6 zener diodes with a lower bias. Q3 is a 2SD9015 and Q1 is a 2DS9014. Q2 is a 2SD882 and Q4 is a 2SD1047. Q4 is much easier to mount on a heat sink, compared to a 2N3055.

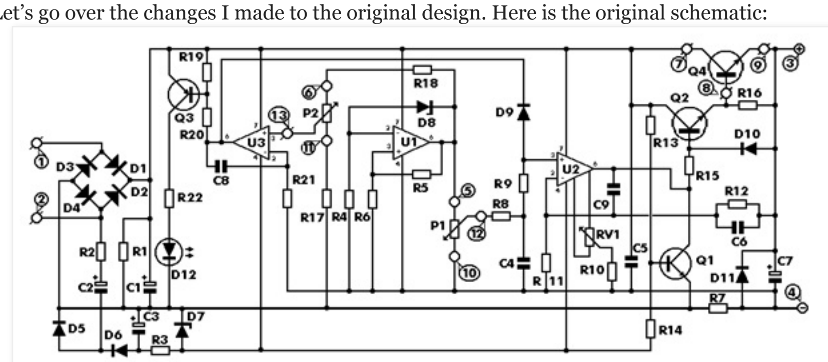
Most other parts are following the original design, however, a few resistors supplied with the kit have a wattage that is too low. R2 of 82R should be 1W, R3 of 220R should also be 1W. The supplied 0.25 Watt resistors will get too hot. R1, which is 2K2 1W will also get pretty hot, so mount that a little above the PCB, or replace it with a 2Watt resistor. R7 should also be mounted a little above the PCB.

An extra part that is not in the original design is an LM7824, to create a 24V DC supply for a fan. If you are like me, you will have a lot of 12V fans, because that is the voltage used in PC’s. In any case, I switched the LM7824 to an LM7812, because I drive a few additional LED’s with it, and also supply a Volt/Ampere display with it. If I decide to change the unit for a higher current, I may need a fan, and I have several 12V DC fans in my stash. If you decide to keep the LM7824, double the resistor value that go to the LED’s. The meter can handle the 24V. (see below) You can mount the LM78XX on the PCB, but I didn’t. It gets pretty warm, so it went on the heat sink.

If you are going to limit the maximum current to 1 or 1.5 Amp, there is no need to go fancy on the transformer, and you can use a more or less standard transformer with 24VAC and 1 or 1.5A current.
The supplied TL081 op amps's have a deficiency that we will want to avoid, so we will not use them.
The kit comes without a schematic, parts list or PCB layout, although the stenciling on the PCB shows the values, but not the part numbers.

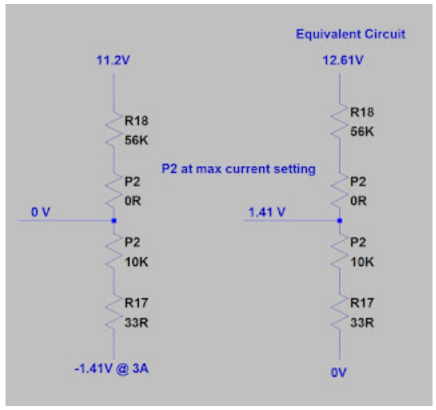


Let's go over the changes I made to the original design. Here is the original schematic:




The parts list supplied with the kit is listed at the end of this post, with the changes and additions.
From that supplied parts list, we will not use D7 and D8 is a 1N4733A 5V1 zener needing a 59mA bias. We will replace this type with a BZX55C5V6 or BZX79C5V6 zener, both requiring only 5mA bias current. U1 will set the reference voltage to twice the zener voltage so 11.2V. With the required 5mA bias for D8, R4 should be 1K, not 4K7.

Because we need to limit the maximum current to either 1 or 1.5A, R18 needs to be recalculated. This resistor had the wrong value (56K) in the original design anyway.
Here is a simplified diagram to help with the calculation, just in case you want to use another maximum current version:




Let's see where the original calculation for R18 went wrong, and resulted in a maximum current that would literally blow a fuse, or more.

To calculate R18 for a maximum current of 3A:
Vref = 2 x D8 of 5V6 = 11.2V
Voltage over R7 of 0.47R at 3A is = R7 * Imax = 1.41V
At max current setting of P2, the top is 0R and the bottom is 10K
P2+R17 = 10K + 33R = 10033Ohm
For the equivalent circuit:
R18 = P2+R17 * (Vref+VR7 - VR7) / VR7
Or
R18 = 10033 * (12.61 – 1.41) / 1.41 = 79K694
The original value was 56K, but that would mean a maximum current of :
VR7 = 56000 / (56000 + 10033) * 12.61 = 1,916V / 0.47R = 4A! Oops...

The following values are calculated for R18 with the new low current 5V6 zener diode for D8:

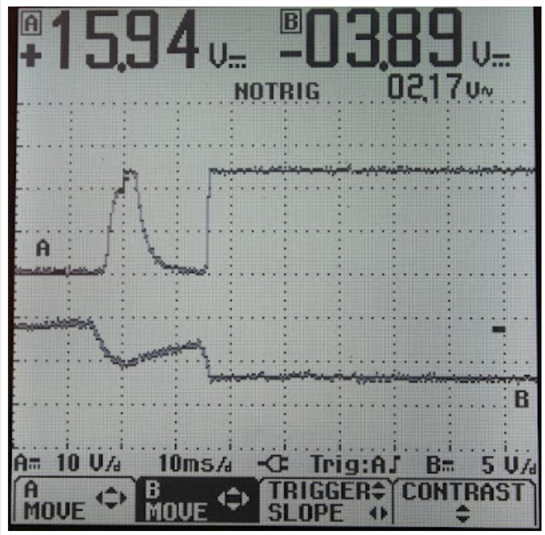
R18 = 72.5K @ 3.0A
R18 = 169 K @ 1.5A
R18 = 259 K @ 1.0A

If you want to be precise, you can still use the original R18 value of 56K, but add a trimmer of 200K or 250K in series. This trimmer can be mounted on P2, so you don't have to mess with the PCB.

So what else was wrong with the original design, (if!) we keep to the 1.5A max. Well, the original design used Op Amps that had a flaw.

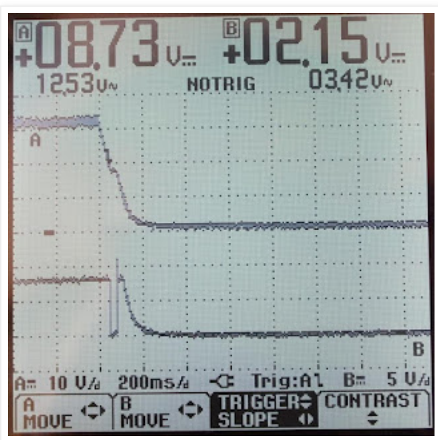
Several more changes are related to their replacement. Because we will not use the TL072, we can drop Q1, R13 and R14. They were needed to remove a glitch from the output that was caused by the TL072. The circuit around Q1 was designed such that as soon as the negative 5V6 supply collapses, when the mains is switched off, it would immediately turn Q2 off, and therefore also the output. With Q1 in place, it would protect the Device Under Test or DUT from voltages higher than what you set the output to. That can be deadly for the DUT.

Unfortunately, the circuit around Q2 is still not perfect. There were still situations by which a glitch was introduced at the output when the main supply is switched-on or switched-off.
Let me show you:




Switching on: The top trace (A) is the output of the PSU at 25V and with a 500mA load. The bottom trace is the negative supply. The negative supply goes down from 0V in the rhythm of the main frequency, in my case 50Hz, until the D7 zener kicks in. The base of Q1 is set to 0V by R13 and R14, but this setting is upset with the supply “swinging” into place, turning Q1 on and off. Depending on the point in time when you flip the switch in relation to the main frequency, you will see this behavior. If you try it 10 times, you may see this effect once or twice.

So what happens when you switch the supply off at the mains level?

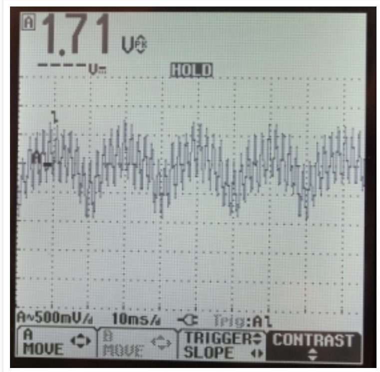



Bottom trace is the output to the DUT. So there is another glitch that can happen. Not always, but it can happen.
So, although the circuit around Q1 did a good job as intended, it removed large spikes above the output voltage setting, it was not perfect.

By replacing the three TL072’s with the TLE2141, we can eliminate the Q1 circuit all together. Furthermore, with the new op amps, the negative supply can be reduced from -5V6 to about -1.3V. That’s why we will not need D7.

We’re not done with the negative supply yet. In the Current Limit (CL) mode, for all practical purposes, the supply actually switches to a Constant Current (CC) mode. U3 does not switch from rail to rail, but switches to about +3V. This is enough to turn the CL LED on, but there is still a voltage at the output. You can now slowly turn P2 counter clockwise, and you’ll see the voltage at the output drop, while the current stays the same. This is the Constant Current mode. So in the CL/CC mode, the output from U3 switches from the positive supply of 26V to about +3V and then slowly goes to the level of the negative supply, at which point the output at the terminals is removed completely.

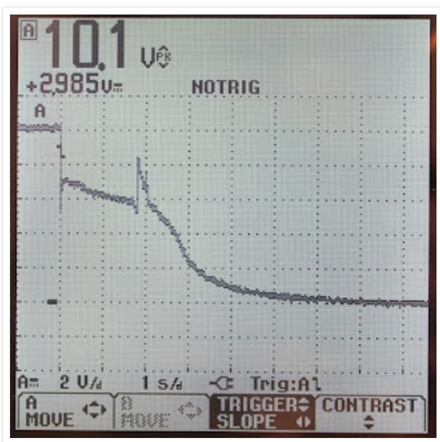
Unfortunately, this is not really a great CC mode, if you look at the voltage output supply:




There are two sources for the 1.7 V p-p “noise” riding on the output of the supply. One is mains hum, the result of the rather crude way the negative supply is concocted. The higher frequency noise is the result of the closed-loop activity between U3, U2 and the output stage. U3 and U2 are in a constant battle to keep the output high (U2) and at the same time, U3 is limiting the output to stay within the current limit. There is little we can do about that without doing a major redesign, but we can at least remove most of the mains ripple.

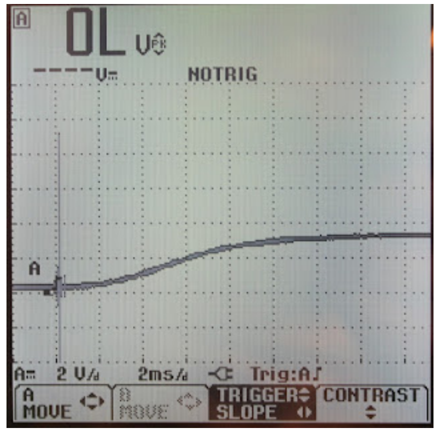
We do that by replacing R3 with an LM337 voltage regulator (U6), and we set the output level at -1.3V with two additional resistors, R25 and R26. We’ll also add a small filter capacitor, C14 of about 22uF/10V.

