

# SOURCE FOLLOWER PCB

*Build Instructions*

*Version 1.2 PCB rev 0.3*

*July 2020*



## INTRODUCTION

---

As a result of playing with DHTs, pre-amps and various circuits for many years, I realised that there was a particular need to have a flexible source follower circuit. This is nothing new and I will not even claim any proprietary design as the source follower is a universal circuit that has been out there for a very long time.

I wanted to design and build my own PCB for my designs and decided to make this available to the wider community. As with every design, there are some compromise decisions to be made to keep the complexity and cost down whilst providing the maximum flexibility.

I hope you will find this little board as useful as I had and please take the time to provide any feedback as is very helpful to all of us in the DIY Audio community.

### PCB Features

The Source Follower PCB is a very useful circuit with multiple use cases in DIY Audio. Some examples of these are:

- Amplifier output stage grid drive (e.g. A1 and A2 grid drive)
- Screen drive amplifiers
- Screen voltage stabiliser for pentode stages
- HT voltage stabiliser for preamps
- Buffer stage for high-mu/high-anode resistance stages – either triodes or pentodes (e.g. Phono)
- The board can also be adapted to work as a capacitor multiplier or electronic choke (aka gyrator) to work as part of the HT/LT filtering stage.

Some key aspects of the board are:

- The PCB has been designed to accommodate all sorts of power MOSFETs (both TO-220 and TO-247). In particular, the high transconductance and low  $C_{rss}$  devices which perform the best in this role.

- The tail CCS is simple and leverage the option of using same MOSFETs if wanted.
- The board takes into account the use of a bipolar supply up to 300V differential. To set a source follower board, you only need to set the value of 2 resistors (CCS current and operating voltage) depending on the supply voltage levels, desired current sink and make sure there is a sufficiently big heatsink on the MOSFETs.
- There is a current limiter circuit built in to protect screen or grid from excessive current. This is also very useful when the board is used as a voltage stabiliser for a preamp. You can limit the peak current and avoid destroying the top MOSFET when capacitors are charged or if accidentally the output is shorted. This circuit can be bypassed easily with a jumper instead of a resistor.
- The PCB has been manufactured to high quality levels:
  - FR4, 1.4mm thickness, ENIG finish, 2 oz. copper.
  - PCB Dimensions: 25mm x 66.04mm
    - M3 mounting holes spacing required (measured from each M3 hole center):
    - Width: 20.32mm
    - Length: 60.32mm
  - 2.5mm PCB plug connector holes
- Rev 03 addition:
  - Decoupling cap on board
  - LED indicator added

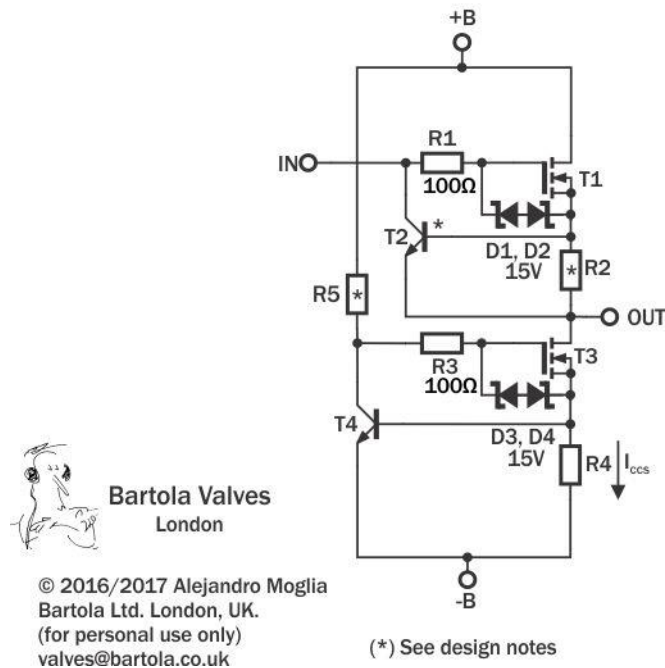
# THE CIRCUIT

---

## Overview

I'm not going to cover in detail this well know circuit. Basically, the PCB includes a source follower circuit formed by high-voltage power MOSFET (T1) which has its own protection circuit. R1 is the gate stopper and D1 & D2 are the back to back Zener diodes to protect the gate to exceed the maximum VGS voltage level. In some cases, these diodes are included in the MOSFET device so can be omitted. R2 and T2 form a current limiter to protect T1. If the current limiter isn't required, T2 can be omitted and a wire jumper can be soldered in place of R2:

## Universal Source Follower PCB Rev 02



**Bartola Valves**  
London

© 2016/2017 Alejandro Moglia  
Bartola Ltd. London, UK.  
(for personal use only)  
valves@bartola.co.uk

*Figure 1- Source Follower Circuit Diagram*

The source follower has a tail CCS instead of a source resistor which is commonly seen in older and simpler designs. This tail CCS improves the performance of the follower by providing a high impedance to the MOSFET source and minimizes the output impedance. The tail CCS is formed by T3, R4, R5 and T4. The CCS MOSFET transistor T3 has its own protection gate resistor (R3) and back to back Zener diodes (D3&D4). Again, if these diodes are included inside the MOSFET case, you can omit them. R4 sets the sink current of the CCS. R5 must be adjusted depending the operating voltage in practice. See “Design Notes” below to understand how to calculate the value of these resistors depending your needs.

