I have created 4 sheets of schematic that is supposed to represent Paul's original design. They are the:

- Power Supply
- Triangle Generator
- The Step Generator
- The XY-Amp and DUT sockets
- There is also a 5^{th} mechanical sheet with no electrical components on them.

This set of schematics have not been checked much by anyone else for "correctness". I ask anyone who has some time to see if I made any mistakes translating the schematics for the Blog site to those shown in the pdf document. I used Altium to create these schematics. I will make any changes to them if mistakes are discovered.

NOTE: I have made some purposeful changes. These are described below.

This rest of this document describes the differences in my schematics and Paul V's original schematics that he published. There are a few differences for several reasons. The reasons are:

1. Surface mount devices are used and some different components are necessary as the through hole components were not available in surface mount or there was a clearly better surface mount option.
2. A few circuits were modified to make the load on the op-amps easier.
3. All of the op-amps have changed. You can in fact use the original op-amps if you can find them in surface mount.
4. I assumed that whoever makes this will probably have to buy all or most of the components so I selected components that are readily available.
5. The power supply uses transformers that are 120/240 compatible. All the secondary's are dual. I assumed you wouldn't have an appropriate transformer around the lab like Paul did.
6. I added bypass capacitors to all integrated circuits.
7. The connectors on Paul's schematic did not have the same number for plug and socket.

Global changes:

This is a 2 PCB design. There are 2 sets of schematics because of this. I will create a 3^{rd} set with the entire circuitry of both boards shown to make it easier to see what is going on with the electronics.

One PCB is for the front panel components like rotary switches, toggle switches and pots. There is some circuitry on the front panel as well. It made more sense to have some circuitry close to the front panel components.

The second PCB is the main PCB. It has everything that the front panel does not.

There are 2, 3 and 4 pin connectors on the front panel and main PCB. They are used to connect the breaks in the circuitry between the front panel PCB and the main PCB. They have the same reference numbers on each board. So for example, P1 on the main board goes to P1 on the front panel board. You do not have to use connectors but if you want to they are included in the BOM.

The schematics combine some of Paul's versions from the Blog

The part reference numbers are all different than Paul's. I have used a scheme where the sheet of each schematic starts with a certain reference number. This helps locate components when comparing the schematic to the PCB. I also have modified connector references so they are consistent with the connectors from sheet to sheet.

Only the connectors share reference numbers. The 2 separate PCB's don't use the same reference numbers. For example there is only 1 R1 on 1 of the PCB's There is no R1 on the other PCB.

In general all integrated circuits have bypass capacitors on the power supply pins. I won't mention this again.

All of the op-amps are rail to rail output. Most are rail to rail input.

I used two ground symbols with different names for the Isolated ground and The schematic should not have changed for this.

I added test points sometimes with a 1K resistor to prevent accidental shorting of signal when the ground lead of the probe "accidently" drags across the PCB.

Some of the op-amps are single op-amps. This was done because the devision of the circuit for the front panel made sense to split up the op-amps based on the circuit and where it needed to be sent to the other PCB.

WANING: I selected particular 1Form and 2FORM relays. Use the ones selected as the pinouts of 1FORM and 2FORM relays can differ by manufacturer.

All switches on the front panel have a rectangular slots for the pins to be soldered directly to the front panel PCB. The rotary switches have round hole to solder them into the PCB.

Particular changes described in detail by schematic page:

Power Supply: This schematic sheet starts with reference numbers 1.

I used dual primary and dual secondary transformers that are PCB mountable. This was an obvious choice considering this will probably be built by people in various countries with different power systems. I choose secondary voltages that allow full wave rectification as I believe full wave yields a better DC than half wave. The transformers are available worldwide so this shouldn't be an issue.

The High voltage power supply could have been made as a switching secondary (series to parallel and back) giving more current capability as Paul considered but I choose to switch between 1 of 2 DC voltages as Paul actually made his. This is because switching between series and parallel requires the capacitor to change voltage every time there is a change in power supply DC at 30V. This can take a too long in some cases.

I used a 2 capacitor 1 resistor transformer compensator as the Qusimoto article recommended. The values of the C and R are not correct yet right now.

The 8mA current sink is made from a PNP transistor with a guaranteed gain of 300. The PNP darlingtom is not available in surface mount. I went to a Wilson current mirror because of the lower gain transistor.

The 2-NPN (2N3904) compound transistor used to drive the relay coils has 2 resistors in it to guarantee there is enough current flowing in the first transistor to keep the gain high.

I split the DPDT relay (Current Sense resistor and CS voltage source) into 2-SPDT relays to make layout easier and help signal integrity.

I added diodes to the regulators to alleviate current flowing backwards through the regulator chip at turn off.

Triangle Generator: This schematic sheet starts with reference numbers 50. I added a second transistor to supply the voltage to the device under test. There was room and the devices don't have to dissipate as much power. You do not have to use 2. Leave one out and short the emitter resistor of the remaining transistor.

Step Generator: This schematic sheet starts with reference numbers 100.

I choose rotary switches that can mount on the PCB. These are the same rotary swithes used on the German ELV KS7000. The 12 position rotary switches come with a ring to limit the number of positions from 1 to 12 so the PCB can accept 12 position switches in both locations.

I added 2 diodes and 4 resistors to the current buffer transistors that drive the step resistors. This is done to make it easier for the op-amp to drive the buffer. If you don't want to do this just short the diode and emitter resistor locations and leave the 10K (R101 and R116) resistors out. This gives Paul's original design. I also added some larger capacitors at the plus and minus power supplies feeding the transistor buffer.

I used 1% 10K and 20K resistors in the R2R network instead of the SIP's that Paul used. They are close together on the PCB so there shouldn't be a significant temperature change across the resistors.

XYAmp: This schematic sheet starts with reference numbers 125. There are no significant changes for this circuitry.

Mark Allie 1 Feb 2019