If you have a habit of supplying your DUT with power by turning the PSU on and off, even with the above changes, you may still introduce a glitch in the voltage at the output terminals. I have experimented with a few possible solutions, but gave up because I could not find a simple solution to fix this.




Here the mains is switched off while we’re looking at the output voltage. The slope is depending on the current that is pulled from the supply, so that curve may be steeper, but it’s still not very pretty.

Here the main power is applied while the output has been set for 3.3V, the most critical voltage level for devices under test. Notice the large spike that significantly exceeds the maximum voltage that has been set.




So to still allow a clean turn on and turn off to the DUT, I added a double throw switch into the mix. One part of the switch connects the anode of D9 to ground, because this will remove the power from the output. To show myself that there is no power on the output, the other half of the switch turns on a red LED. The LED is connected between the 12V and via a 4K7 resistor to the switch, which connects it to ground. Simple and effective.

I also wanted to have a Voltage and Current display, so I purchased one of these:


These are below $10 on Amazon or eBay. The small red and black wires on the right provide the power to the logic of the unit, and that can be anywhere between 3.5 and 30V DC. I connected them to the LM7812. Note that these displays should really be galvanically seperated from the supply to avoid noise injection. The alternative is to do some serious filtering in the supply voltage chain to avoid that noise.

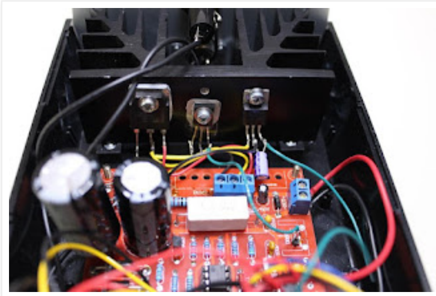
These displays are capable of handling a car battery or big motor currents (up to 10A with the internal shunt), and therefore the current and voltage sensing wires are very thick. I replaced them with different wiring. In any case, the red wire is connected to the output of the PSU, and is the voltage sensing input. This device has an internal shunt resistor, and that is connected between the yellow and black wire. To make it easy, I connected the black wire to the minus output of the PSU (4) and that makes the yellow wire the “new” minus output. The shunt will make a tiny difference because it sits outside of the feedback loop, but the error is extremely low, because the shunt is extremely low in value as well.

On the back of the unit are two tiny trim pots you can use to adjust the voltage and current. To set the voltage of the PSU precise, I used a 10-turn pot meter. Look on eBay, they are not that expensive ( $6.59 with free shipping) if you forgo the super accuracy. (http://www.ebay.com/itm/10K-Ohm-3590S-2-103L-With-Turn-Counting-Dial-Rotary-Potentiometer-Pot-10-Turn-/231499582066?pt=LH_DefaultDomain_0&hash=item35e6734a72)

There are two more additions that I made. One is to add an LED to show that the unit has main power. That green LED is connected between the 12V and through a 4K7 resistor to ground. The final addition is another 3300uF/50V capacitor (C12) parallel to C1, to give more stability to the raw supply and to reduce ripple at higher currents.

I used a large heat sink, and mounted the LM7812, Q2 and Q4 on it. There is plenty of room to add another output transistor parallel to Q4, if I decide to increase the current.

With this heat sink, I will not need a fan with the current staying below 1.5A.



From left to right: Q4, Q3 and the LM7812.
Q4 and Q3 are isolated, the LM heatsink is ground, so does not need it.

To create the front panel, I printed a design on photo paper, used double sided tape to fix it to the metal front panel and cut out the holes.


Here is where the 24V AC comes in. I can use different size transformers, and use them for several applications this way.


I did not use the supplied 10K pot meter for the current setting, because it did not come with a nut. It needs an M7 nut I didn't have, so I used another 10K pot meter I had in my stash.
The supplied pot meters with the kit use an M7 nut, so I ordered a washer/nut set here : (http://www.aliexpress.com/snapshot/6615095977.html?orderId=66887397125609)

After I finished all the modifications and started to experiment with the supply, I saw the need to add a way to show the current limit setting, so I have now added a little circuit to the supply so I can set the Constant Current/Current Limit.

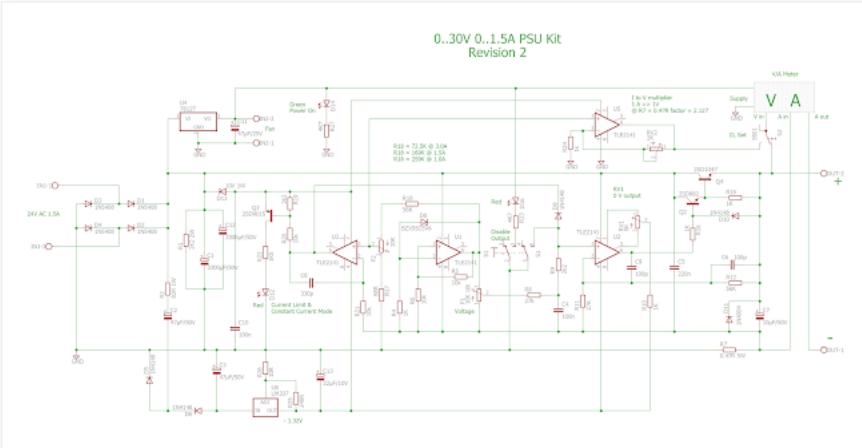
Because I already have a voltmeter, the easiest method was to use that to show the current setting. However, showing the value on the current meter display with the unit I use is tricky.

To show the current setting on the voltmeter, all we really need is a convertor that translates the current limit setting to a voltage.
To show the relation of 1A = 1V, with R7 at 0.47R, we need a multiplication factor of 1/0.47 = 2.127.
By using an additional op amp (U5), we will make this circuit independent of the maximum current of the PSU.

If you look at the schematic, the circuit around U4 implements that function.

RV2 can be adjusted by setting P2 to the maximum value of the current, say 1A. You can measure the voltage at the wiper of P2 with a DMM and set P2 to read 1.00V on the DMM. If you implemented R18 in combination with a trimmer, adjust that trimmer first to show 1.00V with P2 at maximum. Push the CC set button and adjust RV2 to have the voltmeter of the PSU show 1.00V as well.

Here is the final schematic:




Here is the original parts list as supplied with the kit, but with my changes and additions listed as well:

| | |
|---|---|
| R1 = 2K2 1W | Replaced with a 2W version |
| R2 = 82R | Replaced with a 2W version |
| R3 = 220R | Not needed (replaced with an LM337) |
| R4 = 4K7 | Value changed to 1K |
| R5, R6, R13, R20, R21 = 10K | R13 not needed |
| R7 = 0.47R 5W | |
| R8, R11 = 27K | |
| R9, R19 = 2K2 | |
| R10 = 270K | Value changed to 1K |
| R12, R18 = 56K | R18 see text |
| R14 = 1K5 | Not needed |
| R15, R16 = 1K | |
| R17 = 33R | Value changed to 68R |
| R22 = 3K9 | Value changed to 1K5 |
| RV1 = 100K 10turn trimmer | replaced by a 5K 10 turn trimmer |
| P1, P2 = 10K linear | P1 replaced with a 10 turn potmeter |
| C1 = 3300uF / 50V | |
| C2, C3 47uF / 50V | |
| C4 = 100nF | |
| C5 = 220nF | |
| C6 = 100pF | |
| C7 = 10 uF / 50V | |
| C8 = 330pF | |
| C9 = 100pF | |
| D1, D2, D3, D4 = 1N5408 | |
| D5, D6, D9, D10 = 1N4148 | |
| D7, D8 = 1N4733A 5V1 zener | D8 = BCX55C5V6, D7 not needed |
| D11 = 1N4004 | |
| Q1 = 2SD9014 | |
| Q2 = 2SD882 | |
| Q3 = 2SD9015 | |
| Q4 = 2SD1047 | Not needed |
| U1, U2, U3 = TL081 | Replaced by 3x TLE2141 |
| U4 = LM7824 | Replaced by a LM7812 |
| D12 = red LED | |
| PCB | |

Sockets for U1, 2, 3, input and output connectors, sockets and wire harnesses for P1 and P2, heat sink for Q2

Additional parts:

R23, R27 = 4K7

R24 = 1K

R25 = 240R

R26 = 10R

RV2 = 2K

RV3 = 200K or 250K (optional, see text)

U5 = TLE 2141

U6 = LM337

C 11 = 47uF/25V

C12 = 3300uF/50V

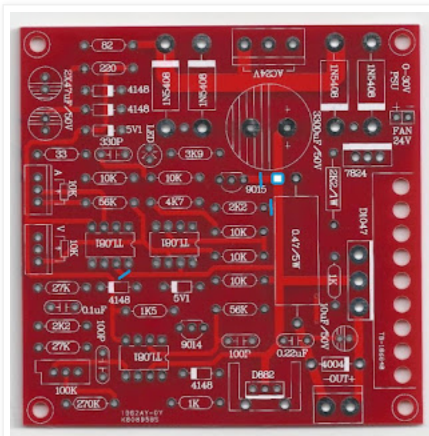
C13 = 22uF/10V
D13 = 10V 1W
D14 = Green LED
D15 = Red LED
Volt/Ampere panel meter
S1 double throw switch
S2 single throw push button

### Modifying the PCB to the latest version of the supply

In the above text, I have given an overview of the changes to the components supplied with the kit, to make it work a little better.

First of all, we need to implement the supply changes to the opamps (through D13), and so a few traces need to be cut on the PCB. This will allow us to also switch to the TLE2142 opamps.

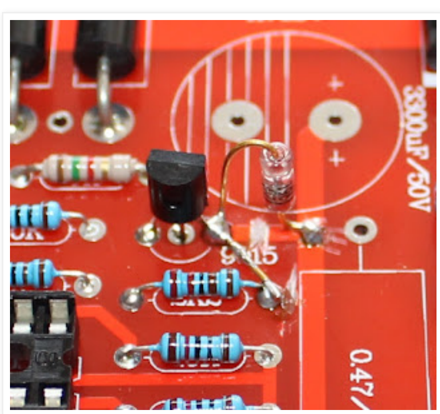
The photo below will show you what traces to cut (in blue) on the component side of the PCB:




1. The connection of the unregulated supply to the emitter of Q3
2. The connection of the unregulated supply to R19
3. The  unregulated supply connection to U3 pin 7

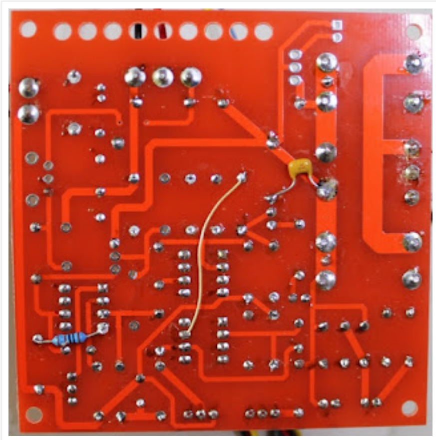
To install the new 10V zener diode D13, you need to remove some of the lacquer on the positive supply trace, as indicated on the photo.