In the previous release of the PCB, R1 and R3 values were 1KΩ for these gate stoppers which don't work properly with high transconductance and low Crss MOSFETS. This can result into oscillation at high frequency. This can be worsened by increasing the loop gain of the CCS by increasing the beta of T4. With the addition of the protection diodes D3, D4 as well as D1 and D2, you can use NPN transistors with lower VCEO than the original suggested high-voltage parts. You can read more on this [here](#).

The coupling capacitors and/or bias voltage setting potentiometers have been omitted from the PCB to reduce its size and provide flexibility as this PCB is not intended only to be used as part of the grid driver circuit of an output stage.

Below is the common use case of the follower circuit when driving a challenging load (e.g. output stage valve in an SE/PP amplifier):

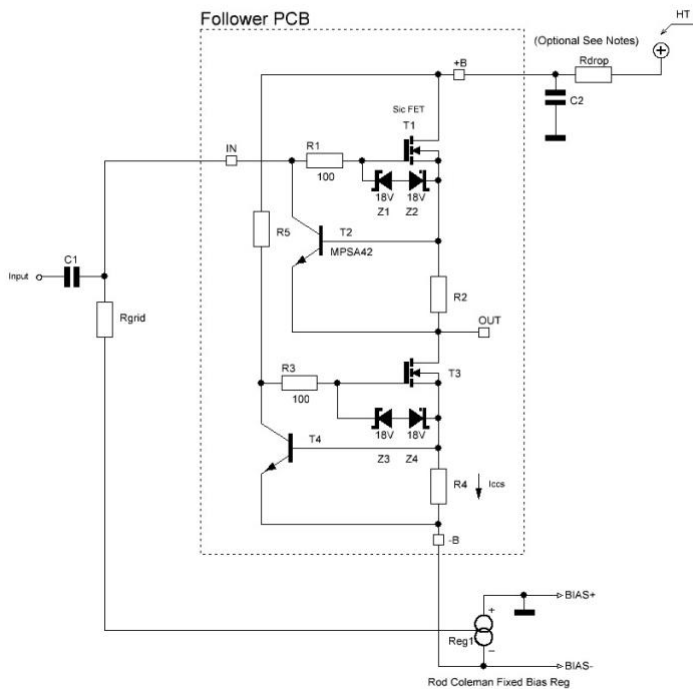
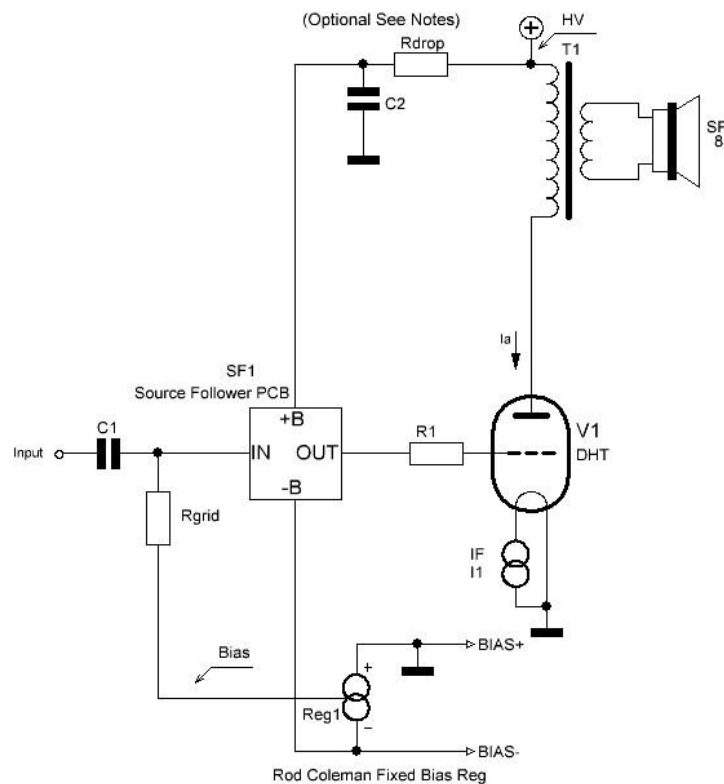


Figure 2- Source Follower – Grid Drive Example

The input is AC-coupled by C1 and biased through Rgrid. C1 needs to be calculated as it forms an LF pole in conjunction with Rgrid and the previous stage output impedance. Rgrid can be high enough (circa 1Meg) to present a light load (i.e. high impedance) to the previous stage whilst keeping the noise balance to an acceptable level. The bias of the output valve can be set via Rgrid with any external fix bias regulator. This can be as simple as a resistor divider/potentiometer with a filtering cap to something more elaborated as the Rod Coleman Fixed Bias Regulator. The -B pin is connected to the negative supply to allow sufficient headroom for T3 to swing when following the input signal.

Ideally if you have a bipolar supply for your fixed bias, you can minimize the power dissipation across the MOSFETs to reduce the size of the heatsinks. This is the price you will pay if you want to simplify the power supplies and derive the follower +B from the HT of your amp. Generally, the HT voltage will be higher than needed by the source follower and in that case you will either need to add a dropping resistor (Rdrop) to share some of the heating dissipation or provide sufficient heatsink to the MOSFETs.

