GYRATOR PCB

Build Instructions
Version 2.0 PCB rev 0.7

June 2017

INTRODUCTION

After 7 years of working with the gyrator load circuit in DHT preamps and valve amp driver stages, a friend of mine suggested in building this PCB for the DIY audio community. The result of many years of experience and testing plus some refinement of the PCB design made by Tom Browne from Sonic Illusions, is this flexible PCB.

If you don't know what I'm talking about, however someone told you about this board, then I suggest you pay a visit to some of the following articles and blog entries on my website:

01a DHT Pre-amplifier (Gen2)
Gyrator boards

PCB Features

- Gyrator load (or voltage-controlled CCS) to build a "hybrid" mu-follower stage for either DHT or IHT valve stages:
  - DHT Preamps (e.g. 4P1L, 01a, 26, etc.)
  - DHT output stage drivers (e.g. 300B, 45, 10Y, 211/845, 813, etc.)
  - Push-pull drivers in LTP configuration
  - Phono output stages
  - Parafeed stages
  - Headphone amps
- Flexible PCB design to fit multiple capacitors
  - From a low cost WIMA MKS 100nF / 630V to
  - Any boutique cap up to 2.7cm x 6cm
- It can be re-wired to build a CCS load instead
- Stable voltage reference source provided by a cascoded pair of LND150 FETs
- The top MOSFET is located next to the edge of the PCB so you can use the heatsink (if needed) of your choice
- Top MOSFET can be either DN2540 or any other HV depletion MOSFET like IXYS 01N100D/08N100D or similar
- Various options in the PCB to fit low noise and low capacitance JFET like:

www.bartola.co.uk/valves
email: **valves@bartola.co.uk**

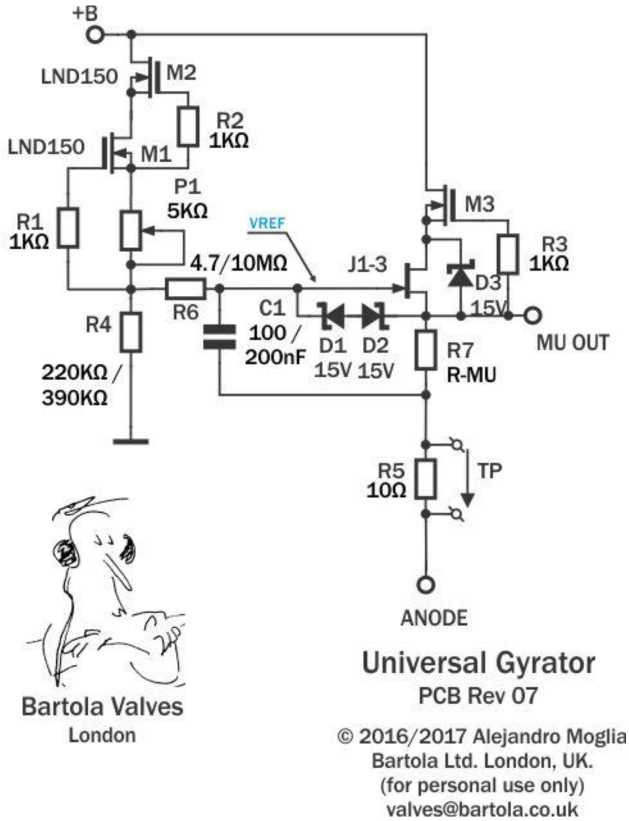
    - J309/J310 (TO-92)
    - 2N7000 (TO-92)
    - 2SK170/LSK170 (TO-92)
    - BF862 / MMBJ310 SMD (SOT-23)
- For higher anode current, various options to fit high transconductance and low capacitance MOSFETs (see Appendix) like:
    - BSH111BK (SOT-23)
    - BSN20BK (SOT-23)
- The jFETs and MOSFETs recommended will provide a wide range of anode currents with great HF response.
- Anode current test point across a 10 Ω resistor
- PCB characteristics:
    - 70x65mm board on a high-quality 1.6mm thick FR4 board
    - ENIG finish, 2 oz. copper.
- Rev 07 updates:
    - New SMD pads to fit BSH111BK/BSN20BK and most of enhancement MOSFET in SOT-23 package

Overview

I'm not going to cover in detail this well know circuit. Basically, the gyrator is a voltage-controlled current load which creates a hybrid mu-follower when used as the load of a valve. There are some benefits to highlight from this topology:

- Maximum gain of the stage (close to mu)
- Minimum distortion thanks to high impedance load presented to valve by gyrator
- Low output impedance when output taken from mu out. Output impedance is determined by the lower FET device rather than the valve. This allows using high anode resistance valves with very low distortion
- Suitable for both triodes and pentodes stages!
- Sink/source of current into load thanks to the totem pole arrangement. This helps to minimize issues with slew rate
- Easy to set operating point thanks to controlled voltage reference
- Ability to drive output stage in class A2 if the circuit is DC-coupled.
- High power supply ripple rejection (PSR) thanks to the cascoded FETs which means you don't need to spent too much effort on the smoothing stage of the HT supply
- It sounds really great! (see DIYAudio threads for reference)

THE CIRCUIT



The circuit is very simple. The depletion MOSFET pair M1 and M2 (LND150) forms an CCS. This CCS provides a stable current to develop a reference voltage across R4 for the gyrator. This voltage will bias the cascoded pair J1-3 and M3 through R6. D1, D2 and D3 protect J1-3. R5 is simply a test resistor to measure the anode current. R7 is the mu resistor which is optimized for each stage. C1 provides the bootstrapping needed for AC operation and achieve the low output impedance in the mu output. The top depletion MOSFET (M3) does all the heavy lifting. For low currents (e.g. 1-10mA) it doesn't need a heatsink, however when the gyrator is used in a driver or when currents are greater than 10mA you'd expect to put a small TO-220 clip-on heatsink or bolt it to chassis if needed.
The low frequency response is primarily driven by the RC pair R6 and C1. Typically, I'd use the following combinations: 4.7MΩ and 220nF or 10M and 100nF. The high frequency response is driven by the parasitic capacitances of the lower FETs. Therefore, you want to use a low-noise jFET in J1-3 instead of another depletion FET (unless it has low reverse capacitances – see the BOM and appendix section for more details). The jFET on this position is operating in very unfavorable conditions (i.e. low drain-source voltage) so best use a jFET here for best results.
The circuit has minimum protection and if you short accidentally any output you will kill M3 and J1-3 for sure. The voltage reference is pretty resilient, though, however it can be damaged as well. For the lower FET protection, you will need to add three 15-18V Zener diodes (D1-D3). This is covered later in more detail in the build section.
The value of R4 is determined by the optimal CCS current needed by the LND150 devices for best temperature compensation. This is about 500uA, so the value of R4 is roughly the output voltage divided this reference current. For simplicity, the following guidance is provided:

R4	Anode Voltage (Va)
220KΩ	100-120V

330KΩ	120-160V
390KΩ	160-200V

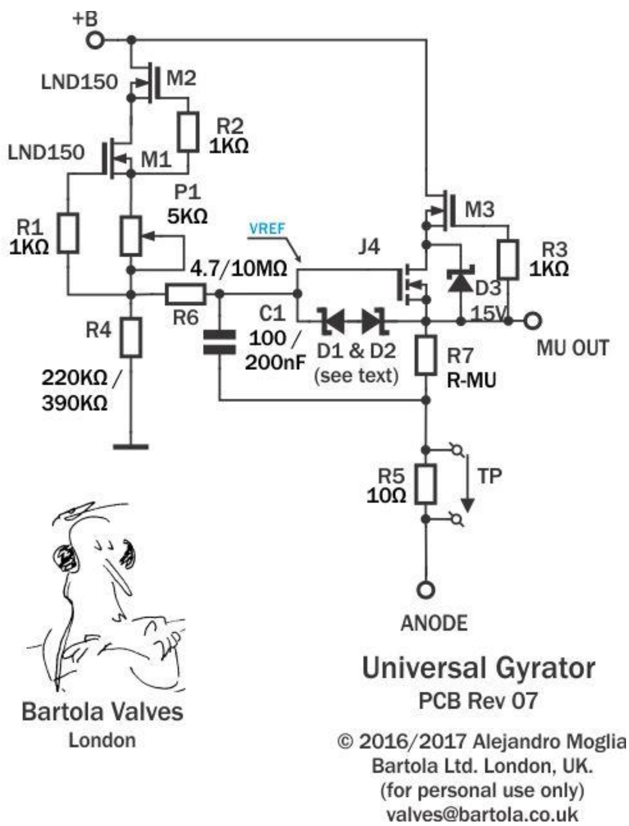
Of course, you can vary the range with P1, however I recommend you aiming for a drain current (Id) on the LND150 cascoded pair below 1mA where possible due to its drift with temperature.
Choosing the value for R4 (Rmu) is somehow more challenging. In theory, best transfer should occur when the Rmu=1/gm where the gm is the transconductance of the lower FET at the operating point (Vds, Id). The transconductance of the FET at lower Vds is lower than the value published in the datasheet (e.g. typically >10V) so it can't be found from the datasheet and has to be estimated. From my measurements and simulations, I came up with a simple table which will help you:

jFET	Pads	Rmu
2SK170	J1	1K5Ω
BF862	J2	330-470 Ω
J310	J3	1KΩ -1K5Ω
2N7000	J1(*)	330-470 Ω