The cathode of D13 is then soldered on this spot, and the anode goes to the emitter of Q3 and also to the disconnected end of R19.
See this photo for a closer look:



The original zener D7 is not installed but C14 will be mounted in this location.
The LM337 will be mounted in place of R3, and I just figured out a way to make the connections to the ADJ pin and R25 and R26 to connections that are near. Make sure the (metal) body of the 337 does not connect to anything, it carries a voltage. Use heat shrink tube if needed. With only about 10mA current, it will not get warm at all.

Turn to the reverse side of the PCB, and look at this photo:



The new C10 is mounted on the reverse side of the PCB.
R10 is mounted on the back to make it easier to connect to the negative supply.
The pin 7 of U3 is connected with a wire to the anode of D13.

The following values of components from the kit are now changed:

R10 (from 270K to 1K),
R17 (from 33R to 68R),
R22 (from 3K9 to 1K5),
RV1 (from 100K to 5K) and
U1, 2 and 3 (from the TL081 to the TLE2141)

Despite what others have posted, I had to connect the minus supply of U2 to the negative supply, not to ground. The reason was that I could not get the output to go to 0 Volt with P1. It did go to 0V with the current limiter. With a negative supply of only -1.2V, it still does not go to 0V, but +25mV is close enough. (RV1 at 5K and R10 of 1K allowed the output to be adjusted from +43mV to + 25mV)

It has been stated that R15 and D10 have no purpose, but if you connect U2 to the negative supply, R15 and D10 remove any negative voltage from the output of U2 to the base of Q2.

Finally, if you only use the supply to about 1A, you can use a 220K value for R18 and you do not need to add RV3.  If you use a 24V AC transformer, you probably don't need to limit the maximum output to a precise 30V, and if so, you don't need to install RV3 and R11 stays at 27K.

So with these changes and a few more parts, the kit can be modified and the total price will still be very attractive.

**Latest update. August 4 2015**

I was still not very happy with the CC mode of operation. Even with the above mentioned modifications, there is still too much noise and a mains ripple on the output during the CC/CL mode.

As it turned out, a lot of this noise comes from the Volt/Amp display I'm using. The switching regulator that is used on this display injects a lot of noise back into the supply. I also was still not satisfied with the ripple on the reduced supply (by D10) for U3, U5 and Q3, and connecting the display to this supply made it all worse.

So, to tackle these problems, I went back to using the LM7824 that was part of the kit, and used that instead of D10, the 10V zener that was used to create the supply to U3, U5 and Q3.

To counter the noise injection from the display, I now used D10 to reduce the raw supply and used that to power the display unit.

While on my quest to reduce noise, I also moved the display current shunt from the output terminal, to outside of the current feedback loop. This reduced some more noise, but also made the current setting more precise. (because the shunt was inside the feedback loop, the voltage over the shunt at higher currents created an error. Small because the shunt seems to be only 25 mOhm, but still)

In order to put the shunt there, you need to cut a PCB trace from the raw ground supply to R7 and connect the current meter shunt output at the supply side of R7. Make sure R21 and R17 are not measuring the current shunt of the meter, but only R7. The current meter shunt input goes directly to the connections of the anodes of D3 and D4 and the negative connections of C1 and C2.

To eliminate a possible ground loop, the ground supply lead for the display is no longer used. The ground for the display unit is coming from the shunt connection to the raw supply ground.

To eliminate large currents on the PCB as much as possible, I connected the collectors of Q4 and Q3 directly to the point where the cathodes of D1 and D2, and the filter capacitors C1 and C2 come together.

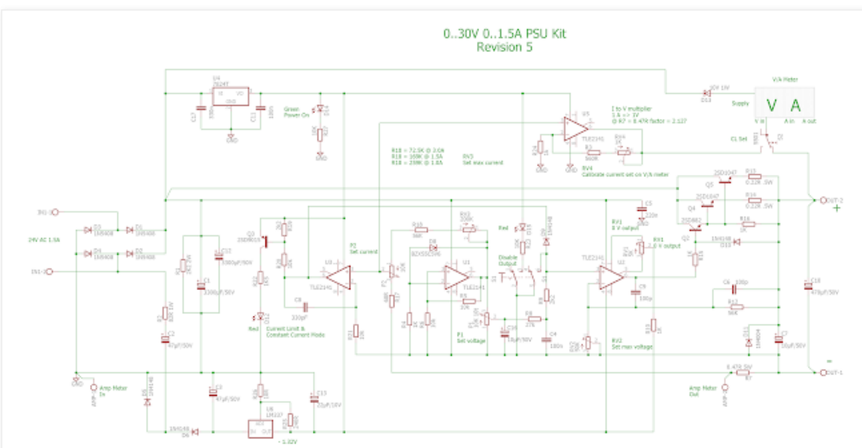
I also installed the "optional" trimmers to set the maximum output voltage (RV2) and maximum output current (RV3). It is important to set the maximum current limit, because the granularity of P2, a normal pot meter, is greatly increased allowing you to set the current level more precise.

C16 is used to eliminate some more noise.

Because the LED's D14 and D15 are now connected to the 24V rails, their current limit resistors (R27 and R23) need to double in value.

Lastly, the output capacitor C7 was enlarged from 10uF to 470uF. That seems a lot, but professional supplies actually use a lot more.

Here is the final schematic with the latest revisions:




The rise time of the supply is now about 5mSec and the fall time is just over 2 mSec at maximum voltage and current, measured with a dynamic electronic load, capable of 50uSec transients.

With all these modifications, the output noise is now 18 mV p-p across the voltage and current spectrum, and, more importantly, stays at that level in the CC/CL mode. To qualify that, the noise floor of my scope with the probe tip grounded is 12 mV p-p, and with the supply switched off, the noise floor is just below 16mV p-p. With a positive mind, you could deduct that the output now only adds 2 mV p-p noise. Mission accomplished!

One future mod I'll do is to add a parallel output transistor to Q4. My typical applications are low voltage, and this is the largest burden for the pass transistor, because it has to bleed-off the excess voltage. I'll rearrange the LM7824 on the heat sink to make room for the second 2SD1047. I'll use .22R emitter resistors (because I have them already) to pair them up.

And yet another update: Aug 14

Not only did I indeed install a parallel series transistor (2SD1047), I also modified one of my two supplies such that it could handle more current.

I'll continue to use one which is fed by a 24V 1.5A transformer, but that maximum output is limited with a current in excess of about 25V, when the regulation starts to falter because the raw voltage starts to collapse.

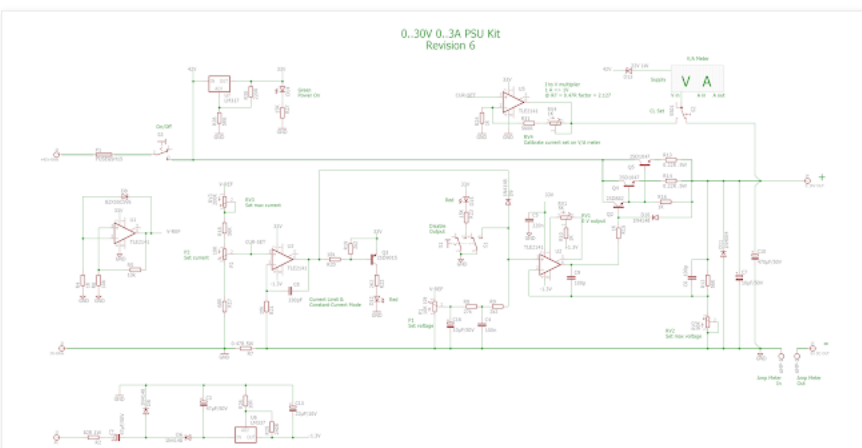
So, I needed a transformer with a higher voltage rating and a higher current rating to pull this off. Unfortunatly, the most common transformers are 15-0-15 or 30V at 3A or more, and that will produce a raw voltage that is too high for the choosen op amps. The TLE2141 can handle up to 44V, but 30V AC already translates into 30 * 1,414 = 42V. Without a load, even with the bridge diode voltage drops, that is still too much. More so, since two op amps are also fed with a negative 1.3 V supply. A 14-0-14 supply would be ideal, but I could not find one.

With the higher currents, you also need a fan to cool things, so that was added as well. See a separate post on a solution that I built. At a later date I'll include that circuit into the main schematic.

The transformer I ended up buying is a 15-0-15V AC at 3,3A. With 3,3Aac, I should be able to get a solid 2Adc, plenty for my purposes. I also changed the 4 diodes that were used in the full bridge configuration and selected a bridge with 600V 10A that can be mounted on a cooling fin. A bit overkill, but it was for the same price as an 8A version. You need some overkill because of the in-rush currents to the main filter caps. The two 3300uF filter caps are inadequate for these currents, so I installed two 10,000uF at 63V ones. I used a separate enclosure to put this all in, and use 4mm banana posts and jacks to connect the raw supply to the PSU. If you do that, remember to also feed an AC signal to the PSU because that is used to create the negative 1.3V rail. The enclosure is completed with a main switch, a main fuse and a power indicator. I also feed the AC 15-0-15 taps to banana jacks on the front panel, so I can use that for other purposes.

While running some more tests, I decided to put the ampere meter shunt back at the output. There was too much of an error in the measurement, because it included the currents of the actual supply itself.

The changed schematic for the new supply is as follows:




You'll notice that I departed from using the original way of showing all connections with wires. I now grouped the functionality so it's hopefully easier to understand.

Because the op amps are limited by their 44V rail-2-rail supply, I went back to using an LM317 to create a nice and steady 33V. This is just enough headroom to regulate the output to 30V. I used this supply to feed all op amps now, and that also required resistor value changes for the LED bias resistors. It also means that the supply modification with D10 needed to be undone on the PCB.

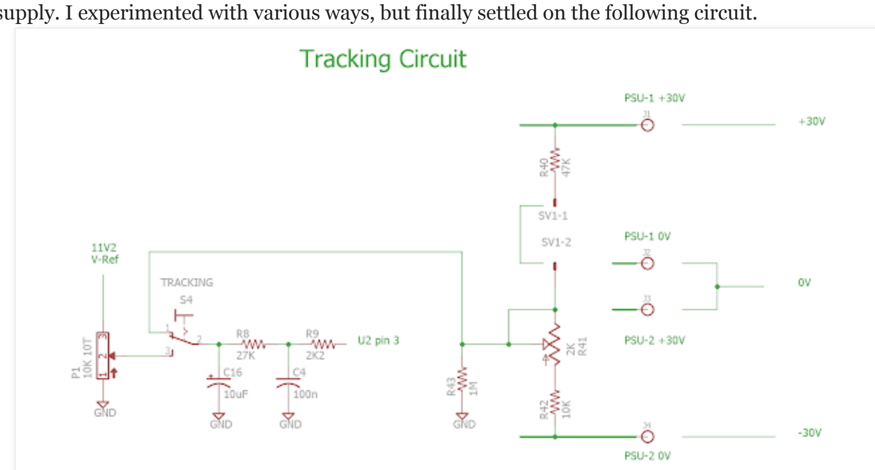
You'll notice that the bridge rectifier diodes are gone, and so are the filter caps and the bleed resistor. They all moved to the raw supply enclosure. I actually doubled the value of the bleeding resistor by putting two 2K2 2W resistors in series, because I found it was getting too hot with the additional voltage. I also changed D13, the Zener diode feeding the V/A display, to a more beefy 1W version, that I only had in a 22V version. I paid special attention to getting the main raw connections ( they are now a bit thicker in the schematic) to the required parts, and avoided going through the PCB as much as possible. C7, the 10uF on the output terminals is an anomaly, I just left it on the PCB, but is has little use compared to C10 which is mounted directly on the output terminals.