Finally, C2 is highly recommended. The SF presents a high impedance node looking at the drain and is advised to add a simple 100nF film cap to help with stability of the circuit and it should be located as close as possible to the board:



## Rev03 board updates

1. Addition of R6 and LED (D9) as current indicator (see below for more details)
2. Addition of decoupling capacitor C1 (100nF Film long leads) right on the bottom of the PCB. This prevents oscillation of the SF board when long supply leads or inductive connections are created.

1. No use of LED at all. You just add a 100R resistor for R6 or place a wire jumper. If the current limiter circuit is in place (e.g. T2 and R2) then R6 is redundant. If you don't add T2 and place a wire jumper in R2 position, then R6 can prevent in-rush current or short current damage of T1 MOSFET. Worth adding it in that case.
2. Normal operation indication of the LED. In that case you can approximate the value of R6 as  $2V/I_{ccs}$ . Where  $I_{ccs}$  is the tail CCS current (explained later in this document).
3. LED turns on additional source current provided by Source Follower (e.g. A2 grid current operation). This is useful if you want to see that behavior in your boards. In that case the calculation is the same, however is  $2V/I_{peak}$ . Where  $I_{peak}$  is the current threshold you want the LED to turn on.

### Follower PCB - Rev03

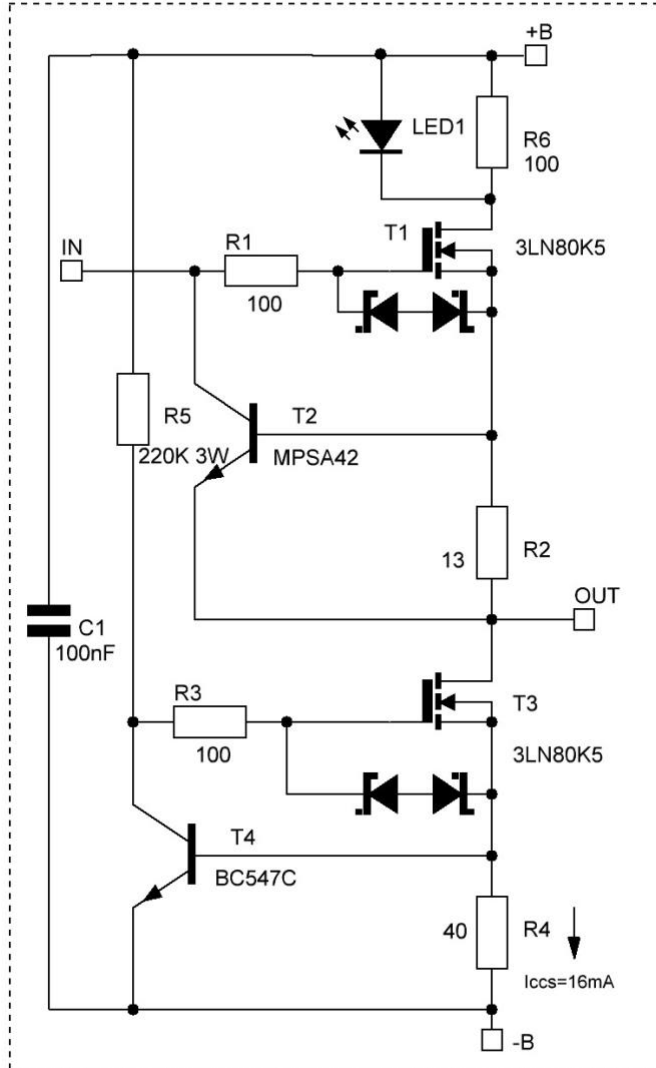


Figure 4- Source Follower – Rev03 design

### Design notes

As described above, there are just a few resistors to calculate on this circuit to fit your requirements. Below are some simple guidelines to consider when designing your source follower stage:

- The CCS sink current is set by R4. To estimate the value is simply:

$$R_4 = \frac{V_{BE(sat)}}{I_{ccs}} = \frac{700mV}{I_{ccs}}$$

where  $I_{ccs}$  is the target current.

- For the CCS operation, you should aim to run T4 at 2mA. If you look at the circuit, you will see that the value of R5 can be easily derived as:

$$R_5 = \frac{V_{+B} - V_{-B} - V_{BE(sat)} - V_{GS(th)}}{I_c}$$

Where  $I_c$  is the collector current for T4 (e.g. 2mA),  $V_{be(sat)}$  is the B-E saturation voltage for T4 (e.g. about 700mV) and  $V_{GS(th)}$  is the threshold G-S voltage for the T4. Typically,  $V_{GS(th)}$  is about 4V depending the MOSFET, so we can simplify this equation to and approximate the value of  $R_5$  to:

$$R_5 \cong \frac{V_{+B} - V_{-B} - 5V}{2mA}$$

In many cases, the value of  $R_5$  tends to fall somewhere around 50-100k $\Omega$  and you should not forget about the power dissipation requirements due to the significant voltage drop across this resistor. Normally I'd aim for a 1-3W resistor part here.