(*) for 2N7000: mount TO-92 inverted in J1 pads for 2SK170. Check datasheet for pinout verification

N.B. You shouldn't play with this circuit if you don't know what you're doing. You will typically modify R4 and Rmu depending the valve used and the circuit conditions. Just follow some of the proven designs and you will be fine!

If you want to use high transconductance and low capacitance MOSFETs (see appendix) instead of J1-3, there is a special pad (J4) introduced in the Rev07 to allow all sorts of SOT-23 SMD devices which have a different pinout from the BF862:

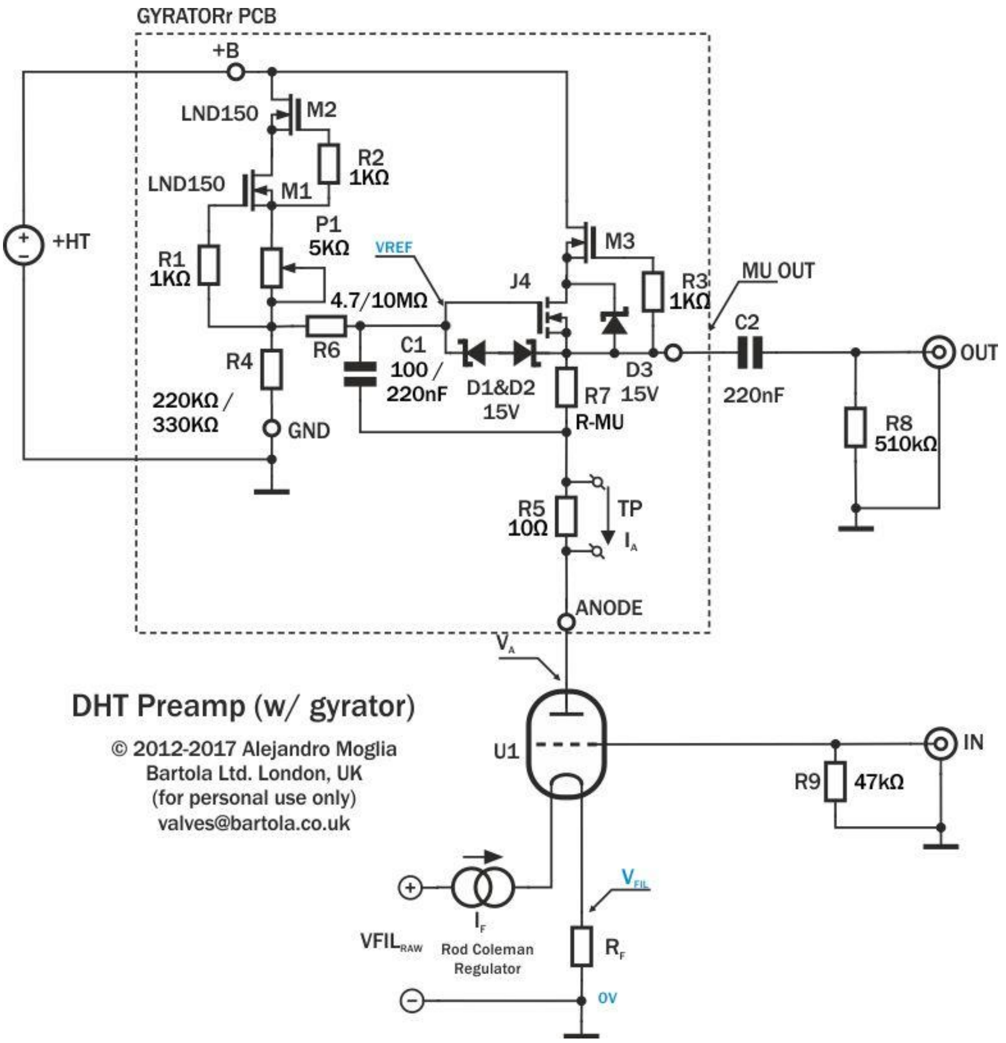


Instead of using J1-3, you solder the SMD MOSFET (e.g. BSH111BK/BSN20BK) in the J4 pad. D1 and D2 may not be required as many SOT-23 MOSFETs come with them internally. Avoiding them is ideal to minimize additional capacitance affecting the high frequency response. This option is deal when building gyrators with anode quiescent currents greater than 25mA. These MOSFETs have higher transconductance than the jFETs, so output impedance of the stage is lower improving therefore the overall bass response.

The choice for Rmu in this case is simple:

J4	Rmu
BSH111BK **BSN20BK**	330-470Ω

Here is the generic DHT preamp circuit model. See the “Circuit Examples” section for practical examples:



The gyrator offers a lot of flexibility. In particular, you can dial the desired current on the anode and adjust unmatched pairs for example. You can also use the same supply for multiple stages with different voltages.

Generally, there is no power supply decoupling needed. Unless you have a high output impedance due to a feedback regulator (not shunt), you won't need it. Otherwise, you will need to add a film cap in parallel at the +B input. Value typically from 100nF to 500nF should be fine.

BUILD

Well, ok. You just want to get going and build the board (I hope you haven't skipped the previous sections as they are very important). The high-quality PCB will arrive as a plain PCB (unless you haven't bought the pre-soldered jFET or MOSFET versions in which case you should look at the end of this section) looking like this:

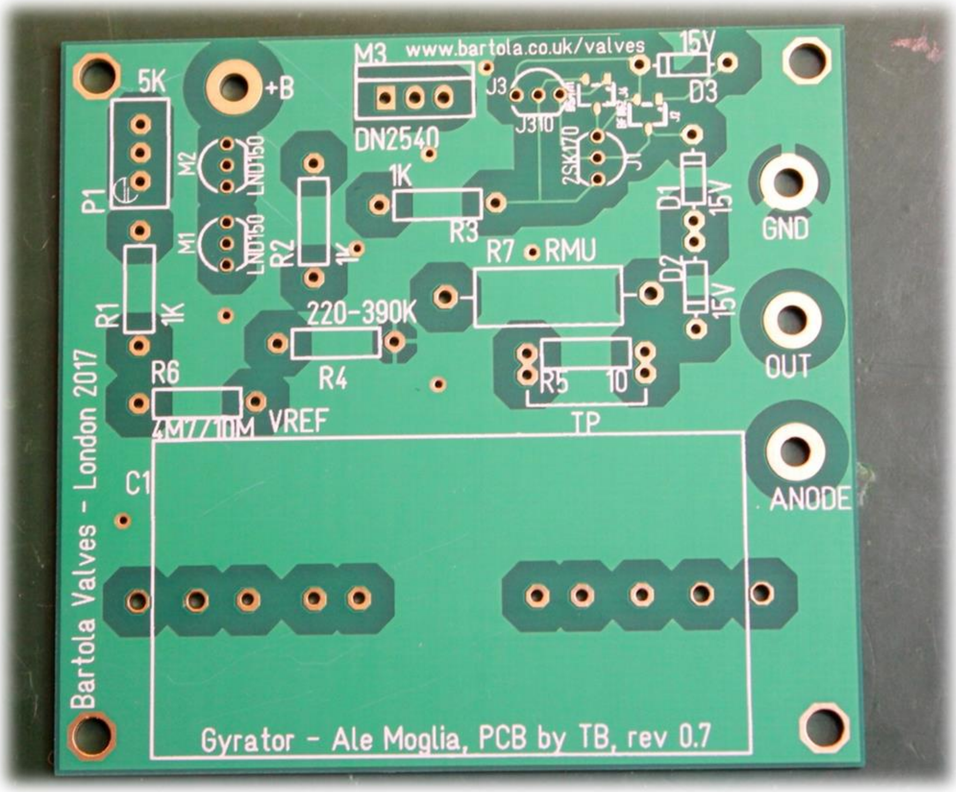


Figure 1 - Build step 0: plain PCB ready to be soldered

The first step in building the gyrator is to solder the SMD FET (either J2 or J4). Alternatively, if you're going to use a TO-92 jFET (e.g. 2SK170/LSK170, J310 or 2N7000 or BS107A) you can either solder it now or later, it doesn't make much difference as it's easier to solder than an SMD device.

