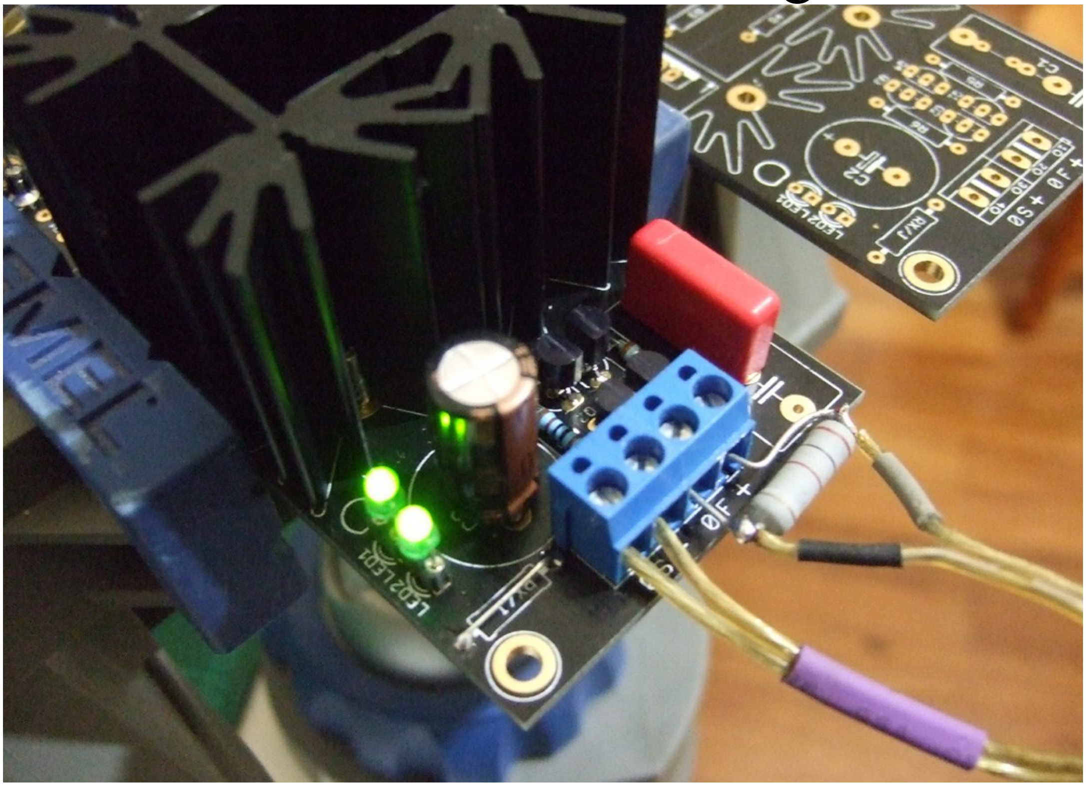
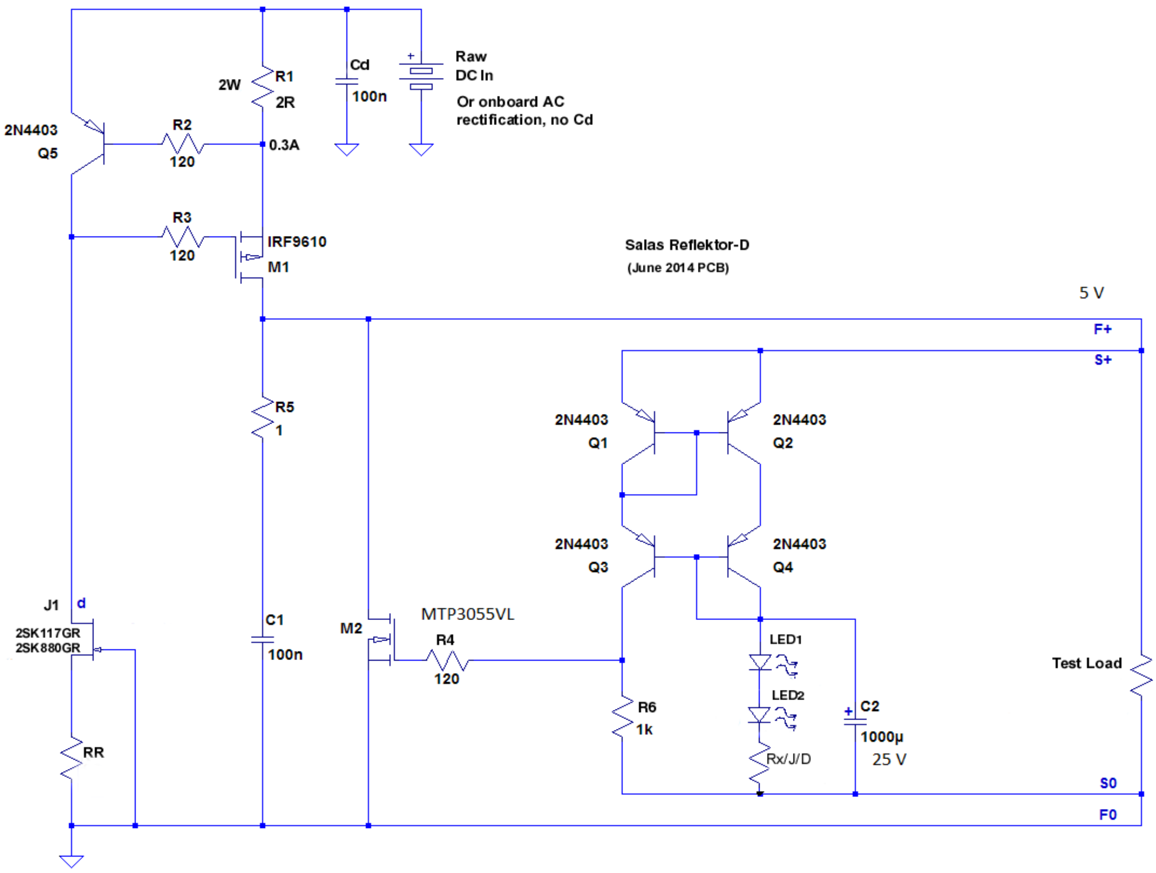
Salas Reflektor-D build guide