- The top MOSFET (T1) should be selected taking into account the following characteristics:
  - Low  $C_{rss}$  is needed to provide better frequency response of the stage and reduce loading to the previous stage.
  - High transconductance at the operating sink current. MOSFET have smaller  $g_m$  compared to BJT though. Higher  $g_m$  helps linearising the stage as well as increasing the bootstrapping effect of the input capacitance ( $C_{iss}$ )
  - It should handle the signal level so  $V_{DS(max)}$  should be considered.
  - To ensure the capacitances are kept to lowest level (e.g.  $C_{rss}$  and  $C_{iss}$ ), the MOSFET should have enough headroom. Check the datasheet graphs for details. This tends to be 25-50V. Therefore, you need to ensure that +B is at least 25/50V higher than the maximum peak of the output. Similarly, for -B
  - Plastic TO-220 case are ideal for isolation unless the dissipation on the MOSFET is significant.
  - SiC MOSFETs are good candidates due to low  $C_{rss}$  and good  $g_m$ . They also sound good though.
  - Should you use depletion MOSFETs instead of enhancement MOSFETs (e.g. DN2540/IXTP08N100D) consider the difference in bias when calculating the voltage drop between input and output on the SF circuit.
  - There is a list of some examples on MOSFETs at the bottom of this document
- For the lower MOSFET (T3), there is more flexibility as this is part of a CCS. Again, we want low capacitances and high transconductance ideally, but T3 is more forgiving.
- The circuit limit  $R_2/T_2$  is not active unless sufficient voltage drops across  $R_2$ . It won't degrade or affect the sound but if you prefer to remove it, it's fine. It's highly useful in an A2 drive to ensure maximum current is limited to avoid grid damage. Also, if it's used as a screen regulator or in a preamp voltage stabilizer. You can place smoothing caps after the follower and the top MOSFET will survive the charge up current of the capacitor as it's protected by this circuit. Should you decide to include the current protection circuit, then the value of  $R_2$  can be estimated as follows:



$$R_2 = \frac{V_{BE(sat)}}{I_{CCS(max)}} = \frac{700mV}{I_{CCS(max)}}$$

where the current limit is  $I_{CCS(max)}$ . Bear in mind that the circuit will kick-in when  $V_{be}$  is about 500mV, so it's an approximation. A more elaborate fold-back option could be added here, but I didn't see the real need for it.

## 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 to help you select the components for the right operation of this board in your circuit). The high-quality PCB will arrive as a plain PCB looking like this:



*Figure 5-SF Board as it arrives*

The first step is to solder the 2mm connectors to the PCB. If you will solder the cables directly, then you can skip this step:



*Figure 6-Step 1: solder the PCB 2mm sockets*



The next step is to solder all resistors. Make sure you leave space underneath R5 to help with the heat dissipation:



Figure 7-Step 2: solder the resistors

**Note:** In the above picture, R3 and R1 should be  $100\Omega$  instead of  $1K\Omega$  shown. If you're not planning to use the current limiter protection circuit, then you need to omit T2 and place a wire jumper across R2 as shown below:

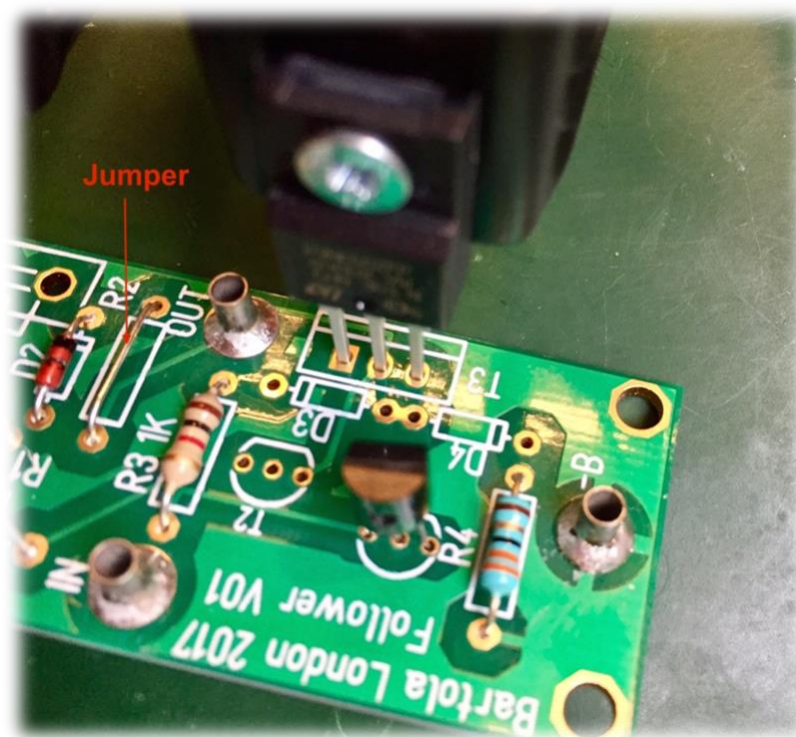
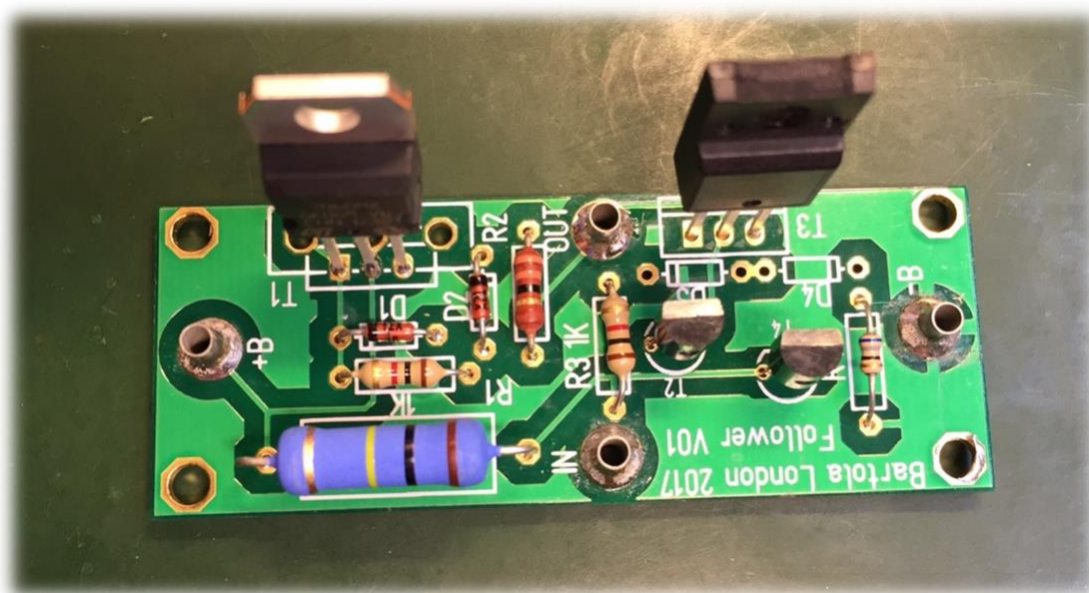