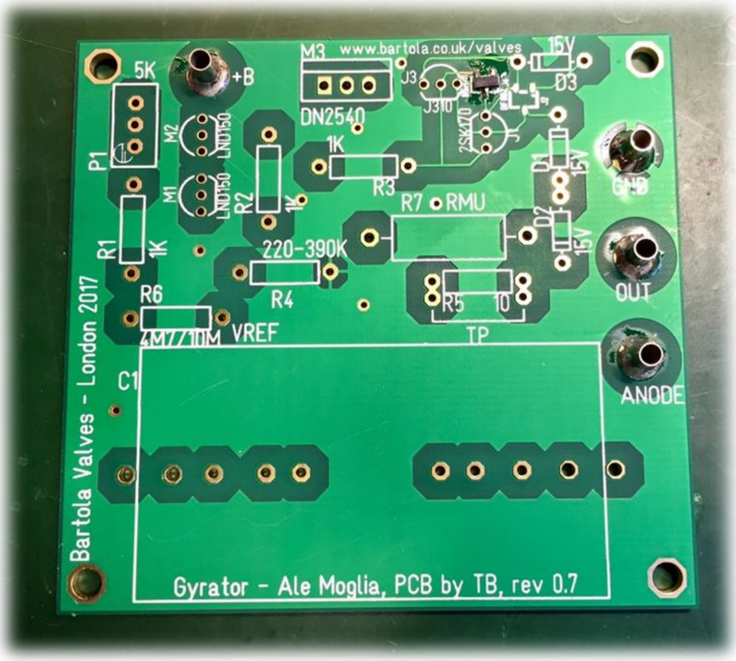


Figure 2 - Build step 1: soldering J2 or J4 SMD FET. (BSH111BK soldered in J4 pads)

Then populate all the resistors. The gate stoppers are typically carbon resistors (R1, R2 and R3). Then you need to solder metal-film resistors R4, R6, R5 and finally R7, the mu resistor which will vary depending your circuit. I like using a nice resistor here like a Kiwame or Takman, but is your choice. Check and measure resistance across all soldered resistors in the PCB

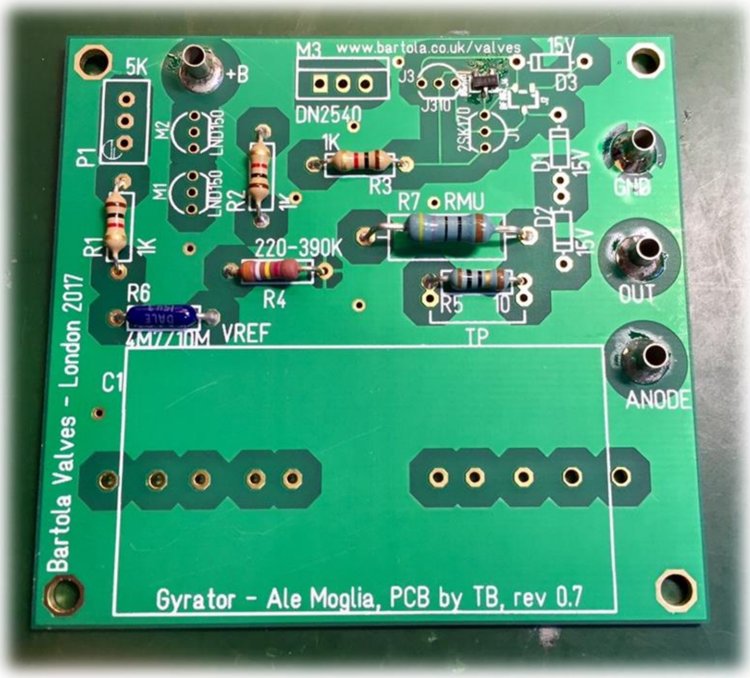


Figure 3 - Build step 2: soldering the resistors

Once you have completed the resistors you can add the protection Zener diodes D1and D2 (if necessary depending on the option of J4) and D3. You can also add the PCB connectors (if you don’t want to solder cables directly to the PCB). Please check the BOM for examples on these connectors.

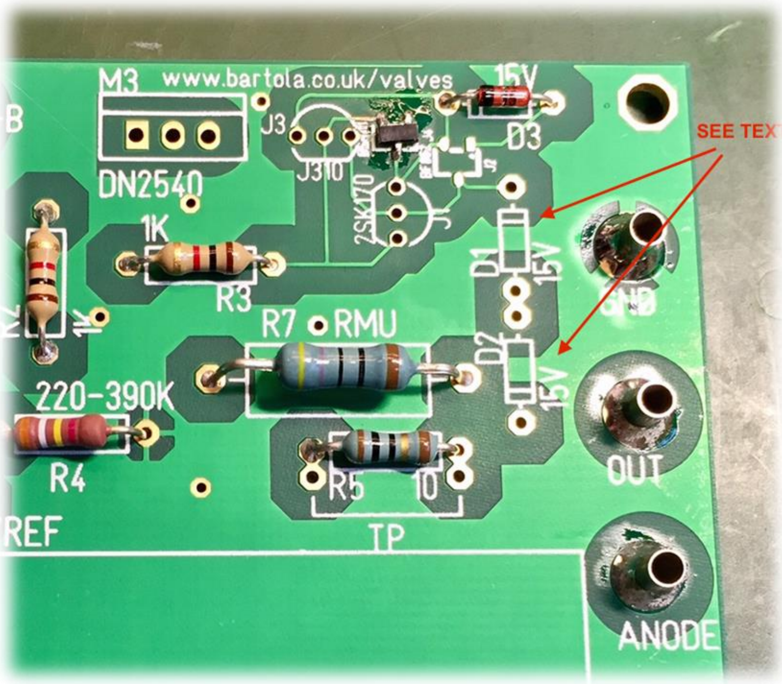


Figure 4 - Build step 3: Protection Zener diodes D1, D2 and D3

The next step is to solder the pair of LND150 in M1 and M2. Check PCB marks to fit the devices correctly with the flat side facing to the gate stopper R2.
You will have to use a fine solder tip here and be careful when soldering these devices. Don’t allow the heat to build up as you will damage them.