The Reflektor concept is a shunt voltage regulator made around a current mirror circuit. To keep equal current between its input and output legs, the mirror reacts to flow disturbances and drives a MOSFET in opposite direction. A current loop simple regulator is another way to describe it. Specifically Reflektor-D is a version focusing on the voltage region common for powering digital source projects. For convenient mounting and setting it utilizes board level sinks and simple ways to choose its current limit and voltage output levels. Lets look into its schematic first.

At the top left a constant current source is formed by a BJT controlling current through a MOSFET.



Q5's VBE is fixed across R1 so its value in Ohms sets the current trough M1. J1 is a low noise JFET biasing Q5 with steady current helping its performance. TO-92 or SMD Toshiba both fit on its pcb. This CCS provides some isolation from incoming raw DC quirks, current limits in case of an output short, and steadily biases the output MOSFET into high enough ON state.

The parallel (shunt) voltage output control circuit is comprised of Q1-Q4 low noise PNPs forming a George R. Wilson improved current mirror which is voltage referenced by LED(S) in this case. This mirror type gives better performance than a two transistors basic mirror, works well even on lowish HFE, and its not peculiar for matching. A consistent enough production type BJT suffices. C2 filters the LEDS noise. Rx/J/D is for extra little voltage parts when the target can not be reached by LEDS Vf only. Should be jumpered if no other component is used.