Other than that, there were no major changes, and the supply works really, really well. I now only need to install the fan controller but I wanted to play with the fan starting point a little more so it's quiet with small loads but kick in when needed.

Update Aug-28-2015:
I finally was able to find a simple but effective method to "tie" my two supplies together and create a tracking +30 0 -30V supply, or a +60V supply.

The principle is easy, if you connect the 0V output of one supply to the +0..30V output of the second supply, you actually can create a +30 0 -30V supply, or a 0..60V supply. You need to adjust both voltage potmeters to set the values, but if you want to measure a circuit with a variable voltage, you need a tracking mechanism. This can also be called a master/slave combination.

The trick is to make the voltage setting of one supply depending on the setting of the other supply. I experimented with various ways, but finally settled on the following circuit.




Let me explain.
The slave supply must be modified as follows. The connection of the wiper of the voltage setting potmeter (P1) must be disconnected, and fed to a switch. The switch connects back to the old wiper connection as you can see in the schematic. The other side of the switch goes to a voltage divider that sits between the positive output of the master supply and a resistor combination connected to the 0V of the slave supply.
To connect the two supplies together, the 0V of the master gets connected with a lead to the + output of the slave, and this becomes the new 0V. The above schematic should make that clear. If you want a 0..60V supply, the + is the + of the master, and the 0V is the 0V output of the slave.
The modification for the master is even easier. You need to add one resistor (R40) to the + output, and feed the other side to a connector such that it can be fed to the slave. As you can see on one of the pictures of my supplies in the beginning of this post, I originally used a 3-pole DIN connector to feed the 24V AC to the PSU. I have now switched to banana jacks, and have used the DIN connectors to tie the two together.

The trimpot R41 needs to be set such that the voltage setting on the master is the same as the voltage output on the slave. The signal going to the switch will be close to the reference voltage of 11V2.

I found that the best tracking accuracy can be obtained if both supplies are set to 30V in the +/- mode as in the schematic. You can then flip the switch to the Tracking mode, and you adjust R41 until the slave also reads 30V. You will notice that the tracking is pretty accurate (about 1%) until you go below 4-5V, it then gets increasingly out of sync to a few 100 mV at 1V. This must be due to the linearity difference in the gain of both the U2 op amps. All the other methods I tried were to eliminate this, but I did not succeed. On the other hand, this accuracy is good enough for me.

I have also added R43 as a security measure, to make sure the slave supply will not have an (undefined) output if the link between the sense resistor in the master is not connected to the slave or when the switch is moved from one position to the next.

You should also know that you need to set both current limits independently for both supplies, but if the master goes into current limit or constant current mode, the slave will follow suit, regardless of it's setting.

Enjoy!

If you like what you see, please support me by buying me a coffee: https://www.buymeacoffee.com/M9ouLVXBdw

Posted by paulv at 11:22 AM

Labels: T&M

## 131 comments:

**riguera said...**

Really interesting hacks. I am thinking about buying the kit, but I would like to use ATX PC power supply directly (+12-0-12) and get rid of the bridge diodes, 3300uF capacitor & resitor. Do you think it will work?

Thanks!

February 19, 2016 at 12:19 AM

**CLignos said...**

*This comment has been removed by the author.*

March 17, 2016 at 7:35 PM

**blogxena said...**

Hy .
Bought two kits from Aliexpress one works well ! !
The second raises not go 12 volts. . .
A help please. .

Thanks!

March 17, 2016 at 7:38 PM

**blogxena said...**

I found fixes the problem. .
27 KOhm 27O KOhm Placing WRONG. .

March 18, 2016 at 1:06 AM

**Unknown said...**

Very interesting I bought 3 kits and will implement your modifications. Thanks a lot for providing all the details and pictures!

July 23, 2016 at 8:39 PM

**VA2AKG Caglar Akgungor said...**

Hi Paul,

Very isteresting blog and very useful explanations regarding this PS circuit, thanks for bothering. I have read all your successive comments, but I'm a bit confused. I want to use this supply as "advertised", 0-30V, 0-3 Amp but with the minimum noise possible. Which of the modifications you've done are absolutely necessary to get a decent lab supply? I'll be glad if you could give some more indications.

Best regards,

Caglar

December 7, 2016 at 5:32 PM

**Unknown said...**

Hi, It is possible use this project in 50V x 10A? Many modifications? Very nice site! Thanks!

Barone

January 4, 2017 at 2:07 AM

**paulv said...**

Hi Eneas,
Thanks for the compliment.
I would not use this design, it lacks the capability to address 50V, the maximum is really 30V due to the way the op-amps are used. You would normally use a reference voltage and voltage/current amps on a floating supply above the main supply, like H-P etc. have done that for years.

Besides, 10A will be taxing any kind of design, think about the power consumption across the series transistor at 10V 10A (say 60V raw - 10V out is 50V at 10A is 500W. You would most likely need tap-switching for the transformer to reduce the voltage.

As you can see this is not a trivial exercise, and this design is totally inadequate for that task. 30V 5A is possible, but again, I would not use this design for that either.

Good luck,
paulv

January 4, 2017 at 9:41 AM

**paulv said...**

*This comment has been removed by the author.*

January 4, 2017 at 9:57 AM

**paulv said...**

Sorry for the late reply Caglar, for some reason, I did not get an email from Google that you posted something, and I was too busy to check manually.

It depends on what you really mean with the minimum noise. The design itself was not for low noise. The noise from the zener is amplified, and zeners are noisy to begin with. You would most certainly need to redesign the reference circuit, or make that as low noise as possible. I like to think I went as far as possible with this design.

If you use a DMM as I do, go for an electrically separated supply. These meters are very, very noisy. You can purchase isolated DC-DC chips that will do that for you. Alternatively, the traditional method is to use a separate (little print) transformer 6-9VAC and use that to power the meter.

January 4, 2017 at 11:32 AM

**Unknown said...**

Hi Paul,

Many thanks for your help and attention.

I totally agree with you, it would be impracticable to make a modification using this project.

Again I will abuse your patience and ask if you know any project that provides me 0-50V and 0-10A?

I know one from Popular Electronics, I believe 1986, it works, but I would like something more modern.

In the meantime I'm drawing your last version on a pcb, in Altium, for a test.

Thank you!

Eneas

January 4, 2017 at 5:17 PM

---

**paulv said...**

I don't know any DIY project off hand for those specifications, but I'm sure that when you use Google, you will find. Otherwise, I highly recommend you look at the service manual from one of the Agilent (H-P) Power Supplies.
Good luck!

January 4, 2017 at 9:10 PM

---

**VA2AKG Caglar Akgungor said...**

Hello Paul,

Thank you very much for your reply. I'm a ham, and mostly deal with RF circuits in the HF-VHF range. Seems like this can be a good general-purpose supply but nothing more. Yet I'll give try, and will use a separate supply for the V/A meter.

Best regards,

Caglar

January 12, 2017 at 6:36 PM

---

**daguero said...**

Good morning
Is it possible to use a simple 12V transformer with no midpoint instead of the 24V transformer ?
I just need 9V of regulated output...

January 30, 2017 at 10:12 AM

---

**paulv said...**

Hi daguero,

I would think so, but you may have to replace the Zener (D8) with a lower value to keep the voltage adjustment in range. Right now, the reference is 10V based on the multiplication of U1, and this is then again multiplied by U2 to get 30V at the output. Give it a try, you won't break anything by applying a lower voltage.

If you use a DMM display, you may have to change that power circuit. Most of these LCD's are happy with 5V and above.

Good luck!

January 30, 2017 at 10:24 AM

---

**Unknown said...**

Good day.... Please can I get a calculative breakdown of this circuit? I know u might have a rough work somewhere.... I won't mind if it is snapped... Would be very grateful email is wrarewhoweare63@gmail.com

March 18, 2017 at 8:47 AM

---

**Unknown said...**

wearewhoweare@gmail.com I would be vary grateful is I am considered

March 18, 2017 at 8:49 AM

---

**Reeko said...**

Hi Paulv, I plan to make this circuit based on your latest revision 6, I need your advise on the following:
1. Is it ok if I replace the Q4, Q5 with 2N3055 transistor which is easily available for me.
2. U1, U2, U3, U5 replace with MC34071 OpAmp, as TLE2141 is more expensive for me.
3. Can I used a 30V-0-30V transformer as the input AC source.

Really appreciate your advised, thank you.
Reeko
Manila, Philippines

March 18, 2017 at 11:19 AM

---

**Unknown said...**

Has anyone been able to simulate this circuit? If yes p
lease I would be happy if you can give me the link to download your file thanks alot

March 18, 2017 at 6:48 PM

---

**paulv said...**

Here is a quick recap of the private messages I sent back to the last three posters.
I did not design this circuit, so I did not make a "calculative breakdown", nor did I simulate it.

The original design has a lot of followers and discussions on various "Boards", I suggest you find and follow them.
The advice is that should not use 2N3055 transistors, they are not fast enough.

Because of the design, you must use opamps that can handle to total (+ and -) supply voltage.
Make sure you check that! This design is at the limits of the raw voltage (see opamp supply voltage in the previous sentence), so an AC 30-0-30 will most likely blow everything up. (do the calculation from ac to dc)

It's best to stick to the original design and components, unless you know what you're doing.

March 20, 2017 at 9:04 AM

---

**Unknown said...**

Thanks again

March 20, 2017 at 3:02 PM

---

**Reeko said...**

Thank you very much for the advise, I will stick to 24V transformer, maybe I am just abit confused about the +42Vraw input in your Rev.6 schematics.

March 20, 2017 at 11:31 PM

---

**sladepointbiker said...**

Hello Paul,
I bought one of these kits online and found your blog while looking for a schematic. Your tuning instructions are very persuasive and I think I will implement most of them. I reckon I will double up on the final 2SD1047 but my trustworthy local supplier (RS Components) no longer has the 2SD1047 {"obsolete"). The nearest equivalent is on backorder! Excuse me if you have answered this before but is there a substitute you have tried yourself and can recommend? I'd like to have a few spares on hand. Plenty of 2SD1047 on Ebay but I wonder how good they are?

March 21, 2017 at 4:30 PM

---

**paulv said...**

Hi sladepointbiker,

I do not have experience with other transistors myself. I also use a double 2SD1047. I got a couple from eBay and they work fine. If you buy them from a reputable supplier that has a lot of positive marks, the risk is pretty small.

Success!

March 21, 2017 at 8:33 PM

---

**sladepointbiker said...**

Hi Paul,
You were quick! Thanks for the reply! 2SD1047 it is, from Ebay Deutschland, cheap, by tracked Airmail. Should be okay. Have the TLE2141's ordered too. I can take my time with this as it's only meant to be an extra current limited bench psu.

March 22, 2017 at 6:44 AM

**Robert Philpot** said...

*This comment has been removed by the author.*

March 28, 2017 at 8:25 PM

**Robert Philpot** said...

I bought two of these kits and the first one failed miserably _ unusual for me as I have worked in electronics for 50 years and check each component before installation. The second kit worked first time. Then I looked again at the first kit and the smaller diodes marking were reversed on the PCB! Yes one board has the small diodes marked backwards neg shown as pos and vice versa. This should be noted if buying the kit as there is obviously a dud batch of pcbs out there!

