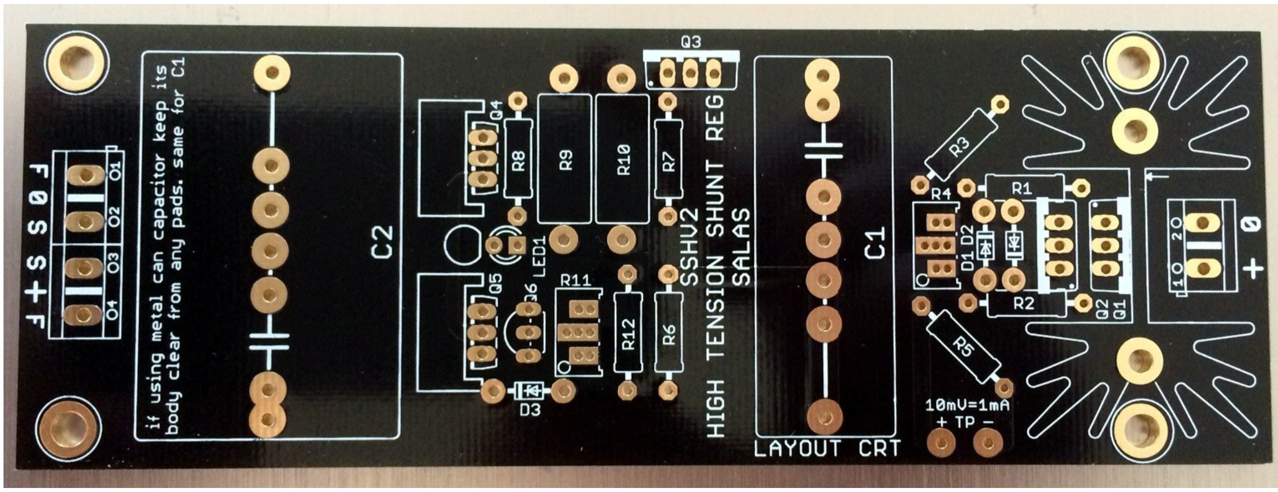
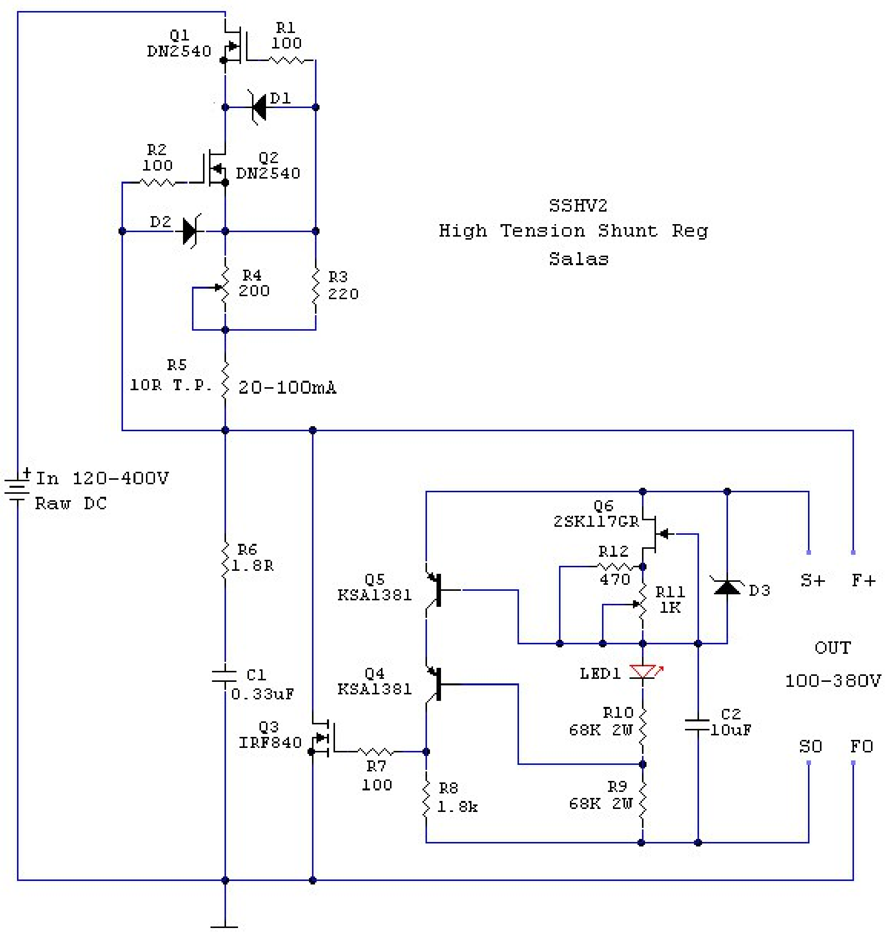
SSHV2 regulator Build Guide
(Salas Shunt High Voltage Two)