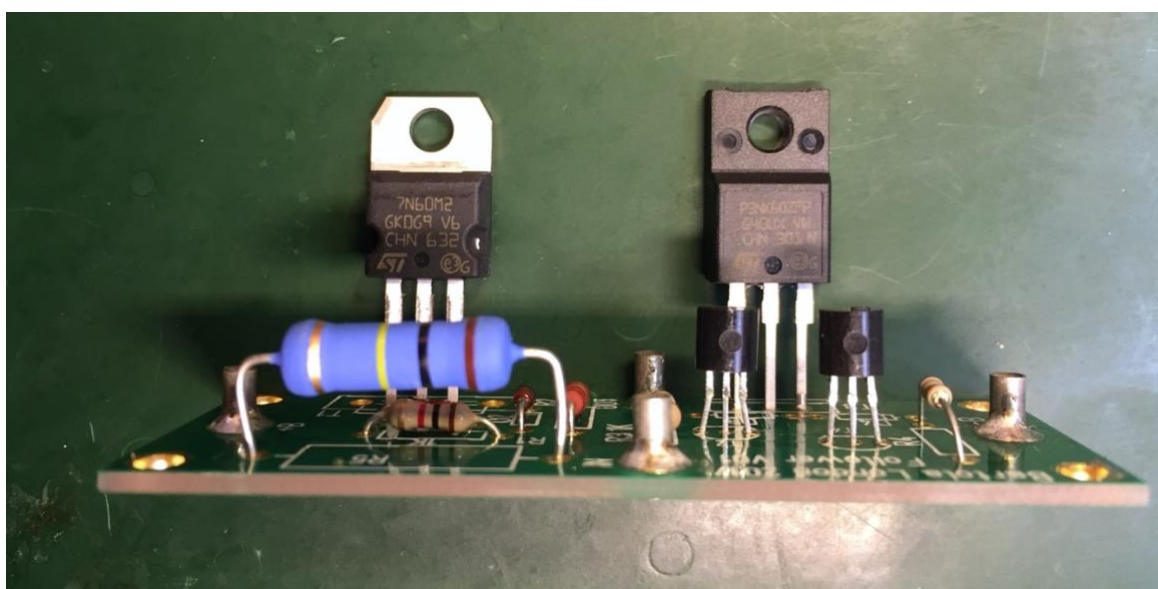
Figure 8-Step 4: Placing the jumper across R3 when current limiter is not used

Now as next step you can proceed with placing the protection Zener diodes (as needed subject to the MOSFET device being used) and the NPN transistors. Finally, you can solder the MOSFETs T1 and T3. Make sure you adjust the lead height based on the heatsink requirements. Generally, a clip-on heatsink will be ok for up to 2W. The TO-220 pads for T1 are not at the edge of the PCB board due to the provided pads for TO-247 devices (e.g. SiC MOSFETs) which are located at the edge. You should consider the pin length to ensure they can be bended slightly to reach out a bigger heatsink (or top plate) if required:



*Figure 9-Step 3: solder the resistors*

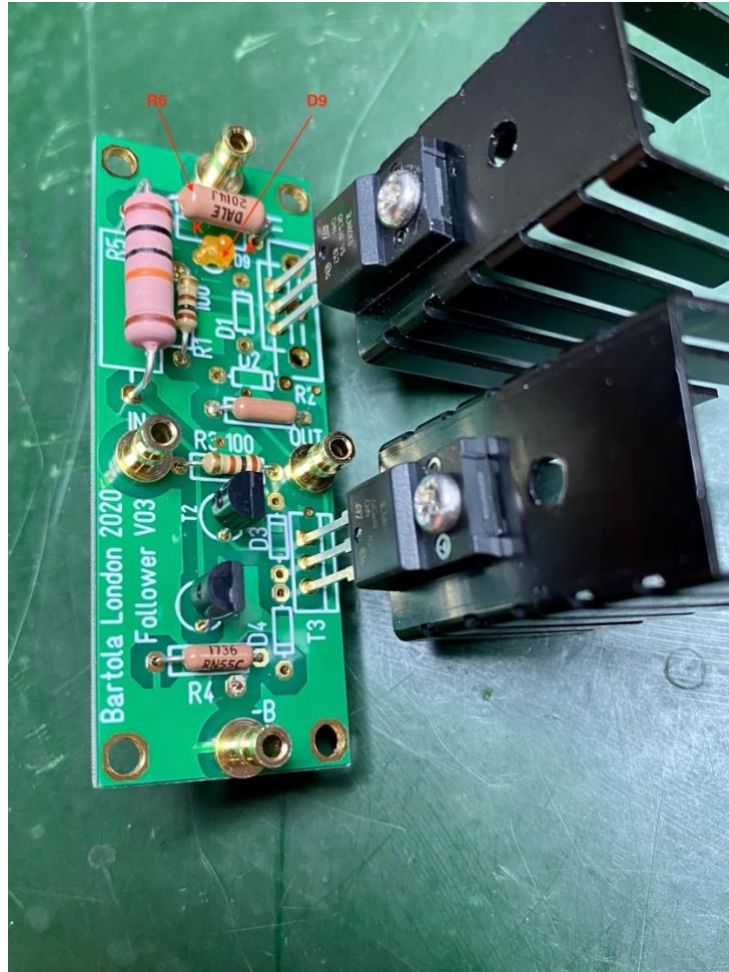
Make sure you mount T4 with short leads. Longer leads in T4 may lead to oscillation of the tail CCS. A completed board should look like the one below. In this example, I included current protection (T2 and R2) fitted, a 3NK60 MOSFET for T3 (which has protection diodes built-in) and a 7N60M2 for T1 (which doesn't have protection diodes so D1 and D2 are included):



*Figure 10-Completed SF board*

## Rev03 Build additions

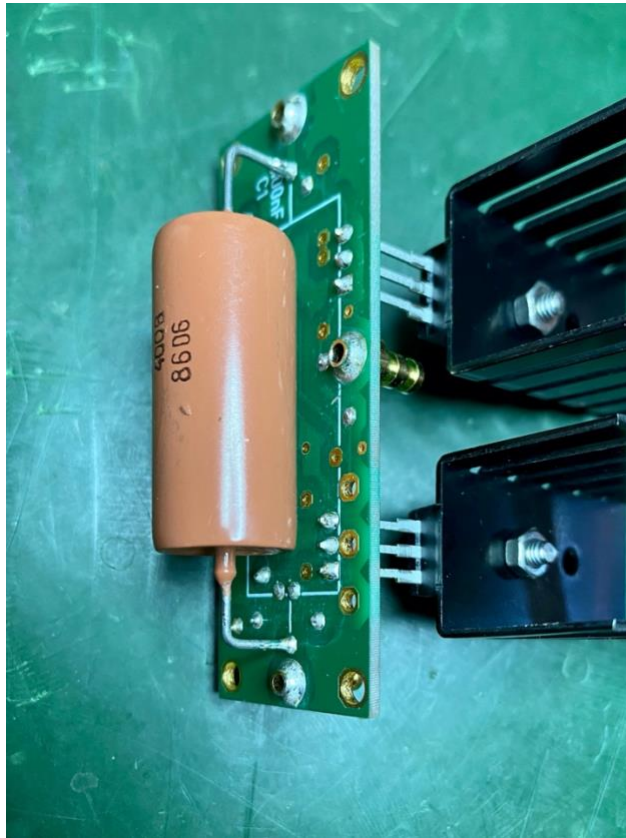
With the addition of D9 and R6 to Rev03 PCB, you will only have to pay attention to the LED cathode orientation as it's not marked on the PCB. The cathode is marked on the below picture where the pin is closer to R1 and R5. The anode is closer to D1 and T1:



*Figure 11-Rev03 update: added LED and R6*

Also, C1 should be fitted last:





*Figure 12-Rev03 update: added decoupling cap C1*

In some cases, I have soldered in parallel to C1 a 100uF/250V electrolytic capacitor. This may only be needed in extreme cases when full decoupling is needed in case of oscillation.

### Fitting the board

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

The boards can be stacked, provide sufficient space for the MOSFET and their heatsinks

## 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. In particular, if the clip-on heatsinks aren't isolated. The TO-220 devices have the drain connected to the metal tab.
- I'd strongly recommend using a VARIAC to test the PCB or a variable +B supply.
- Use a 47K-100K $\Omega$  dummy load resistor instead of the valve, don't avoid this step. Slowly bring the +B up and check the voltage across the dummy resistor. You should be able to

adjust the bias voltage by adjusting your bias pot. The output voltage should be the bias voltage minus the  $V_{GS(th)}$  which is around 4V.