March 28, 2017 at 8:27 PM

**paulv** said...

Hi Robert,

What can I say. Considering where these kits come from, and the price, we should not be surprised. In one of the two kits I ordered, they delivered a PNP instead of an NPN power transistor. That also took me a little while to figure out.

March 28, 2017 at 8:46 PM

**Unknown** said...

Use TIP3055 in place of 2SD1047 15 Amp and no problem!

April 5, 2017 at 7:54 PM

**Νασος Παπαστυλιανού** said...

Congratulations for this excellent work!!!
Can I use LT1007 or LT1037 instead of TLE2141?

April 10, 2017 at 9:36 PM

**paulv** said...

Before you decide to use other opamps, you need to look at the total supply voltage of the opamps in this design, and also consider that the raw rectified voltage can be quite a bit higher when there is no load at the output. The data sheets will tell you what the maximum supply voltage is for each of these chips.

April 11, 2017 at 9:33 AM

**Unknown** said...

Paul,

Nice work, a final design far from the starting point. For my needs the standard kit seems adequate ..... except, like you, I have brought the dual meter to display output. A minimum modification to integrate it well would seem to be to lift the true ground side of R7 and tack the current sense shunt of the meter between the vacated pad and the freed end of R7. Do you see any problems with that?

Like the use of the fan feed as independant psu for the meter too .... and I might break my minimum mod rule to add that nice converter between CC trip point and meter display. But astonished that the meter injects so much noise, but hey, you have been there and measured.

Thanks for making the life of some other so much easier

April 15, 2017 at 1:51 AM

**paulv** said...

Hi Unknown...

You can indeed connect the V/A meter that way (I initially did the same), just be aware that the Amps and Volts meters in these kind of displays use a common ground, so the Volts accuracy will be compromised by the amount of current flowing through R7.

These meters are notorious for the amount of switching noise they inject back into the power supply. For the optimum results it is better to make this meter floating by using a separate supply without a common ground.

Success!

April 15, 2017 at 10:03 AM

---

**Unknown said...**

Hi thanks again for the excellent job done.... Please I'm wondering can I power this circuit with a 24V 6A transformer? What would be the effect of the bridge diode

April 21, 2017 at 9:06 PM

---

**paulv said...**

I do not recommend this particular design for much more than 3 Amps and the 24VAC will generate too high of a raw voltage for this circuit to handle.

Please do not forget that the basic design is extremely simple, and only works well within pretty tight specifications. If you doubt that, have a look at the schematics of a professional lab supply.

April 21, 2017 at 9:41 PM

---

**P.Kroon said...**

Hi Paulv,
Aren't the OpAmps running in too high voltage? According to the spec sheets the TL081 should have max +/- 15V, and the TLE2141 should have no more than +/- 22V. Either way, when using a 24AC transformer, there will be about 24V*1.4=34V DC. That is much more than then allowed max of the OpAmps.

May 7, 2017 at 4:36 PM

---

**paulv said...**

The TLE2141 OpAmps have been especially selected for this application, because of this particular design.
You are correct, the TLE2141 has a maximum supply of +/- 22V, but that means that the total max supply voltage between + and - is 22+22=44V in total, well below the maximum voltage of the rails in this design. That is the reason you cannot replace this chip with just any OpAmp.

May 8, 2017 at 8:28 AM

---

**Unknown said...**

Hi Paul.

My R7 resistor keeps dying. Can you please tell me how I can solve it?
I'm using the same Chinese kit without modifications.

Thanks

June 6, 2017 at 4:34 PM

---

**Unknown said...**

Actually, it's not dead, whats happening is, almost no current is flowing through this. Somehow current is getting to the output. if current was flowing through R7, it should be hot, but it's not heating up at all. and I'm getting up to 9 amps on the output! CL is working, but the maximum CL is 9 amps. I checked R18, R17 and P2. They all work fine.

June 6, 2017 at 6:30 PM

---

**Unknown said...**

I found the problem. It was the front panel volt amp meter. there was a ground loop. I should've read this article properly.

But now that the ground supply lead for the display is no longer used. The ground for the display unit is coming from the shunt connection to the raw supply ground. But this is causing a small current draw on the power supply's output and causing the power supply to go to constant current mod when P2 is turned completely counter clockwise. Any solution?

June 7, 2017 at 1:28 PM

---

**Unknown said...**

Hello Paul,

I've been reading your Blog about this PSU extensively and I must say It looks very thought out! ;)
I do have some questions for you regarding the modifications you made. I'm looking at Revision 5 as this will certainly fulfill my needs. I don't need +30V to -30V.

- You say "If you want to be precise, you can still use the original R18 value of 56K, but add a trimmer of 200K or 250K in series. This trimmer can be mounted on P2, so you don't have to mess with the PCB." Don't we still need P2 for adjusting the current? And if the answer is yes, where do I install RV3

- You say "Because we will not use the TL072, we can drop Q1, R13 and R14." Looking at the schematic (rev. 5) Q1 isn't there anymore, but R13 and R14 are. They have different value's then the original resistors, but do they still need to be soldered on the original location?

- You say "We do that by replacing R3 with an LM337 voltage regulator (U6), and we set the output level at -1.3V with two additional resistors, R25 and R26. We'll also add a small filter capacitor, C14 of about 22uF/10V." And a bit further on you continue with "The LM337 will be mounted in place of R3, and I just figured out a way to make the connections to the ADJ pin and R25 and R26 to connections that are near." Where does one solder the ADJ pin of the LM337 and where are R25 and R26 soldered?

- You say "So, to tackle these problems, I went back to using the LM7824 that was part of the kit, and used that instead of D10, the 10V zener that was used to create the supply to U3, U5 and Q3.
To counter the noise injection from the display, I now used D10 to reduce the raw supply and used that to power the display unit." Do I install the LM7824 on the pins from D10 and if so can I solder the GND pin to the negative side of the OUTPUT? And where do I put the D10 now used for the display?

- You say "I also installed the "optional" trimmers to set the maximum output voltage (RV2) and maximum output current (RV3). It is important to set the maximum current limit, because the granularity of P2, a normal pot meter, is greatly increased allowing you to set the current level more precise." Do I need to use RV2 if I have a 10 turn 10K potmeter?

- And lastly, In the schematic there is an additional capacitor C12 which is placed in parallel with C1. Can I solder this on the AC24V pins?

As you already must have noticed I'm fairly new to the whole design you own PSU scene, but I'm eager to learn. I just need something by which I can connect the dots (ie, the schematic vs the post you made and the photo's that are already there. Maybe it is possible for you to make a couple of detailed photo's and mark these (as you did with the PCBlayout.jpg) with which part goes where? If that's not to much trouble?

I'm eagerly awaiting your answer(s). :)

With kind regards,

Tom

June 19, 2017 at 1:10 PM

---

**paulv said...**

Dear Tom,

Nice of you to like my work, but I hope you understand that I cannot help all people that are interested in this supply.
I wrote this as a helpful guide for others, but it is not a step-by-step instructional description. If you can't follow what I describe, then maybe this supply is not the right one for you.

There are many, many postings on several blogs of people that have made comments on this particular design, I'm sure you can get more information there too.

Sorry I can't help you further, my time is limited, and I'm currently working on other projects in the little time I have.

Paul
June 20, 2017 at 9:50 AM

---

**Unknown said...**

Paul,

No problem at all. I'll figure it out myself as I usually do. I already committed to buying all the extra parts you described in your blog. Just have to wait until everything arrives and then start playing around with it and connect the dots.

Have fun with your current project

Regards,

Tom

June 21, 2017 at 12:00 AM

---

**jose said...**

*This comment has been removed by a blog administrator.*

July 18, 2017 at 1:18 AM

---

**Unknown said...**

Hello Paul,

It's been a while since my latest comment on your Blog. Again, nothing but kudoz for you ;)

I've been busy building the psu according to your alterations and except a couple of parts I'm still waiting on I've finished a lot and think I've figured it out.

Here are a couple of pictures (I hope I can put them here in the comments).

[img]https://farm5.staticflickr.com/4309/36111238575_4c84e7e51b_k.jpg[/img]

[img]https://farm5.staticflickr.com/4327/35271066104_0b6ce041eb_k.jpg[/img]

[img]https://farm5.staticflickr.com/4298/36111233085_f468263994_k.jpg[/img]

[img]https://farm5.staticflickr.com/4295/35977564761_2a73653e12_k.jpg[/img]

Regards,

Tom

July 23, 2017 at 6:38 PM

---

**paulv said...**

Well done Tom,
I suggest you double check the proper power transistors and the Zener diode used for the reference. In the bag of parts that I got, there were mistakes. As an example, Q4 was a PNP, instead of an NPN type, that will ruin your day. The delivered Zener required a much higher bias current, so make sure you have the right type.

Good luck!

paul

July 24, 2017 at 7:44 AM

---

**mars said...**

hi paul
how is your psu doing have you made any modifications lately

July 31, 2017 at 11:53 PM

**paulv** said...

Hi mars,

The supplies (I have two) are doing fine, I still use them.
I did not see the need to make any modifications to them.

However, as you can see from the beginning of my post, I'm in the process of designing another, hopefully much better, supply that will replace these two.

You may want to look that over before you decide to invest.

August 1, 2017 at 9:07 AM

---

**bhishmar** said...

Hi Paul, Thanks for your blog. It was a learning experience.
I have a question on the need to provide -1.3v to the –ve supply pin-4 of U2 (Vol-control-Amp), instead of simply Bridge (-ve) ground. Let me quote your own blog statement below :-

"Despite what others have posted, I had to connect the minus supply of U2 to the negative supply, not to ground. The reason was that I could not get the output to go to 0 Volt with P1. It did go to 0V with the current limiter. "

I followed the discussion in electronics-lab forum, to some detail. The circuit consensus by AudioGuru/ Redwire/ & Liquibyte, after many discussions have U2's pin-4 tied to bridge-ground (-ve) only.

I want to know whether the problem described by you was due to using older TL081 opamps, if so understandable, since the of TL081 does not swing all way to –ve supply ( some 3 volts above), which explains U2 output not reaching 0v. (both at input & output)

Or are you saying you had this problem, even with TLE2141 opamps, which could easily swing to –ve rail.
Are you willing to clarify your experiences further on this particular point: the real need to put -1.3 v there?
Rgds bhishmar

August 30, 2017 at 8:46 PM

---

**paulv** said...

Hi Bhishmar,
The output of the Opamp (and I used the TLE2141) has to overcome the approx. 0.7V of the driver transistor to let the output go to truly zero volt. I'm afraid this is inherent to this particular design. If others can get away with it, so be it, but I couldn't.

August 30, 2017 at 9:01 PM

---

**bhishmar** said...

Hi Paul,
Thanks for your quick reply.
I know you are busy with other things. But let me beg your indulgence.

Your reply statement (I have expanded it) below:-

"The output of the TLE2141-Opamp (U2) has to overcome the approx. 0.7V of the driver transistor (Q2), to let the PS-output go to truly zero volt. I'm afraid this is inherent to this particular design."