M2 is a high speed Logic Level MOSFET dynamically adjusting the output current as driven by I to V action via R6 on its gate. Its low VGS threshold allows small enough Vout settings. R5 & C1 form a local Zobel termination for proper phase shaping. Test Load is a dummy resistor for closing the Kelving output connections and aiding the M2 workload while setting and testing a build.
Lets prepare the regulator to work in practice now.

A. Planning the current set.

First thing you need to know is your digital project's current draw. If its not listed in its spec ask the designer. Last resort is to use 0.1 Ohm resistor in series to its rail and measure mV across it, then solve Ohm's law for current. DMM in series and A mode can be used too but prone to accidents.

Next thing is you need to decide a wise overhead of available current to load's max current draw. CCS(A)=0.61V/R1 in Ohm. You can calculate any settings from that. But here is a look up table:

CCS (A)				
0.6				X 350-450 mA
0.5			X 200-350 mA	
0.4		X 100-200 mA		
0.3	X Recommended load draw is 50-100mA			
	2	1.5	1.2	1 R1 (Ohm)

You can see the CCS settings for typical R1 values meeting with an X and the load range to use.

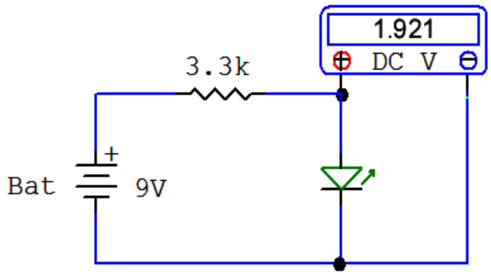
Those settings are expecting a 9VAC transformer secondary. No higher current setting or secondary voltage is wise to be applied. That is for keeping the dissipation and the heat within safe bounds. Shunt regulators are generally hot and you better provide ample ventilation in the vicinity of the sinks in any setting. Some ventilation under the PCB location also helps a lot.

B. Planning the voltage output.

Your application (DAC etc.) dictates certain Vout (VCC). To meet spec decide the right voltage reference components combinations. Since there is always1.22V residual from the current mirror, the rest to a Vout target is provided by LED(S) Vf and any Rx/J/D extra part.

Three green LEDS on LED1-LED2 and Rx/J/D positions give near 7V.
Two green LEDS on LED1-LED2 positions and a jumper on Rx/J/D position give near 5V.
One red LED on LED1, jumper on LED2, and 1N4148 on Rx/J/D give near 3.6V.
One green LED on LED1, jumper on LED2, and 62R on Rx/J/D give near 3.3V.

There is ~2mA bias for the voltage setting components when in circuit, and if you want to predict Vout well you can measure and pick your LEDS Vf with a 3K3 and 9V battery to simulate that.



Example with two Kingbright greens: 1.22V residual + 1.92 + 1.92 + jumper = 5.06 VDC

Building the PCB.

Solder all resistors first. If you got SMD J1 solder that too. If you are using 1A axial diodes (for up to 0.3A CCS only) solder them too, but little off board standing. Also do any Rx/J/D item other than LEDS. Orientate any diode on Rx/J/D for cathode stripe towards the output connector. If to use test pins for LED positions, solder them now. Then do the TO-92s. Now solder C1 and the input-output screw connectors. Then the electrolytic(s) and any TO-220 bridge diodes. No diodes or CM electrolytic if planning raw DC IN.

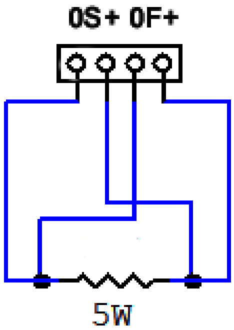
Mount the MOSFETS on the sinks middle holes using a bit of thermal paste but not tight yet. No insulators, use metal screws and bolts. Place them on the PCB so feet and pins clear the pads well, sinks sitting nice. Turn the board upside down while holding the sinks, rest them on a flat surface. Solder the sinks feet. Turn the pcb around again. Tighten the MOSFETS while manipulating them parallel to the sinks fins with their tabs sitting a few mm below the top sink holes. Finally turn it around to solder their pins.