What is SSHV2

Its a constant current high voltage shunt regulator of simple topology based on MOSFETS. Useful for powering tube circuits. Same few parts count as in SSHV1. But easier current setting, stabler, higher resolution. There is an official PCB for it like the one in the photo.



It implements the above schematic. For circuit description click this link CCT details

How to build and test it

Populate the board. Q4-Q5 are plastic backed so watch for the letters side to face R8 and Q6. LED1 cathode goes to the square pad. Fix Q1 on its board level sink. Use a side sink for Q3.*

Check you had insulated Q1 & Q3 electrically well from their sinks.

Calculate a dummy load resistor for simulating your target circuit's max consumption current at nominal B+ voltage. The idea is to test on safe but life like load conditions, so to set the regulator and evaluate its good working and heat emission order, before applying it to your precious real circuit.

If you are aiming the reg for a preamp that needs 20mA @ 200V for instance, Ohm's law equates 10K load (200V/0.02A=10000 Ohm). Dissipation on it will be (V*V)/R=4W. Use a 10W cement resistor to avoid overheating it.

Attach S0 & F0 to one resistor's leg, S+ & F+ to its other leg. All 4 wires should be used always, there is damage danger even if not. Set R4 and R11 to middle positions. Attach to the DC input connector short enough twisted cables from your raw DC reservoir capacitor. If cabling is long, decouple with 0.1uF 450V across the regulator's input right there at the connector.

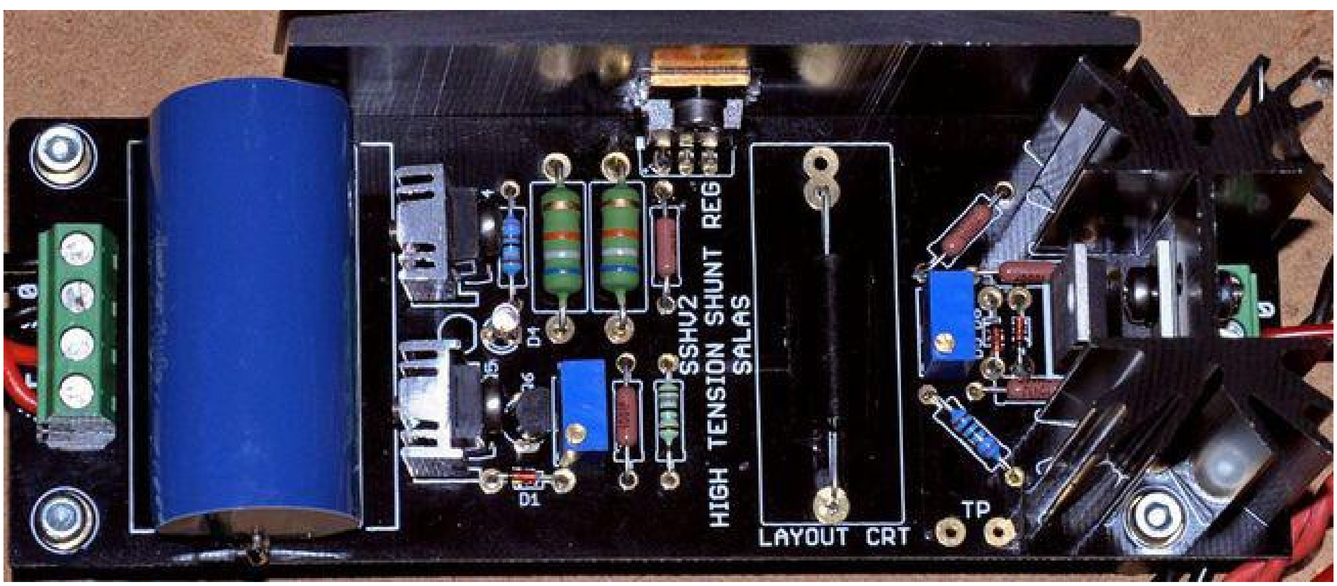
20V higher DC input than the target DC output level is good drop over the regulator to work correctly. The CCS current to set with R4 should always be the client device's max consumption current ***plus*** 20mA spare so to feed the reg's internal circuit correctly to spec. Can spare more, but avoid less.