I beg to disagree here.
U2's output has to overcome +0.7v, NOT for the PS-output to dip to 0v, BUT for it to raise above 0v. i.e For PS-output to raise upwards from zero, both Q2 & Q4 has to be switched on (I mean active). This requires a minimum +(2* 0.7v) at Q2-base (or U2-out) point. Practically this need be only 2*0.5v. Also R15 drop should be negligible.

Reciprocally to force PS-output to 0v, Q2+Q4 should be switched off. For this to happen, U2-output need not reach 0v, LET ALONE DIP below 0v (-ve side). i.e The moment U2-output dips to +2*0.5v, Q2+Q4 will be switched off.
The scenario described above is with output Caps C7/C10 discharged to zero. If they have any +ve voltage, switching off of Q2/Q4 happens even much earlier.

So this cannot be an argument for the problem you faced : PS-output not reaching 0v.

Dont get my criticism wrong, I am only trying to get to the bottom of the problem you faced, & try to learn something from the tremendous build, testing & analysis experience you had with this PS.

I am in the process of building one with the chinese kit you mentioned, & wanted to see whether I can customize better than you, with it, since my benchsupply failed on me.

Do you remember how low U2-output reached with P1 turned all the way down, when you tested? (with no -ve supply)
Your answer also indicated parametric differences (my guess), why others could get away with it (Non negative supply).
Would you care to comment?
rgds bhishmar

August 31, 2017 at 12:08 PM

---

**paulv said...**

bhishmar,
Sorry, I can't help you any further. I have ditched both my supplies based on this design because I was not happy with them, and I don't remember the specifics anymore.

Good luck with your efforts.

September 7, 2017 at 10:09 AM

---

**edpmasmdn said...**

would Sir share to me a good psu for 0-30V 5A schematic?tq sir

September 12, 2017 at 6:03 AM

---

**Bo Lefness said...**

Hi Paul,

based on your work I have decided to build my own PSU.
my idea is to go step-by-step
The first part is the rectifier and I already have a problem with it.
Let me describe the issue then ask help. Maybe others also can learn a bit from it.

well, parameters I want to achive
40Volt/2Amper DC which will be the "raw" of the PSU
Wen its load is 2 Amper the Voltage expeceted to drop to about 37 Volt
because of 2 x 1.2 Volt loss on diodes + about 0,6 volt "spikes" on condenser.
This 37 Volts would good like the raw voltage of my planned PSU.

I use a simple transformer -greatz bridge-and puffer condenser
Measured and selected values:
Transformer secunder voltage is 30 volt without load and 28 volt if load is about 2 Ampers
This is as expected since the resistance of secunder of transformer is about 1 Ohm,
so when load is 2 Ampers the loss is about that 2 volts what are lost on secunder's resistance.
OK.
I use 2 x 2 SY710/711 diode. its Uf at 2 ampers is about 1.2 volt
Then I choose 6800 uF condenser.

At this point my expectation was when no load there will be about 40 Volt (30x1,41) filtered voltage and
about 36.5 Volt (28x1,41-3) when load is 2 Ampers.

I have put togather things and measured.
For my big surprise the practice is not in harmony with the theory...

Without load everything seems to be OK, I get 40 Volt DC
But when load is 2 Ampers the voltage drops opposite 36,5 volt -as expected- to about 32 Volts
which is NOT ENOUGH for the PSU.

Why is there so "big" loss? Opposite expected 3-3,5 volts total to about 8 volts which ruins my plan.

My first idea was maybe there is problem with the capacity of my 6800 uF condenser.
So I measured its real capacity! I used t=RxC formula which says if I load a condenser trough R resistence
from Uo voltage then after t time voltage of condenser should be about 0,7 x Uo = 7 Volt

If we know R and measure t up untilUc will be 0,7xU0 then we can count the real capacity of condenser.
well I used 1 KOhm series with the condenser, used U0=10 Volt and measured the time until Uc raised from 0 up to 0,7 x U0 = 7 Volt
From this measurement I got for C which is about 5000 uF, so the capacity of condenser seems to be OK.
Then I have checked diodes and even changed them. Unfortunately result is same.

At this point I just can think for the reason of problem what is comming from the transformer...but
unfortunately I have no oscillsocope to check waveforms.

Do you have any idea what the problem can be, how would possible to localize it and even how would be possible to solve the issue.

To sum up the problem is:
I expect about 36,5 Volts filtered DC on the output of rectifier/puffer when load is 2 Ampers
In practice I just get 32 Volts.
If the load is just 1 Amper, everything is in harmony with expectation.

I'm inclined to that the problem is the transformer.
It is comming from an old EPSON FX800 printer, its core is about 10 cm2, and if I load it with 2 Ampers for long time it is not too hot.
SO it seems to be OK from the task.

Any help would be appreciated.
thanks

October 18, 2017 at 11:01 PM

---

**paulv said...**

Hi Bo,
This blog is not the right place for your questions. You really need to look for a suitable Forum to post your questions, such that many others can contribute.

Here is an example: http://www.electro-tech-online.com/threads/how-to-calculate-the-value-of-a-smoothing-capacitor.106374/

In case, designing a power supply without an oscilloscope will be a daunting task. If you had one, I'm pretty sure you would have seen that the filter capacitor is way too small. Whatever your DMM is, it cannot give you the full story.

Success!

October 19, 2017 at 8:11 AM

---

**Anon said...**

*This comment has been removed by a blog administrator.*

January 4, 2018 at 11:24 AM

---

**brass said...**

*This comment has been removed by the author.*

February 19, 2018 at 5:37 PM

---

**Amsnick said...**

Hi Paul,

Thank you for sharing all your work and thoughts. I printed off a picture of the pub with component values and numbers with a smile!!!

I'm here because I burned up my power supply. I dead shorted it to death. I know very little about electronics but am not totally out of the loop.

I can see that the regulation of the voltage is the part of the circuit that is in trouble. I was wondering if the numbered points on the schematic diagram would be paired somewhere so I could use them as test points. Also it would help if the pairs of test points might have published voltages or ranges I could expect. Could test point 5 be used as a reference?

Thanks for your thoughts.

Nick

April 7, 2018 at 12:15 PM

**paulv said...**

Hi Amsnick,

Test point 5 is not a test point but one of the connections from the PCB to (mostly) the front panel parts.

Having said that, that point is indeed the reference for the supply.

Enjoy!

April 7, 2018 at 7:18 PM

**Amsnick said...**

Thanks Paul,

I got two more power supplies today. I'll check one of them against the inop supply and see if the bad component turns up with my VOM.
Have a good.
Nick

April 9, 2018 at 11:38 PM

**Unknown said...**

Hi Paul

I like your mods, I have 'n problem were my Limiting light stay on, power supply work perfectly can change voltage and set current but light stays on.

Regards
Jannie

May 9, 2018 at 10:59 PM

**paulv said...**

Hi Jannie,
If your current limitation indeed works, then U3 is operating correctly, but there is something wrong with the circuit around Q3. Q3 only deals with driving the LED. Is Q3 indeed a PNP and correctly installed (are the collector and emitter switched?), and is the LED connected properly?

If you have a scope or DMM, look at the voltage level at the output of U3 and see if it indeed switches state when the current limiting gets activated. (a simple trick to do this is to set a low voltage of say 5V, turn the current limit potmeter about a quarter turn from fully anti-clockwise and then short the output to enter current limiting)

Then move to the Base of Q3 and you should see a level change there too. Then look at the Emitter of Q3 to see if that changes, and then the Anode of the LED. This should lead you to the wiring error, or the defective component.

Good hunting!

May 10, 2018 at 8:10 AM

**iDeeW said...**

Paul, Thanks for this nice post. Very helpful. I question for you, is there anyway I can hook up a switch mode power supply to this?
I understand the negative voltage generator should be there, but without interrupting all that circuitry can I plug a 24V SMPS to the final transistor?

May 24, 2018 at 11:30 PM

**Madhu Nuggehalli said...**

Hi Paul,

Any chance you could release the eagle files?

Thanks!

June 9, 2018 at 12:42 PM

---

**Ariel said...**

Hi, Paul, how are you? My English is bad so I use Google, and my question is, do you say above that zener make noise (with respect to the original circuit), so the use of LM regulators can be replaced with transistors to achieve the same stabilized voltage? Thanks for your response from Argentina, greetings Ariel.

August 3, 2018 at 7:47 PM

---

**Unknown said...**

To get -5V needed for the opamps, and having myself a toroidal for 1.5A, could I wind some wire on it, rectify and use a LM7905 with a couple of caps? I guess that would perform better than a zenner clamp?

November 8, 2018 at 4:26 PM

---

**paulv said...**

Hi Unknown, yes you could do that.

November 9, 2018 at 8:08 AM

---

**Glen&Jen said...**

Hi and thank you for this blog. It has made the short list of options to go with in building this supply that I bought and found out can be improved. The other option is from the discussion on the electronics-lab community (Rev 7). I was looking to build it with the full 0-30V 0-3A. The reason the Rev 7 option is appealing is the use of a 28V transformer which I have in my box of transformers now. My question is, can I use the 28V with your modifications or are the differences in either version enough that I should follow that instead of what you have posted here?
I really would like to use your version as the tracking option and use of the original PCB is what I would like to have/use as well.

December 24, 2018 at 8:01 PM

---

**paulv said...**

Hi Glen&Jen,
Yes, I think you can use the 28VAC transformer with my latest schematic. The main reason is that I use a regulated supply (LM317 set at 33V) to power the opamps, rather then the unregulated supply as in the original version. Make sure you read that section carefully.

Enjoy the build & good luck!
Paul

December 24, 2018 at 9:30 PM

---

**Ozzy said...**

Hi Paul,

Where are the Eagle files?

Thanks!

January 25, 2019 at 8:59 AM

---

**paulv said...**

Ozzy,
There are no Eagle files posted, they never were.
I dropped the use of Eagle several years ago, and because I no longer have my two supplies based on this design, I ditched all my Eagle stuff anyway, so I can't help you. Besides, I only used Eagle for my schematic capturing, never with the intent to create a board, so the files would have been pretty much useless.

January 26, 2019 at 10:32 AM

**Jamie** said...

Hi Paul
First off fantastic walk through, think I need to read it more than once tho, as I'm a beginner when it come to electronics, I brought the above board a while ago but iv been put off by reviews but after reading this I'm going for it, I have a 28vac transformer, but I have no idea about the amps. Would this work OK with your end design.
Thanks again

April 8, 2019 at 11:37 AM

**paulv** said...

Hi Jamie,
As long as you use the regulator in my design you're OK with this transformer. See previous comments above. How many Amps you can pull also depends on the output Voltage, you need to give that a try. Not many need a lot of Amps, as long as they can get 30V.
Success!

April 8, 2019 at 3:49 PM

**Fabius** said...

Hi there, I implemented the change from R3 to a LM337 and used C13 as specify but with the rest of the kit original (keeping the TL081´s)

When I remove power from the mains the output goes all the way up to 30V and then starts to fade slowly (i suspect there is a capacitor discharging through Q1 base-emitter)

I think is C13. I was considering putting 1N4148 diode in the same place as D7 originally is (and parallel to C13, in order to allow the current to go to GND through D7 instead of allowing C13 to discharge through Q1 when the charge pump collapses.

Is this idea correct?