Testing

You should test the regulator on its own first before going to actually power your DAC etc.
Its to confirm it works stably and reaches your target voltage output while showing some healthy power dissipation by radiating heat.

If you had installed pin sockets push the LED(S) in them with trimmed legs in equal length or lightly solder the LED(S) to their pads edges only, for a confirmation test run. Square pads are for cathodes. There is a D symbol for the LED(S) flat rim side. Cathodes point to the corner mounting hole in other words. Make sure that any empty LED position gets a jumper. The Rx/J/D position should have a jumper, resistor, or diode, depending on what you picked for Vout from the examples in section B. (Complete the voltage Ref circuit, any empty positions will prevent it from working).

Connect the test load resistor as in this schematic. Never forget to connect that dummy load resistor before any power on attempt! 20 Ohm 5W plain part for up to 5V Vout or 30 Ohm 5W for 7V Vout.



Now connect the transformer's 9VAC secondary to AC IN or 10V raw DC from some available rail to DC IN. Ability for DC IN connection means that you haven't installed any bridge diodes, no main capacitor CM either, but the CD decoupling cap and a DC IN connector in its place. In case of AC IN, the transformer's primary should be fused to its label spec. In case of DC IN, it should come from a current limited bench PSU or any other DC line fused at 750mA with a slow blow (T) fuse. Time to power on.

LED(S) must shine. If not, power down fast and troubleshoot fuses, connections, orientations etc. If they indeed shine, probe the outer left and right Vout connector screws with the DVM (red probe to F+ for correct polarity indication). You should now see a reading close to your planned Vout. The temperature in the sinks will rise slowly. The test load will feel hot too eventually.
Twelve minutes are enough to heat the reg up when twenty five minutes should bring the sinks to full. You should get 40-65C on the sinks. Recheck your target output voltage. If accuracy is not to your satisfaction, turn it off and exchange the LED(S) with more likely forward voltage drop ones.

Keep in mind that Vout goes down about 0.05V from cold to hot due to the mirror's BJTs settling for a hair lower VBE with rising temperature. Its not much, but maybe you want to plan for it.

After you are happy with your settings and readings, power down, remove any test pins or mini sockets and solder the finally selected LED(S) normally. Remove the test load dummy resistor, install the build to its final place, and connect it to power your digital source or other device.

Wire from S+ and F+ output connector points to your device's positive rail power input.
Wire from both labeled with 0 output connector points to your device's power ground input.
The wires can be medium to light gauge. Its a Kelvin connection that ignores cable resistance.
Counting from left to right when facing the output connector, 0S+ (1,2) and 0F+ (3,4) can be twisted pairs. (1,2) can even be light gauge shielded coaxial if an interference problem is suspected.

Parts list

R1 = 2W Pick value from A table. 100ppm or better like Vishay Dale CPF2 or Caddock MP930.
R2, R3, R4 = 120 Ohm 1/4W or 0.5W 100ppm or better, axial metal film.
R5 = 1 Ohm 1/4W or 0.5W 100ppm or better, axial metal film.
R6 = 1 kOhm 1/4W or 0.5W 100ppm or better, axial metal film or Vishay VAR.
RR = When IDSS J1<4mA use 47R, 4-5mA 62R, 5-6mA 82R, 6mA+ 100R .
Rx = 62 Ohm 100ppm or better, axial metal film. Useful in the 3.3V setting mainly.
Test Load = 20 Ohm 5W for 3.3V-5Vout. 30 Ohm 5W for 7Vout. (0.3A-0.6A CCS).

C1 = 0.1uF MKP or other high quality dielectric. 5mm pitch to 22.5mm pitch.