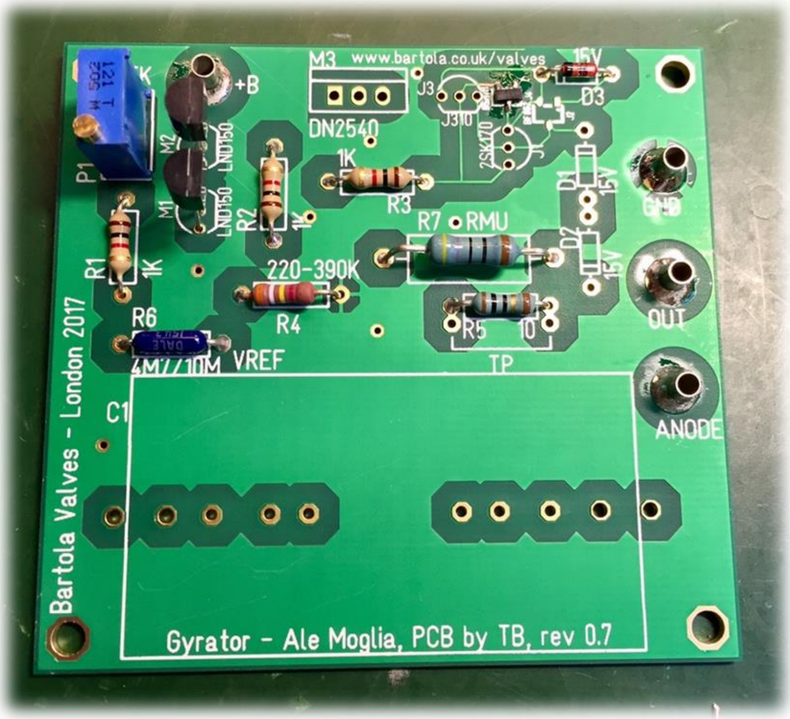


Figure 5 - Build step 4: LND150 CCS devices and P1 trimmer

If you are using an LSK170 or 2SK170 device for J1, then you need to place on the pads labeled "2SK170". The flat side will face to the gate stopper resistor R3.
If you are using the 2N7000 in TO-92 case, you will need to mount it in place of the 2SK170 (J1) but you need to invert the orientation. This means, flat side opposite to the 2SK170 mask drawing.
If you're using instead a J310, then you simply solder it in J3 position marked as "J310". The flat part needs to face to the Rmu resistor.
Once you have fixed the JFET (whichever option) you can then add the top MOSFET in the place marked as M3. Bear in mind that the flat side faces out the PCB. If you need a heatsink you can add it at the end of this build process.

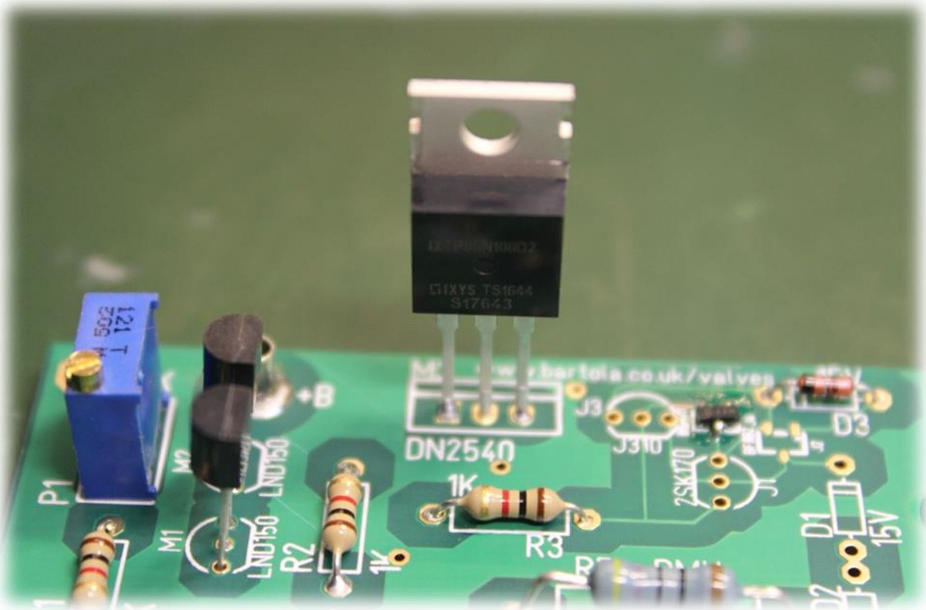


Figure 6 - Build step 4: Placing the top MOSFET M3

www.bartola.co.uk/valves
email: valves@bartola.co.uk

The capacitor C1 comes next. Depending your choice you will have to play around to fit it through a pair of pads. If this is a standard WIMA MKS4 or similar with 15mm lead spacing will fit through on the first set of pads.

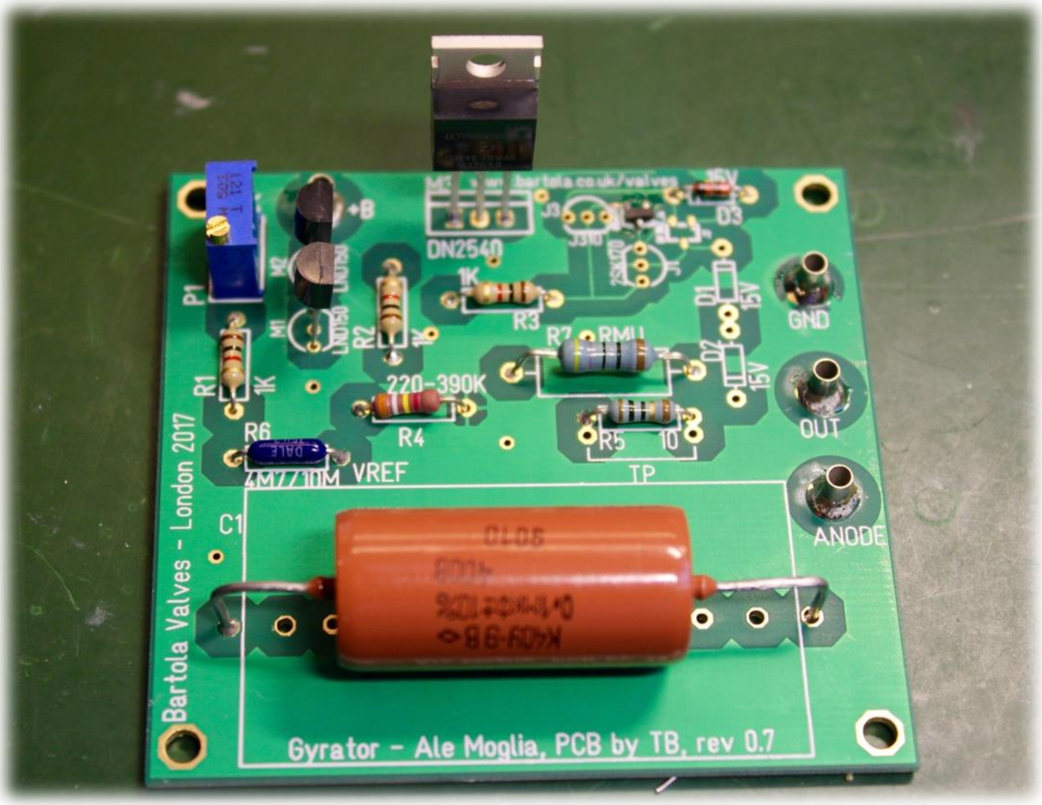


Figure 7 - Build step 5: Capacitor C1 (Option 1)

Soldering the SMD device

Ok, if you adventure yourself in solder the SMD device, make sure you get a proper glass magnifier or alternatively an SMD microscope. They aren't expensive and worth the investment if you're looking to do a lot of SMD soldering.
So, we start with the clean pads to place the BF862 like shown below:

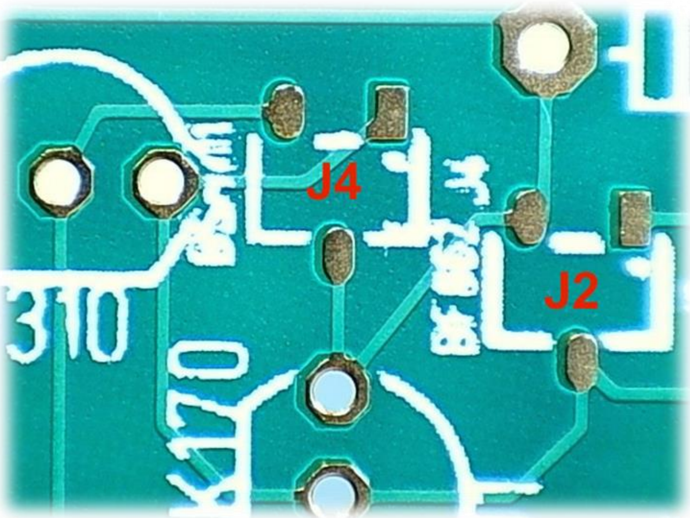


Figure 8 – Soldering SMD FET (Rev07)

We apply gently a tiny bit of SMD solder flux (the one that comes in a syringe) to the three pads:

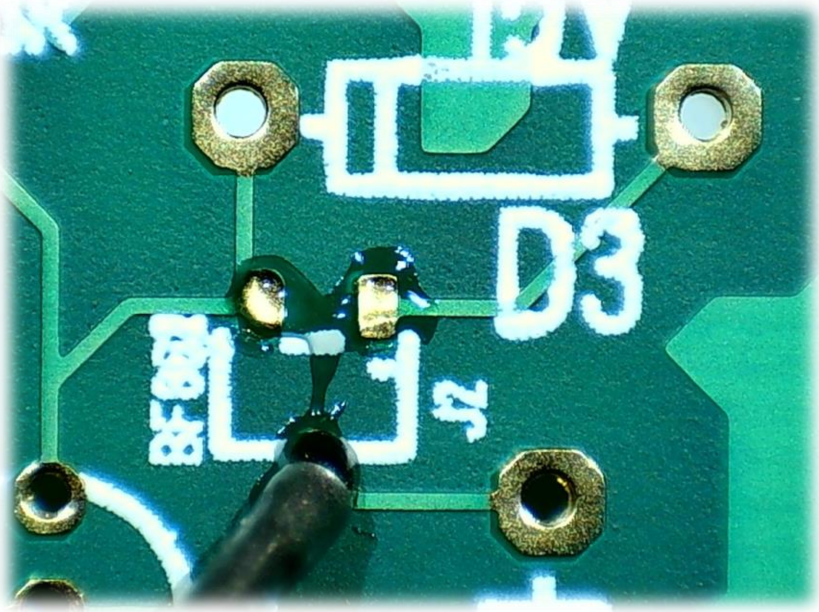


Figure 9 – Soldering SMD FET, applying the solder flux (Rev06 picture)

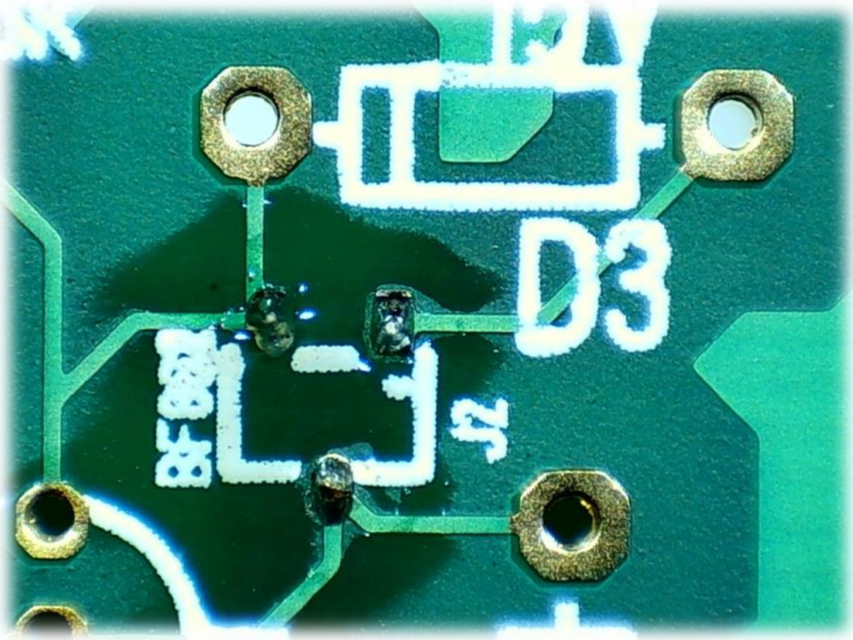


Figure 10 – Soldering SMD FET, pads soldered (Rev06 picture)

We then need to place with a pair of tweezers the SMD FET of choice like this:

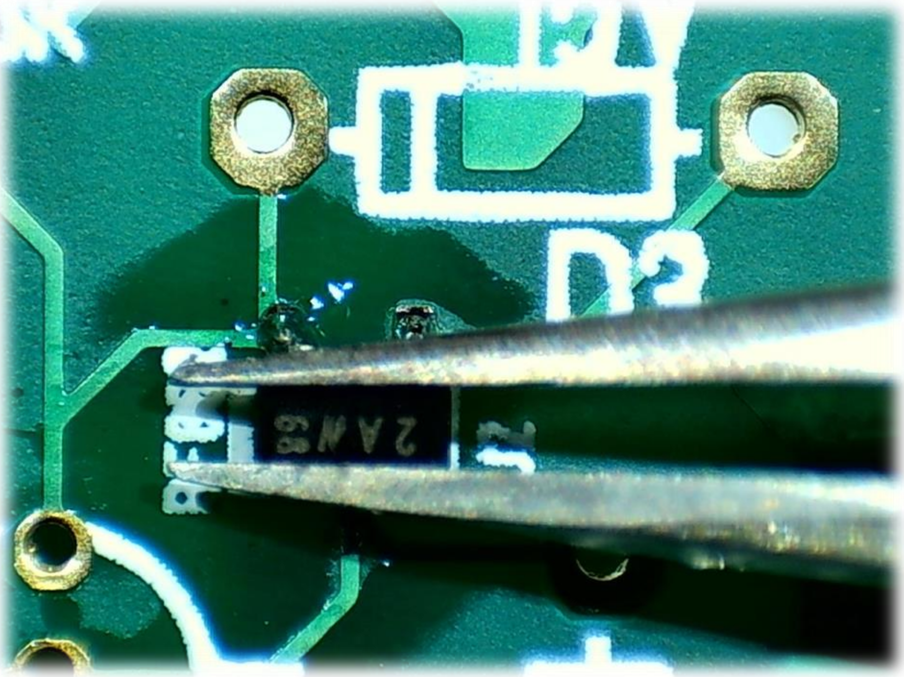


Figure 11 – Soldering SMD FET, placing the SMD device

It's quite tricky to place the SMD but make sure you can solder the gate first (middle pin) and then proceed touching gently the other pins. Once secured properly, you can press with the tweezer (again very softly) from the top of the FET and touch each pin to ensure you can level the SMD device with the PCB. Job done!

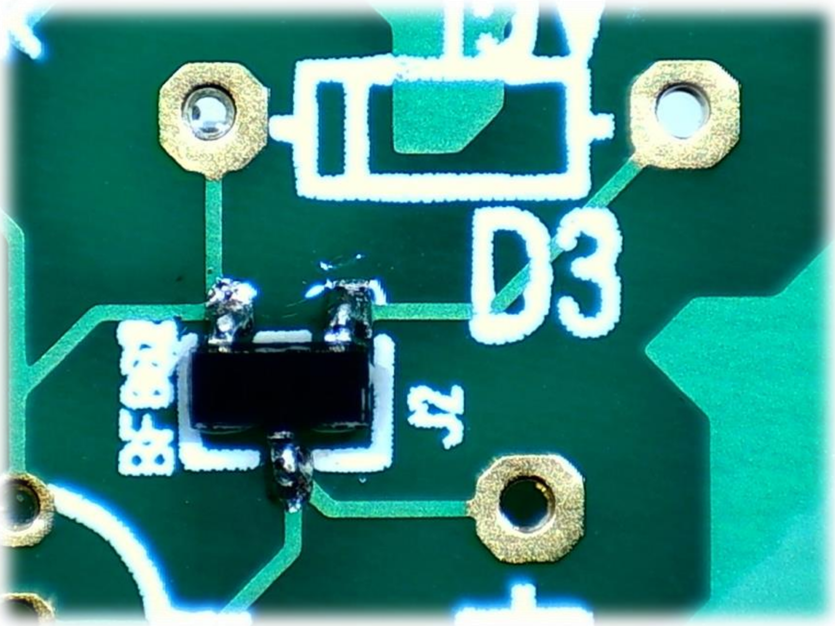


Figure 12 – Soldering SMD FET completed (Rev06 picture)

You can then clean the remaining of the solder flux with isopropyl alcohol and a cotton swab.

Fitting the board

Make sure you use isolated M3 hex standoffs as the M3 holes are connected to ground. Otherwise you will have ground loops resulting in hum.

The boards can be stacked, provide sufficient space for the top FET and also make sure you have P1 with a 90 degree preset screw. Otherwise you won't be able to adjust the boards once stacked!