April 14, 2019 at 9:52 PM

**paulv** said...

Hi Fabius,

I have not seen the effect you describe, so there may be something else going on. Unfortunately, I don't have these supplies anymore so I cannot do any testing for you.

I suspect there is something else going on but you have to be aware that the circuit around Q1 is far from perfect as an output protection.

The principle idea is that the negative supply will always decay much faster than the overall raw supply and as such clamp the output through Q1. This is without any significant load on the output. If there is, and the raw supply decades faster, the protection will not work.

Look at the section above where I describe the turn-on and off effects with the scope screens, and describe the solution. To circumvent any possible output effects, I use a switch (S1) to disable the output, before I take the mains away. Don't forget that this supply is a very, very simplistic design that has virtually no DUT protection.

Instead of the diode, you could also use a small resistor across C13, ensuring a faster decay.

Success,

Paul

April 15, 2019 at 9:27 AM

**Abhijit Dey** said...

Hi Paul, I am assembling the power supply (Rev 6.) I read all your comments and liked your recommendation. I did not understand the CL mode. What does that do? Can you please let me know the procedure of adjusting the various presets step by step? Can I power up wo connecting the ICs and check the voltages?

April 17, 2019 at 5:35 PM

**Fabius** said...

Thanks Paul for haven taken time to answer my question.

For now I have been using a 5W 220 resistor and a Led strip @12V 300ma. But for the example of the led strip there is a transient both at mains startup and fallout. You can notice this by a quick flash on the led strip.

I still didn´t implement the switch output function but will definitely do. I´m thinkering with the pcb until the enclosure box arrives from ebay.

Also added a F5A and a 5ohm NTC to protecto from inrush currents on the secondary side of the transformer.
Now I´m looking to make a short circuit protector for the output and making the pcb for the CC and CV tracking display

Thanks again!

April 17, 2019 at 10:30 PM

---

**Fabius** said...

Just a quick update regarding the raw and negative supply decay causing in mi case the erratic Q1 clamping circuit behavior.

Originally in my assembly I swapped the 7824 for a 7812 to power a 8x8 cm 12V fan rated 0.05A (directly, without current resistor, oops)

I removed the 7812 to assemble it in another location and continue testing the supply with the fan connected to an external power source.

Now when I remove power from the mains the clamping circuit around Q1 works as designed (still with is design flaws) but at least for my case works way better than before.

So it seems that the raw supply was indeed decaying faster than the negative thanks to the load caused by the 7812 + the fan.

Thanks Paul, I leave this info if anyone else has a similar problem. And people, keep Paul´s recommendation of using a switch to turn of the non inverting input of U2 to safely shut down the output of the PSU.

April 19, 2019 at 2:14 PM

---

**paulv** said...

Thanks for the update Fabius!

April 19, 2019 at 3:48 PM

---

**Abhijit Dey** said...

Hi Paul, I have assembled the modified circuit (Rev 5). I have noticed the following :

1. While doing the load testing with 10ohms 25watt heatsinked load. Noticed that the raw DC voltage dropped to 32 volts from 36 Vlots (at C1) with the voltage setting at 8.472 VDC and the current set to 0.80 Amps. Even the AC voltage in the secondary dropped by 1.5 volts with the same settinga mentioned above. The temp measured in the heatsink rose to 89.1 Degrees C with the enclosure closed. Is that normal. I have set the current limit to 1.5 amps. Will appreciate for you feedback at your earliest.

May 27, 2019 at 5:47 AM

---

**paulv** said...

Hi Abhijit,

What you are describing can be attributed to two possible causes.

One being your raw supply (transformer, diode bridge, filter capacitor) not being capable of delivering that amount of current. In that case, the problem is most likely with your transformer. To make sure, disconnect the series transistor(s) from the raw power supply and do the same measurement with only the load connected to the raw supply. It would be nice if you could measure the current through the load, and measure the voltage across it to verify that there is no issue.

The second cause, this is more likely, can be due to an incorrect series transistor circuit section. I cannot explain the drastic rise in the heatsink temp. The amount of Watts for the series transistor (32V - 8.5 = 23.5V) with the current of 0.8 Amps is less than 19 Watts. So something is not right at the output section of the series transistor circuit causing excessive currents to flow through it.

Make sure there is no "hidden" load on the output. If there is no significant load at the output, does the supply regulate normally?

Good hunting!

May 27, 2019 at 8:36 AM

**Abhijit Dey said...**

Thanks Paul. Let me do some inspection and revert back to you.
Cheers!!

May 28, 2019 at 2:11 PM

**Abhijit Dey said...**

Hi Paul, what is the max voltage expected out of pin 6 of U3? In any case I won't be using more than 1.5 amps at 30 VDC. So I am using single 2SD1047. But my toroidal tx is rated at 24 VAC @ 2.0 amps. I am using bridge rectifier 1000 Vlots @ 10 amps. And C1 as 15,000mfd at 50 VDC.

May 28, 2019 at 5:15 PM

**paulv said...**

With a 24VAC @ 2.0 Amps you will be struggling to get 1.5 Amps @ 30VDC. It will be more like 1 Amp @ 30 VDC, more current with less volts, as you can imagine. Look up the conversion factors for AC to DC voltage & current.

The theoretical max voltage at pin 6 of U3 is just below the rail voltage. I have no means of measuring it for you, I don't have these supplies anymore.

May 29, 2019 at 10:07 AM

**Abhijit Dey said...**

Thank you Paul for your quick reply. Regarding the voltage drop, I found out from the tx mfg that the drop is ok as far as the tx regulation goes. Mine is rated as 15%. I know the lower the value, the better it is. What is recommended as the min value. The input voltage taping is set to 240 VAC where the AC main voltage is 217 VAC. I will change the tapping and will let you know the result. Cheers!!

May 29, 2019 at 11:50 AM

**Tiredmg said...**

Excellent job.

June 10, 2019 at 1:32 PM

**UsamaAk said...**

*This comment has been removed by a blog administrator.*

July 15, 2019 at 9:54 AM

**Johnyp said...**

Any idea what should i do to use 32v AC 5a as input transformer? When i do the CC led is ON and no output voltage at all. Thank you

July 19, 2019 at 12:46 PM

**paulv said...**

Johnyp, you need to adjust the resistors that set the maximum output voltage and output current. Make sure you don't go above the VCC levels of the op-amps otherwise they'll blow up.

July 19, 2019 at 2:53 PM

---

**Maher said...**

Hi Paul,
Thanks for all advice. I made this PSU with yours upgrade. Only I dont use 7824 and made separate power board for display and fan. All working almost perfect, only I cant setup 0V to output and minimum value is 2.3V. When I tray change voltage with P1 will start change up after 3/4turn what corespond about 2volts. Can you help me, waht is theoretically wrong ?

November 18, 2019 at 2:17 PM

---

**Mick said...**

Hi Paul

Have you seen this project using the original 0-24 3A cheap PSU PCB and then uses a daughter board with LCD to give you over voltage & over current protection.
Works great using your recommendations
Let me know your thoughts?

https://www.danielrp.net/projects/proj-lab-power-supply

March 11, 2020 at 2:31 PM

---

**paulv said...**

Hi Mick,
Thank you for bringing this development to my attention. It looks like a very good addition for this supply, although the supply itself remains a very simple and with a very limited design. My advice is that if you consider this addition, get a better supply to begin with. This addition will work on just about any general purpose power supply.

March 12, 2020 at 8:44 AM

---

**Unknown said...**

Hi Paul, after making your changes we at the output of the power supply is +1.8 V. P1 is at minimum and RV1 cannot be set to 0 V

March 25, 2020 at 9:58 PM

---

**Unknown said...**

hey...Why only 24 v output and no more vith 22v AC. And why current led light on with 25v AC.

April 17, 2020 at 10:56 AM

---

**roland said...**

Most of the problems with the original circuit comes from the 36 volts supply limits of the tl 081.
Or for the new texas tl 081 ops the supply voltage is 42V according to the data sheet...

December 27, 2020 at 10:18 PM

---

**johanh said...**

Very informative blog post. Great work!

I'm building a similar PSU and noticed in your tracking circuit you probably have the 1M R43 resistor in the wrong place. As it is now in your schematic, it is just in parallel with R42 and doesn't really do anything. To help contact bouncing etc. and giving the input of the non-inverting op amp U2 a path to ground, it should be for instance attached to the same point where C16 is attached, right besides the S4 switch. Now in the middle of the switch throw, or if the master PSU is disconnected, there will still be a path to ground for the amplifier input.

January 30, 2021 at 7:06 PM

**paulv** said...

Hi Johanh, to be honest, I don't remember why I put the 1M at that location and I don't have the supplies anymore to check what I did in reality.
Sorry!

January 31, 2021 at 11:18 AM

**rupe01** said...

Easily done! Glad you got it!

March 23, 2021 at 4:06 PM

**Janux** said...

Hello Paul,
I had read your interesting article about the Chinese power supply also sold on Amazon for just 10 €.

Since it had been a while that I wanted to build a 0-30V and 0-6A linear power supply and having a 24V 150VA transformer in stock, I decided to buy it.

I realize that many years have passed since you worked on this project but I ask you for some information anyway.
Forgive me.

I have not yet made the changes you suggested in Rev.6 but on the prototype I assembled a strange fact happens that maybe it did not happen on yours. When the circuit is powered at 24Vac, the current limit LED lights up immediately and remains always on even when the current potentiometer is adjusted. Is this likely caused by the supercharged TL081s? The RAW voltage is about 33V and the negative one is -5 so we are at 2V above the maximum supported by the TL081s. By powering the circuit at 18Vac, on the other hand, everything works regularly.

I can't easily find the TLE1241s, someone on the net suggested using the MC34071 opamps, do you think I can use them instead of the TLE2141s in your Rev.6 schematic without modifying other components?

To drive this power supply, I am developing a controller with Arduino, with LCD 1602 display for the measurement of Volts (with a resistive divider 50 / 10K) and Ampere (with an ACS712) and the regulation of voltage and current using a double digital potentiometer MCP42010 which replaces the two 10K potentiometers of the power supply. I have also foreseen the measurement of the temperature of the heatsink which is displayed on the 1602 display, the same measurement is also used to start the fan at a certain temperature and if this continues to rise after the fan starts, an acoustic signal is emitted by a buzzer .

April 27, 2021 at 1:36 PM

**Janux** said...

Sorry,

...using two rotary encoders and a dual digital potentiometer MCP42010 ...

April 27, 2021 at 1:53 PM

**paulv** said...

Hi Janux, I'm afraid I have to disappoint you.
It's been too long since I worked with this supply... Two comments: From your description, it seems that you may have a problem with the negative supply, have a good look if that is indeed working properly.

You are embarking on a nice project, something I wanted to do myself but in the end, never did, and while I built a modern supply myself, after I learned a lot, I purchased a professional supply.

Keep in mind that the power supply where you base your development on is not very good. It's inexpensive and widely available but has a very simple and very basic design that is now decades old. Don't forget that.

April 27, 2021 at 3:57 PM

**paulv said...**

Oh, I forgot. The Opamp type is not very important, but the total rail-to-rail voltage is. This is one of the limitations of this design.

April 27, 2021 at 3:58 PM

**Janux said...**

Hello Paul,
Thanks for your very quick reply.