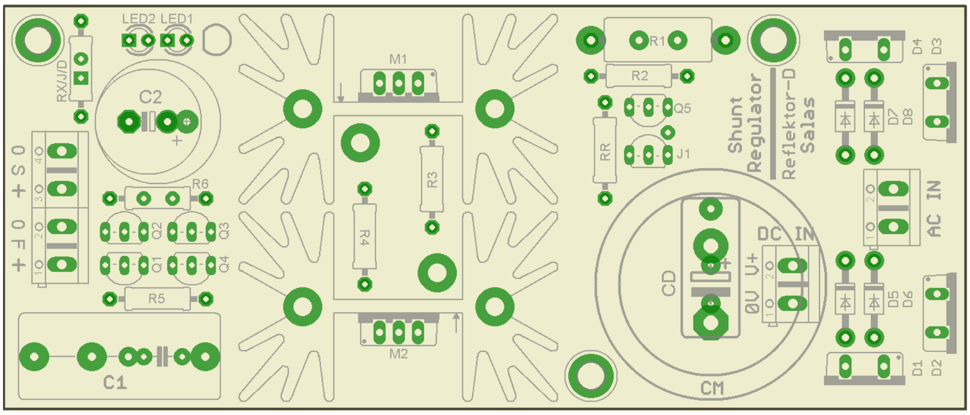
C2 = 1000uF 25V Nichicon KZ or alike. 5mm & 7.5mm pads. Up to 16mm Φ.
CD = 0.1uF MKP cap 5mm or 15mm pitch. For decoupling when using DC IN configuration only.
CM = 10000-22000uF 25V 10mm pitch snap in. Up to 30mm Φ if 105C spec. 22-25mm if 85C.*

Q1, Q2 , Q3 ,Q4, Q5 = 2N4403. (Fairchild 2N4403TA is one good consistency tested example).
J1 = 2SK117GR through hole, or 2SK880GR surface mount.
M1 = IRF9610 P-MOSFET.
M2 = MTP3055VL N-MOSFET.

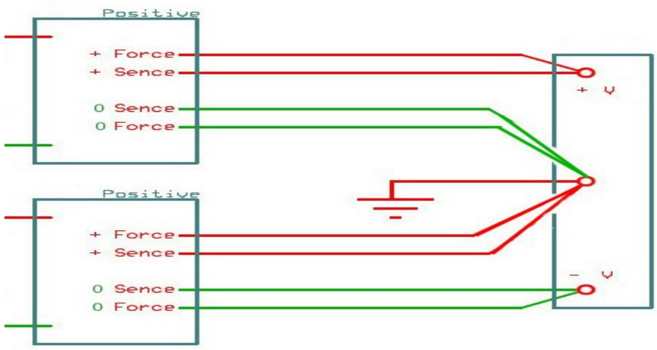
LED1, LED2 = LED types that showed proper Vf and good consistency at 2mA bias are :
Kingbright red 1.735
Kingbright yellow 1.860 (All colors come from the WP2773 line)
Kingbright green 1.921

RD = 1N4148 fast diode. Useful in the 3.6V setting mainly. Or a third green LED for 7V.
D1, D2, D3, D4 = 8A TO-220 diodes. MSR(F)860G, MUR820, U08A20 etc.*
D5, D6, D7, D8 = 1A DO-41 diodes. MUR120 etc. Only for up to 0.3A CCS setting.*

Connectors = 5mm pitch screw type terminal interlocking connectors. Three pieces.
Test pins = 2.54mm pitch female header pins. Cut four from a row. Another two for 7V.
Heatsinks = Wakefield 647-20ABPE. Two pieces.
Transformer = 9VAC secondary 30VA or bigger. *For AC IN configuration only.
PCB = 128x54x2mm matte black official Reflektor-D PCB with thick tracks and gold pads.



The PCB hosts a positive regulator but two builds can be stacked in symmetry also:



Special thanks to diyaudio.com forum & members Kazuo Ozawa, RCruz, Crt, Tea-Bag, Vgeorge.