TESTING

There are some simple testing steps to follow:

- Please don't fire up the HT straight away, you have been warned!
- I don't need to say anything else to remind you about high voltage and the danger around poking your fingers on the PCB when there is high voltage present. Please be careful.
- I'd first test the CCS without any load connected to the anode. You can make sure that the voltage can be set by varying P1 and measuring the reference voltage at VREF. If you measure about HT despite turning P1, then there is something wrong with your CCS and likely you have damaged the pair of LND150.
- I'd strongly recommend using a VARIAC to test the PCB or a variable HT supply.
- Use a 10KΩ -22KΩ / 1 W dummy load resistor instead of the valve, don't avoid this step. Slowly bring the HT up and check the voltage across the dummy resistor. You should be able to adjust P1 to set the right voltage.
- If you accidentally short the anode or the mu output, you will damage the M3 and will short the HT into the output. This can be easily detected when you adjust P1 and you still get HT level regardless of P1 position.
- Once voltage setting at the dummy resistor is achieved, you can replace the dummy resistor with your valve and test again. It should work like a charm!

CIRCUIT EXAMPLES

01a DHT Preamp

A classic example is the 01a preamp as described here. You will need the following:

- HT somehow decently regulated. You can get away with just 150 or 200V and you don't need 235V as shown below. This was my existing supply.
- Filament resistor for filament bias and Rod Coleman's regulator
- Output coupling Cap (220nF) or your preferece as well as input resistor R1
- The gyrator board and you're done!

Here's the latest circuit implementation:

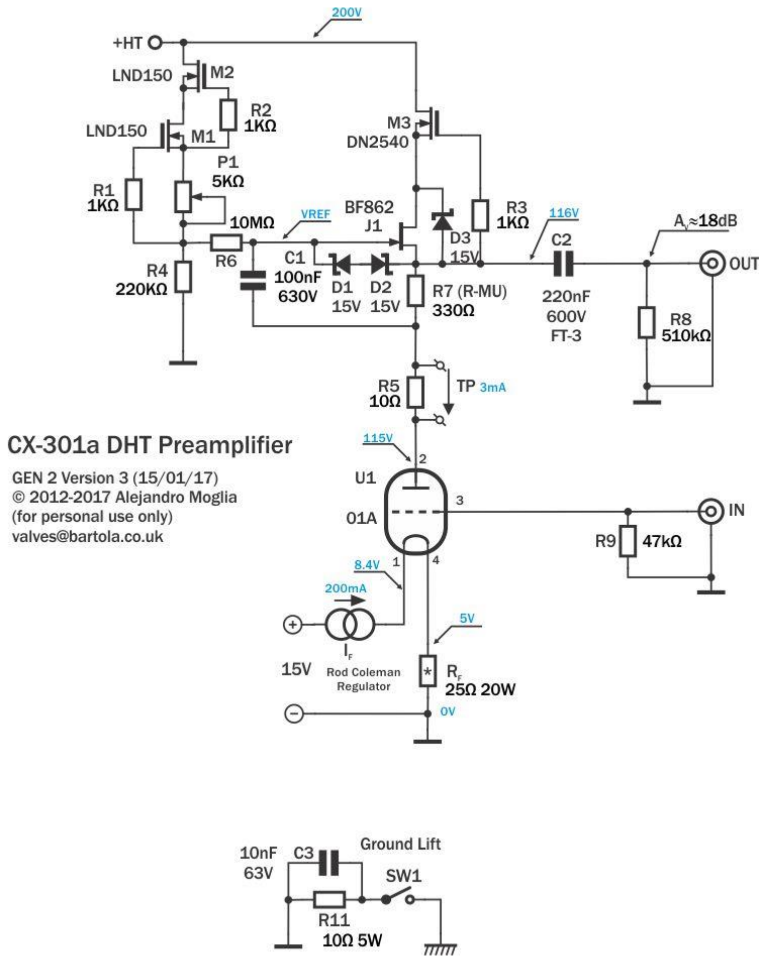


Figure 13 – The classic 01a preamp

This is a circuit which works well with all DHTs (and IHT as well) but you need to change some parts.

26 DHT Preamp

This is the famous 26 preamp but with a gyrator load. Notice that filaments are starved as well. The back to back zeners are missing on the below diagram, they should be included as per the 01a diagram above:

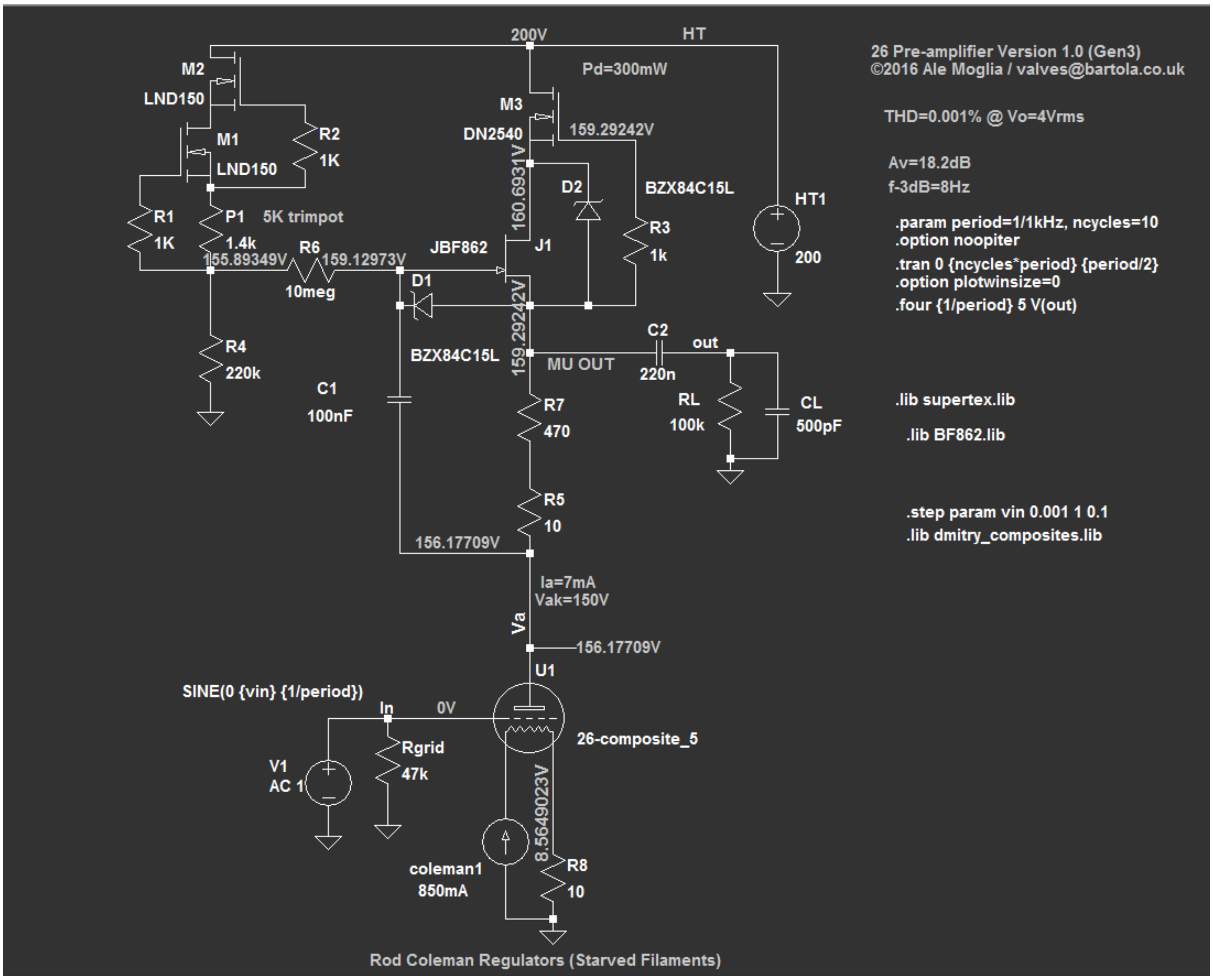


Figure 14 – The 26 DHT preamp with gyrator load

4P1L DHT Preamp (Gen 1b)