I don't think the problem of always on DC mode powering the 24Vac circuit, is due to the negative power supply, I measured it, it is exactly -5.1V. As expected, the 82 ohm resistor was very hot and I replaced it with a 2 Watt one.

Honestly, I don't really care about the specific badly designed PSU. I'm just using it as a "guinea pig" to test my Arduino controller, just for the simple fact that it has two 10K potentiometers for voltage and current limit. Then when the controller is final I will look for a better power supply scheme than that.

Indeed, while I am there I ask you if, in recent years, before buying the professional power supply you mentioned earlier, you have found a 0-30V, 6A power supply scheme, better than this badly copied and now sold as a kit. If in case you found something I would be grateful if you post the link.

Just out of curiosity, since I consider you an expert in this field: what power supply did you buy?

I had been eyeing the Siglent SPD3303X-E.
Currently sold in Italy for € 450.00 (approximately $ 540.00).

https://www.batterfly.com/shop/it/testandmeasurement/alimentatori/siglent-spd3303x-e

It is an investment but perhaps it is worth it.

Hello and thanks for the advice.

(sorry for the bad english but i translated with google).

April 27, 2021 at 8:07 PM

**paulv said...**

Hi Janux,
My own designed supply could be scaled up to this very high current. You''l have to find other DIY designs yourself.
This Siglent model is what I have myself, and I'm very happy with it.

April 28, 2021 at 9:09 AM

**Janux said...**

I have to apologize Paul, I didn't see you designed another power supply.

Thanks for sharing the project.

April 28, 2021 at 11:24 AM

**Unknown said...**

So can MC34071 opamps be used in this?,i see this has been asked a few times but you never give an anwer,why??

March 19, 2022 at 7:17 AM

**Unknown said...**

do MC34071 opamps work in place of the 741 opamps?

March 19, 2022 at 7:19 AM

**paulv said...**

I don't know anything about MC34071 Opamps. You would have to dig in the datasheets yourself.

March 19, 2022 at 10:13 AM

**Wandows said...**

According to the revision 6 diagram, what would be the maximum voltage of the transformer?

August 5, 2022 at 2:04 AM

**paulv said...**

Wandows,
Just above the diagram is information about the transformer I selected. There is also a lot of information on the original Blog about suitable transformers. The problem with this power supply design is that the raw voltage is also used to power the Opamps, and they have a limited rail maximum. This means that you can't get any lower but also not much higher than a 15-0-15 AC transformer.

August 5, 2022 at 10:14 AM

**Anonymous said...**

Hello Paulv, I have yet to read your entire blog, but I have purchased 3 of these kits so far and all three of them will only reach 25V max and the CC led is on continuously. Also the units will not respond when I try to adjust the 100k ohm tuning pot. I am using a 25V AC transformer as my input and I have thoroughly checked to make sure I assembled them correctly. Any help will be greatly appreciated!!

January 10, 2023 at 6:25 AM

**paulv said...**

This could very well be caused by an incorrect part, as I have had myself. They simply put a wrong transistor type in the bag. Carefully check all parts.

January 10, 2023 at 10:19 AM

**Anonymous said...**

Yea that’s what I was thinking but I checked them over and over, I was thinking maybe one of the components were bad or fake but on all 3 units though I don’t know about that lol, because they have all done the same thing

January 10, 2023 at 9:03 PM

**WiValdiBB said...**

Hi Paul, really appreciate Your work and time which You spent over this project, going to modify those as well
As You are in Holland, How far are You from 3881LJ Putten ?
I did some job as service with Bizerba, German made wieght scales, I have batch of trafos from dissasembled labelers and just want to give You some as a gift, if needed. Groetjes
https://www.fotosik.pl/zdjecie/afc2576bff0a6d01
https://www.fotosik.pl/zdjecie/a4c208c57c39e69

April 15, 2023 at 8:56 PM

**WiValdiBB said...**

What kind of opamps You used and how You made -Vcc? Finally how much it is ? Which circuit exactly You build a sis many variations, many modifications done

April 15, 2023 at 9:18 PM

**paulv said...**

Many thanks for the compliments and the offer! I'm just over an hour away from Putten by car, but when looking at the transformer, I have no need for one at the moment, but thanks for offering!

April 16, 2023 at 1:54 PM

**Anonymous said...**

I like the way you arranged the schematic in revision 6. it became immediately clear for me how this supply works. in the original schematic and the other it was not so easy to follow.
I exchanged the potentiometers with a MCU controlled PWM. To keep the gains in the circuit and reference voltage and stability untouched, i used a level converter CD4504 to lift the 3.3V of the PWMs of the MCU to the level of the voltage reference. It works good, the given duty cyles and the measured real values on the output of the supply are strictly linear for both current and voltage. That was one of the aims i wanted to achieve.
Second aim was to get the output current reading against GND with an ADC on the same shunt (R7 in the original design) as the supply circuit itself does. The voltage drop across the shunt is negative (referred to the output minus as GND). So i put an extra OPamp (inverting) over the shunt, with its V+ on Vref (11.2V) and its V- on the negative voltage. In this way the measured current on this OPamps output is converted to a positive voltage which is easily readable by the ADC on the MCU.
So despite the already given linearity between setpoints and output one can also read the real values with the MCU and display it on a LCD, TFT or 7segment-LEDs.
After that work one can control all the necessary things with the MCU, and can also use rotary encoders and USB or RS232 controls.
One can further sweep/ramp voltage or current in a defined way and can even produce more complicated waveforms or sine curves in the low kHz range. with lots of power, but i haven't tried to use it as an audio amplifier for now :-)

One question i have concerning the gain of the voltage amplifier, which is roughly given by R12/R11. I changed the zener diode on the reference opamp to 2.4V in a first attempt to bring down the control voltages on the potentiometers to values close to typical MCU supply values (3.3V - 5V). So the voltage chooseable on the V-poti-tap was not 10V anymore, but max. 3.3V. To still get 30V at the output, i reduced the R11 (27kOhm) to 6.8kOhm, which also works. So it's all fine with this, but: Is the stability of the supply now worse because of the increase in gain of the output stage? I haven't measured it, but in principle it maybe be oscillating earlier in special circumstances...

April 17, 2023 at 12:07 AM

---

**paulv said...**

Hi Anonymous (;-)) Good work with the changes! I don't think the changes in the gain will adversely effect the supply stability. Unfortunately, I abondoned this supply years ago so I can't verify your changes. I'm afraid you're on your own, but since this is not a great design to start with, there is little you can do wrong to make it worse.

April 17, 2023 at 11:12 AM

---

**WiValdiBB said...**

But If in need then write me
I can give You few of them for absolutely free.
I'm in Poland right now but I have sometimes Dutch guests here :)
Or I'm going to NL sometimes too, its so easy now, better roads, no brders, no passports needed, fuel could be some cheaper :) but no comparable to begin of 90s :)
Parameters are shown on attached photos
https://www.fotosik.pl/zdjecie/afc2576bff0a6d01
https://www.fotosik.pl/zdjecie/a4c208c57c39e69
I were working long time with electronic assmbly by R&R Electronics at Walwijk :) Groetjes

April 18, 2023 at 7:47 AM

---

**paulv said...**

Thank you I will. Transformers with these qualities and different windings are hard to find and not cheap. When I need one, I will contact you.

April 18, 2023 at 9:34 AM

---

**Anonymous said...**

Hi Paul, thanks for the answer. So one has the choice to use Vref=11.2V or go to lower lower voltages, that's good for people who want to feed the PWM directly from a controller.

>There is little you can do wrong to make it worse
:-)) i read your opinion concerning this supply before and i saw your detailed investigation and development stages to build a better one.
I think it's a good, cheap project to learn something about current limiting and how it works.
As mentioned above the schematic you provided is really great to support the understanding.

April 19, 2023 at 10:48 PM

**WiValdiBB said...**

Paul, I have a simply for You question: Why designed negatove power supplley for TL081 OpAmp is 5,6V and with TLE2141 1,3V ? My chinese kits has completely no marks on them so I dont know whta they completed, what I dont like
I must order other OpAmps

April 21, 2023 at 8:46 PM

---

**paulv said...**

The negative voltage rail level is not so important as long as it is at least -1V. It is there so you can adjust the output of the supply to zero Volt.

April 22, 2023 at 9:27 AM

---

**WiValdiBB said...**

OK, Thank You for an explanation, understand that .
My question were knocking my head as I thought as due to negative voltage differencies must be any min paramerter of each OpAmp type
Negative voltage closer to zero is better as we have higher max voltage to use as all voltage range goes up
Why then someone designed with 5V6 Zener diode?
Maybe specifically for 741 which were used in first design which I found
741 has Recommended Operating Conditions Supply voltage VCC– -5 to -15V
Many poples call this PSU as a Greek design, why? Where I can find original Greek design ?
The oldest design which I found is Practical Electronics 10/1978
https://worldradiohistory.com/UK/Practical-Electronics/70s/Practical-Electronics-1978-10.pdf

April 22, 2023 at 11:27 AM

---

**johanh said...**

There are many designs using this concept. In the past projects were not so easily shared, because language barriers. So you can find it in various electronic magazines from the time, for instance a Greek design and I think also an eastern European has been mentioned, was it maybe Czech. The oldest one shown so far is the one in the 1978 Practical Electronics magazine.

April 22, 2023 at 1:07 PM

---

**WiValdiBB said...**

Yes, I have also Czech edition :) Zdroj G400 - ELEKTROinzert 6-7/96 http://paja-trb.cz/konstrukce/zdroj/zdroj_G400.pdf
from http://paja-trb.cz/konstrukce/zdroj.html

April 22, 2023 at 2:56 PM

---

**Anonymous said...**

Interesting how that Early 70's circuit, and it's component values appear in a lot of recent circuits, How much is copied vs how much is worked out?.!

June 20, 2023 at 6:19 AM

---

**Anonymous said...**

Hi everybody,
I improved by digitizing the chinese circuit and replaced the final stage with that of the equally famous K7200.
Link to the schematic:
https://devincentiis.it/ADVe231/DeVincentiisLinearPSU.pdf
firmware:
https://devincentiis.it/ADVe231/ADVe231firmware.zip
video:
https://devincentiis.it/ADVe231/ADVe231DigitalLinearPS-vid01.mp4

November 2, 2023 at 9:41 PM

---

**Anonymous said...**

new links:
https://devincentiis.it/ADVe/DeVincentiisLinearPSU.pdf
firmware:
https://devincentiis.it/ADVe/ADVe231firmware.zip
video:
https://devincentiis.it/ADVe/ADVe231DigitalLinearPS-vid01.mp4

November 2, 2023 at 9:44 PM

**aero said...**

hi paul!! very interesting observations you made on the power supply kit regarding o/p overshoot and undershoot..i bought the kit just few days back and after coming across your blog i felt hope less... regarding full 3amp capability and overshoots on power on... By the way can you suggest me a +/-24V dc fixed supply using LM317 ,Lm337 ,2n3055 and MJ2955 transistors with overload and short circuit protection scheme. Thanks very much.

January 28, 2024 at 8:21 AM

**Anonymous said...**

The schematic it's wrong, Paul isent understand noting only write and put pic.

February 13, 2024 at 8:25 PM

Post a Comment

Newer Post Home Older Post

Subscribe to: Post Comments (Atom)