Now power on. Quickly measure voltage on the two pads marked TP. What you see there in mV is representing CCS mA times 10. When you target 40mA, you should turn R4 to read 400mV across TP for example.

Now measure the voltage across your load simulator resistor. Turn R11 until you reach target B+. Watch it for 15 minutes. If voltage drift is just a few volts and heat on sinks is acceptable, trim out any voltage drift due to parts warm up by turning R11 back to target again and power off.

Next is to remove the dummy load and attach S+ & F+ to your circuits B+ connecting point, also S0 & F0 to its PSU ground point. Twist S wires together and F wires together in sense and force pairs. Wire gauge is largely indifferent since this 4 wire technique that is called "remote sensing" is devised to ignore wire resistance. Just use manageable high voltage insulation wire.

No need to keep the regulator board very near to hot tubes, its performance will hold for logical distances due to remote sensing. You get significant freedom of arrangement. A finished regulator shall look like this:



Some print details only are revised on the newer board of page one vs this finished prototype example

***Side Sink**

Q3 dissipates Watt power of what spare current you allowed times the B+ voltage. Calculate C/W sink spec for no more than 60C sink temperature in worse ambient conditions. Look for sinking guides on the web if never done it. 1.5C/W spec sink should cover most applications for SSHV2 range though.

Specifications

120V-400VDC input. 100-380VDC output. 20-110mA current limit.
60dB open loop gain. 24KHz open loop bandwidth. 90 deg phase margin at 7 MHz.
11mOhm output impedance 20Hz to 20kHz for 20mA spare current. PCB 136x48mm double layer.

Parts list

Resistors (1/4W else otherwise noted):
R1,R2,R7=100 Ohm
R3=220 Ohm
R4=200 Ohm multiturn trimmer 0.5W top screw type
R5=10 Ohm
R6=1.8 Ohm
R8=1.8K
R9,R10=68K 2W (Up to 300Vout 1W permissible)
R11=1K multiturn trimmer 0.5W top screw type
R12=470 Ohm

Capacitors 400V MKP:
C1=0.33uF axial or radial
C2=10uF axial or radial

Semiconductors:
D1,D2==12V 0.5W-1W Zener or 1N4007 diode (same cathode orientation for Zener or diode)
D3=12V 0.5W-1W Zener
LED1= 3mm Red LED generic
Q1,Q2=DN2540 DMOSFET Supertex
Q3=IRF840 MOSFET N channel
Q4,Q5=KSA1381 Fairchild (2SA1381 or "SA1381" Sanyo FBET)
Q6=2SK117GR Toshiba NJFET 3-4mA IDSS or 2SK880GR on SMT adapter

Sinks etc.:
On board 25.4mm pitch sink for Q1. Example Wakefield 647-10ABP
TO-126 mini sinks for Q4,Q5 (not necessary below 300Vout)
Outboard sink for Q3. See the top of page paragraph
Insulation pads and grommets for two TO-220s, thermal grease, 3pcs 5mm pitch Molex connectors.

CAUTION

Building and testing high voltage circuits takes much caution. There is potential for electrocution or fire. If not familiar with high voltage safety handling precautions you are asked to avoid this project.