Below is an improved version of the 4P1L Siberian preamp. You will need a J310/LSK10D/MMBF310(SMD) or other FETs explained later in this document for 30mA. You can get away with a BF862 (SMD) for 25 mA:

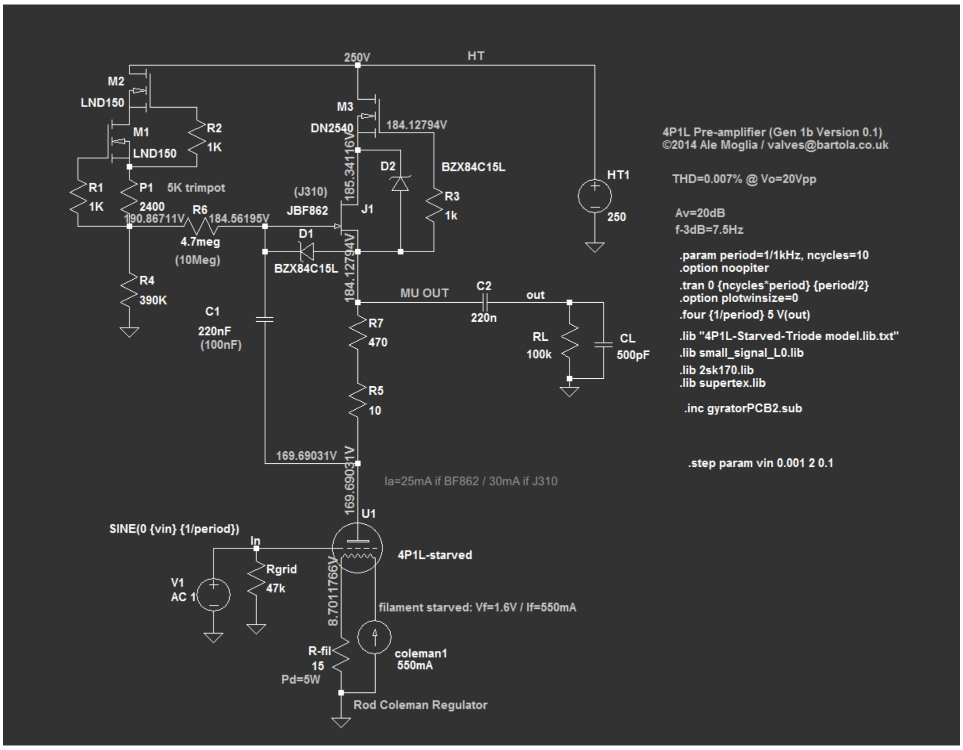


Figure 15 – The famous Siberian preamp (4P1L)

4P1L DHT Preamp (Gen 4)

Here's the updated Siberian preamp in its fourth generation. You can read more about this design on my blog [here](#):

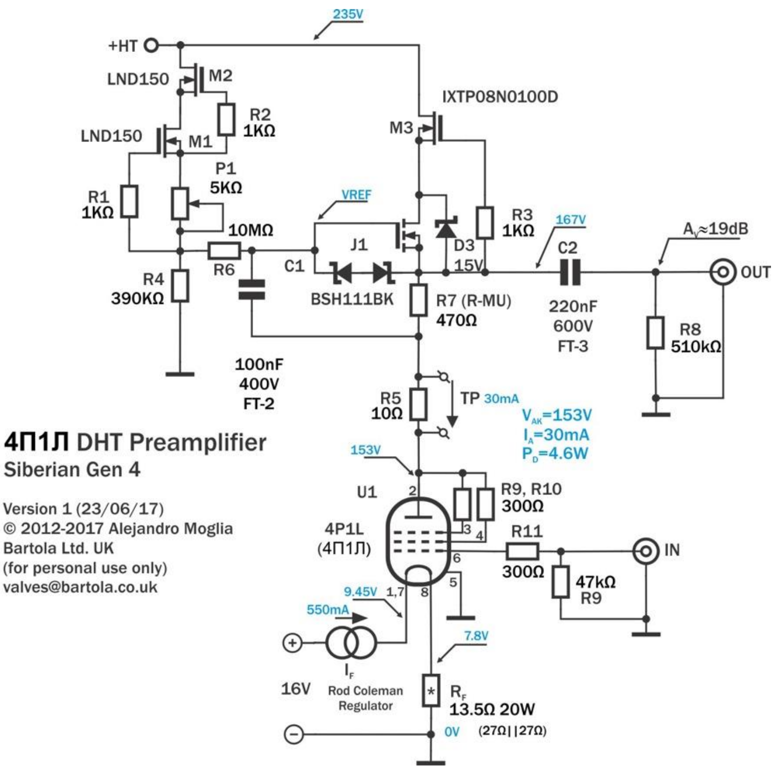


Figure 16 – The famous Siberian preamp (4P1L) – Gen4

2P29L DHT Preamp

A very nice sounding stage alternative to the 4P1L. You can read more about this design on my blog here:

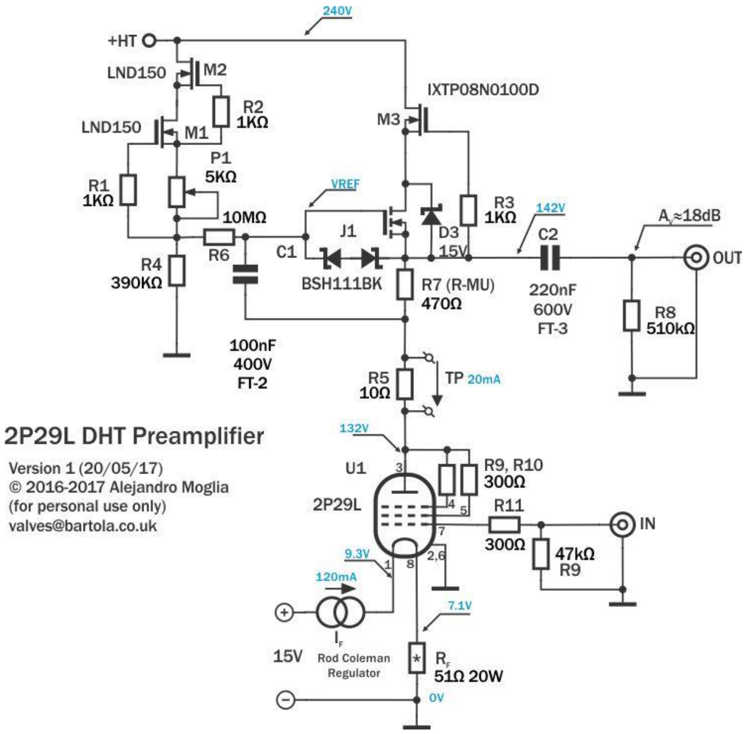


Figure 17 – 2P29L DHT Preamp

VT-25/10Y DHT Preamp

One of my favourite preamps is the VT-25/10Y. You can read more about this design on my blog here:

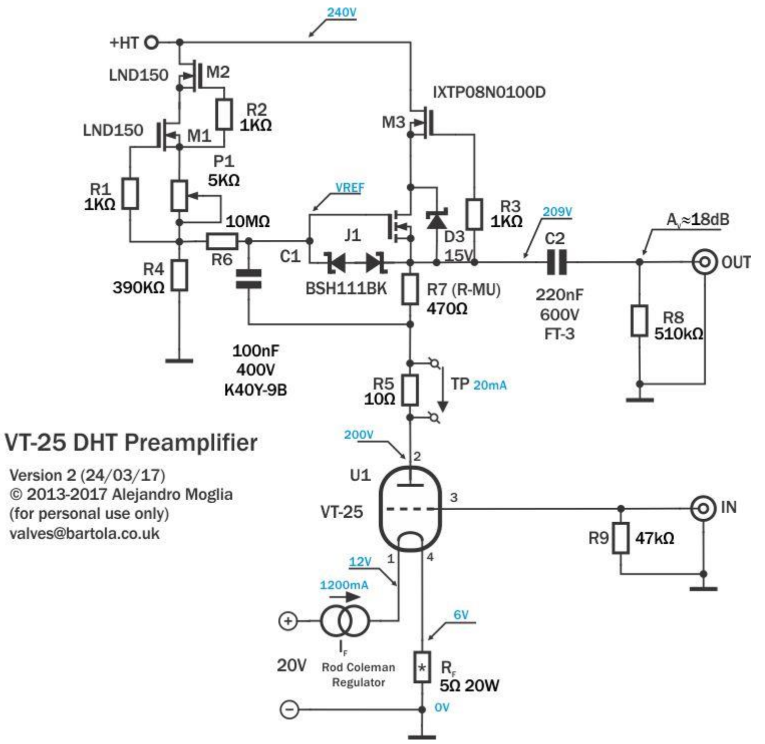


Figure 18 – VT-25/10Y DHT Preamp

Output SiC MOSFET follower buffer

The final example for discussion here is the additional circuit needed when we are using a low current / high anode resistance DHT. There are plenty of nice candidates here, but if we don't take the right measures, we will end up with Slew Rate issues. The workaround is to add a source follower as shown below:

