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

This set of schematics have not been checked 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:**

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.

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 various ground symbols with different names for the Isolated ground, the device common and CS- power supply. The schematic should not have changed for this.

All of the components on the front panel have a green rectangle around them on the schematic pages.

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.

**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 set of connections on the PCB usually in sets of three. In particular the 12PDT switch has 4 sets of 3 connection points. They are setup as 4 independent DPDT switches as drawn in the schematics. Please note that there are alot of similar connection points on the 4PDT switch so 12 wires are not needed if you short some of the terminals on the switch itself together. These 3 pad connection points will accept a 3 pin connector if you wish to use connectors instead of solder wires. They will accommodate 2.5 mm or 0.1 inch pin spacing connectors. The switch pads on the PCB are marked with the node of each pin to make it easier to see what pins get connected to what switch pins.

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. No substantive changes in this schematic.

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

I choose rotary switches that can mount on the PCB. They are not cheap. About $20 US. You can just use wires to the pads of the switches on the PCB to use much less expensive panel mount rotary switches.

Since only 10 position and 12 position switches are readily available as PCB mountable I have a 10 position rotary switch instead of the 7 position that Paul used. In this case the last 3 positions are duplicates of position seven. If you use a panel mount switch for the seven position rotary switch then this isn't an issue.

You can also use through hole 1% resistors on the 12Step rotary switch if you use a panel mount switch. The resistors can all be soldered together across the switch and 1 wire run to the PCB instead of 12 wires. You can see that one end of each resistor is connected to the same end of all the other resistors in the schematic. The surface mount resistors are there if you use the PCB mounted rotary switch.

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 27 Dec 2019