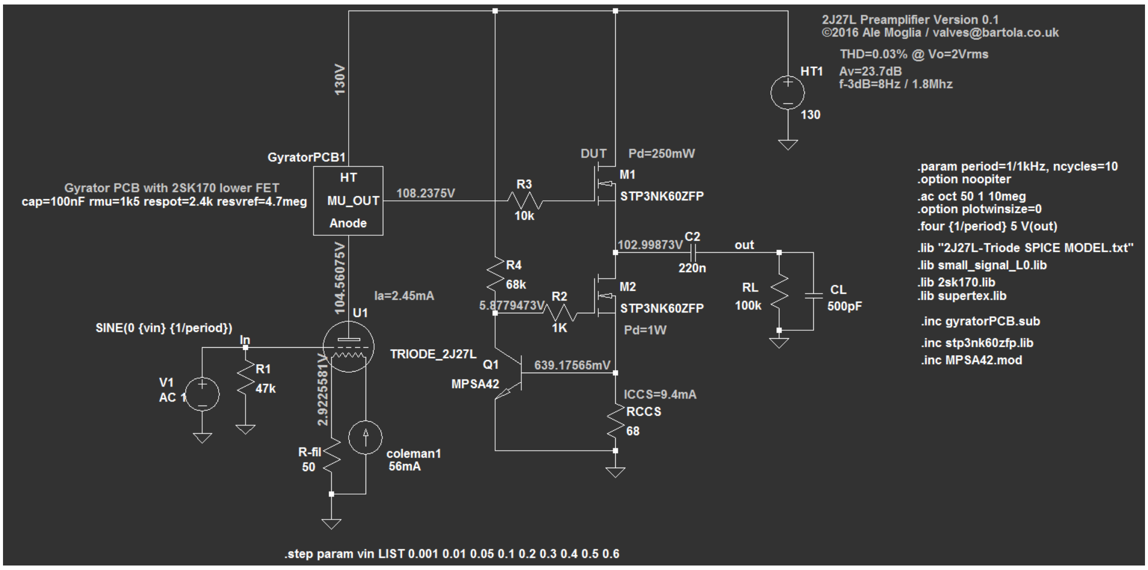


Figure 19 – Adding and output buffer stage

Pentode Gain Stage

Pentodes can benefit from a gyrator load. The gyrator facilitates driving loads which are typically the challenge of small signal pentodes. Here is an example which can be used in a phono stage. Up to 60dB can be achieved in a single stage with the 6Z49P-DR:

- 6Z49P-DR pentode mode test
- RA=68K
- Va=150V, Vgk=-1.6V (SiC Diodes)
- Vs=150V
- Ia=9mA, Is=1.7mA
- Av=52dB
- Bandwidth: 90kHz

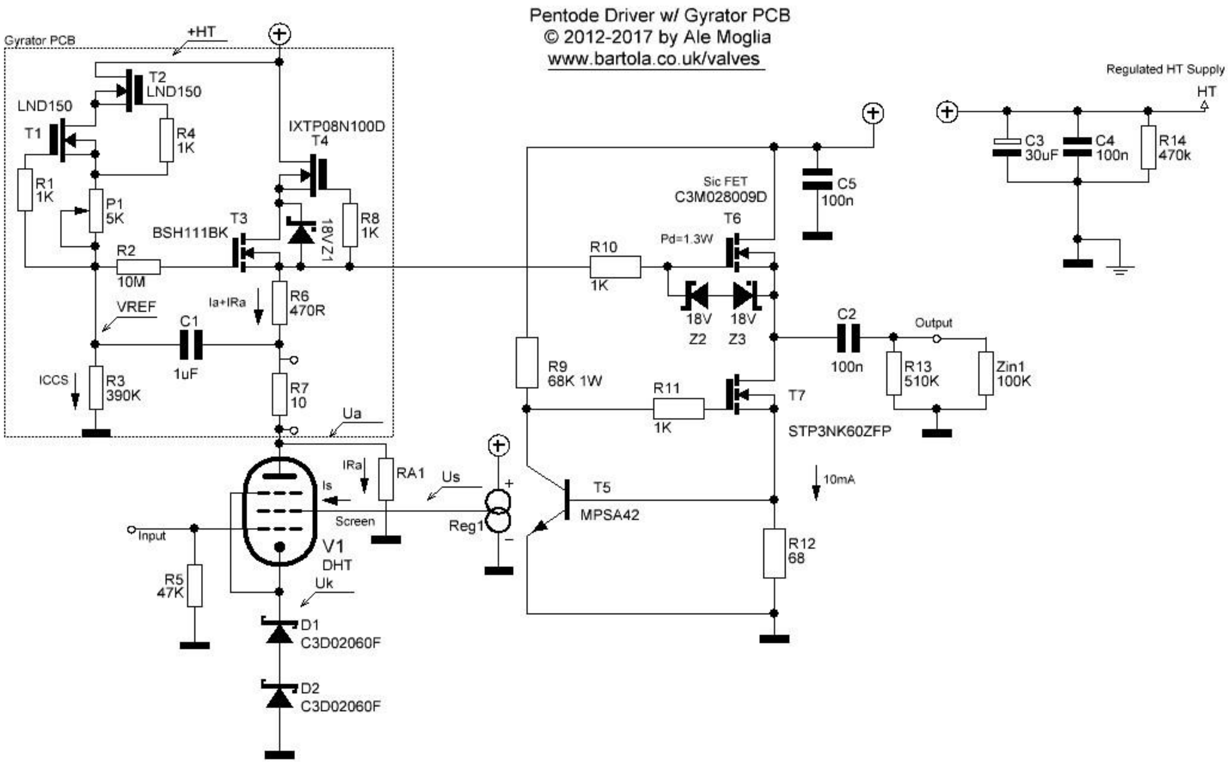


Figure 20 – Pentode gain stage with output follower buffer

I hope to soon post some other uses of this PCB including output stage drivers, LTP, parafeed stages and more! Please check www.bartola.co.uk/valves for the latest updates.

JFET OPTIONS

If you are looking for alternative options for the lower FET (either jFET or depletion/enhancement MOSFETs), below is a table of the best candidates available you can try on the board:

	BF862	BSH111BK	MMBFJ310L	BSS123N	BSS138N	2N7002	BSN20BK
Ptot (W)	0.3	0.3	0.225	0.5	0.36	0.5	0.31
VDS	20	55	25	100	60	60	60
VGS off	-1.2		-4	1.4	1	2.1	1
IDSS (mA)	25	210	60	190	230	300	265
Gfs (mS)	45	640	18	410	200	360	710
Ciss (pF)	10	19.1	5	15.7	32	13	20.2
Crss (pF)	1.9	1.5	2.5	2.1	2.8	2	2
Coss (pF)		2.7		3.4	7.3	4.1	3.1

Bear in mind that most of these FETs are SMD and will fit either on J4 or J2 pads.

ORDERING

Just send me an email to valves@bartola.co.uk or order on-line here:

http://www.bartola.co.uk/valves/for-sale/gyrator-pcb/

Bartola Ltd. UK
81 Telford Avenue
London SW2 4XN
United Kingdom

Disclaimer

The PCB and circuits described in this document use or generate potentially lethal electric currents. If you use this information to kill yourself, your friends, family members, acquaintances, total strangers, pets, electronic devices or burn down your house, it is not my problem!

www.bartola.co.uk/valves
email: valves@bartola.co.uk