- If you decided to omit the current limiter circuit (T2 and R2) and you accidentally short the output, you will damage T1 which will result in a drain to source short and +B voltage present at the output. You will see full +B at the output. Change T1 immediately.
- Once voltage setting at the dummy resistor is achieved, you can replace the dummy resistor with your real circuit and test again. It should work like a charm!
- Don't omit C2 (or C1 in Rev03) if you experience any HF oscillation. If push comes to shelf, you can add a ferrite bead to the gate and drain leads. Also try keeping short leads where possible. As said before, I have soldered in parallel to C1 a 100uF/250V electrolytic capacitor. This may only be needed in extreme cases.
- Any short at the output is likely to burn out R6 and destroy D9 LED due to excessive current. This is generally cheaper and easier to replace rather than changing the MOSFET T1.

# CIRCUIT EXAMPLES

## 01a DHT Preamp with SF output

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.
- Filament resistor for filament bias and Rod Coleman's regulator
- Output coupling Cap (220nF) of your preference as well as input resistor R1
- The gyrator board
- The source follower board

Here's the latest circuit implementation, although it may seem complex at first glance, it's highly simplified thanks to the 3 PCBs used: gyrator, source follower and Rod Coleman filament regulator.

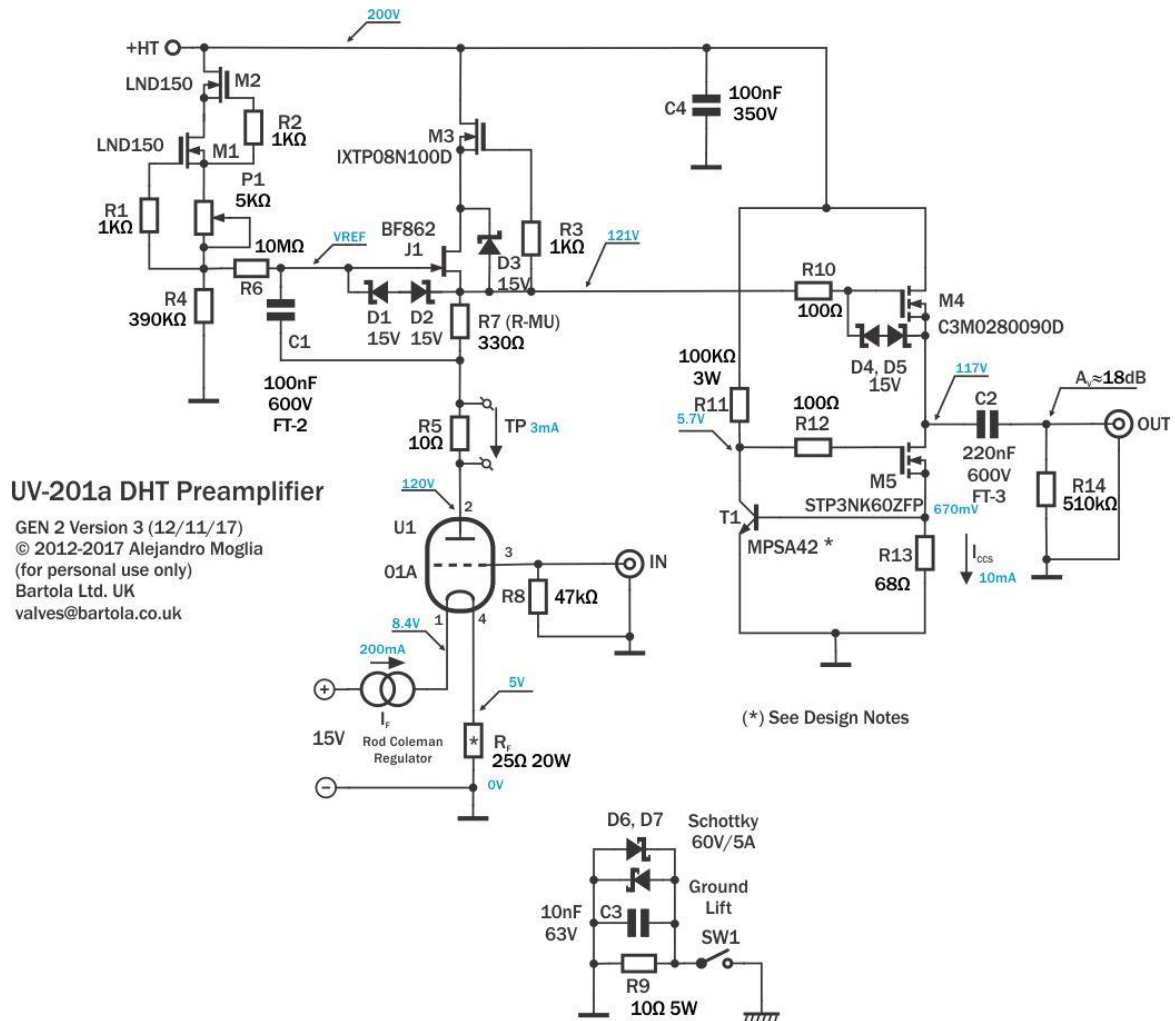


Figure 13-UV-201a DHT Preamp with Source Follower output buffer

In general, you can use the SF board with any triode or pentode (DH or IH) which has low anode current when driving demanding loads. Here is the generic diagram:

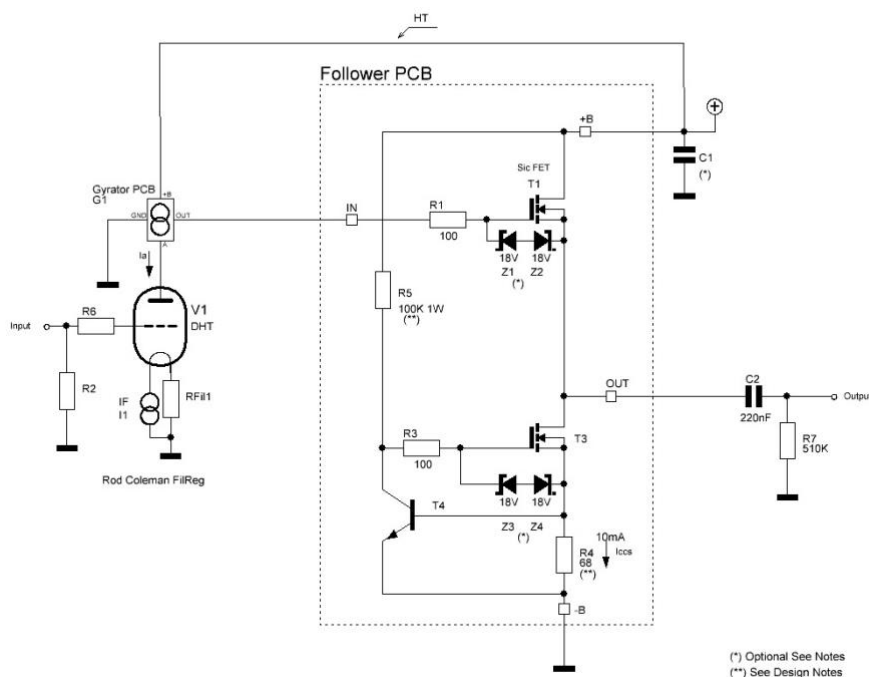


Figure 14-Adding a SF board to a low anode current driver

The frequency response of this stage is improved as a higher impedance load is presented to V1 coupled via DC. C1 is needed to keep it stable in some cases. You can set R4 to 68Ω for 10mA CCS current. You can increase the CCS current up to 20mA or more. This will provide a lower output impedance and improved performance of the SF. Penalty is higher power dissipation across T1 and T3. The available driving current will avoid any “slew-rate” issues which result in the loss of treble.

Just as a taster, here is an example of the measured performance of a 6SF5GT triode driver which with a gain of 40dB. The bandwidth and low distortion of this circuit could not be achieved without the SF stage when driving an output stage like 300B:



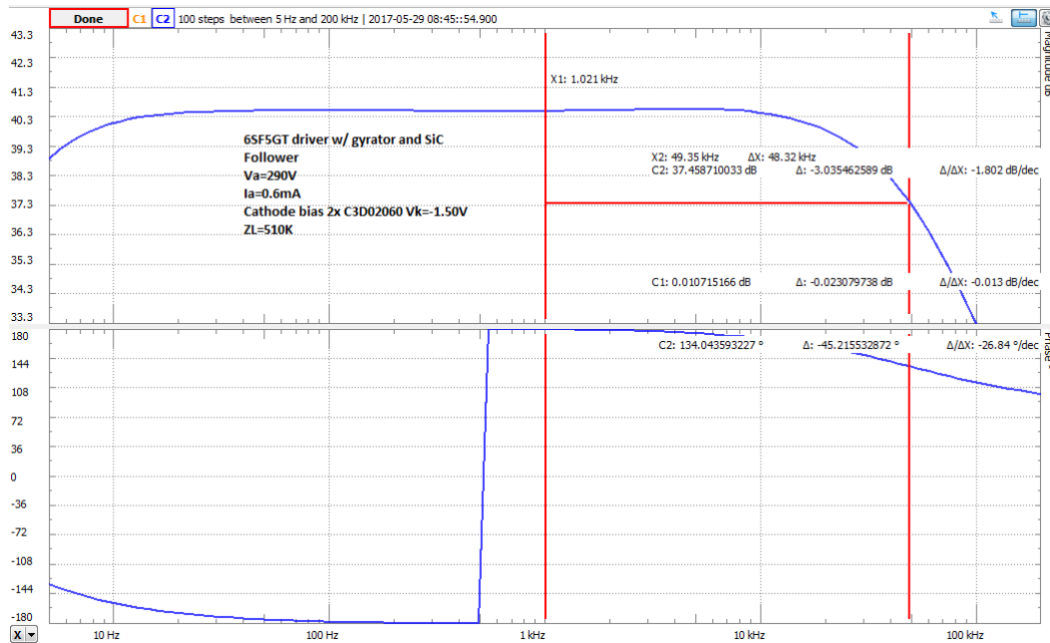


Figure 15-Frequency response of a high-gain triode driver using 6SF5GT

The following example combines two use cases of the SF board. Here is a single-ended amplifier with plate to grid feedback (a.k.a “Schade” type feedback in the DIY audio community). The intention is to provide an overall view on how useful the SF board can be in multiple designs. The design of this amplifier is not trivial and won’t be covered in this example:

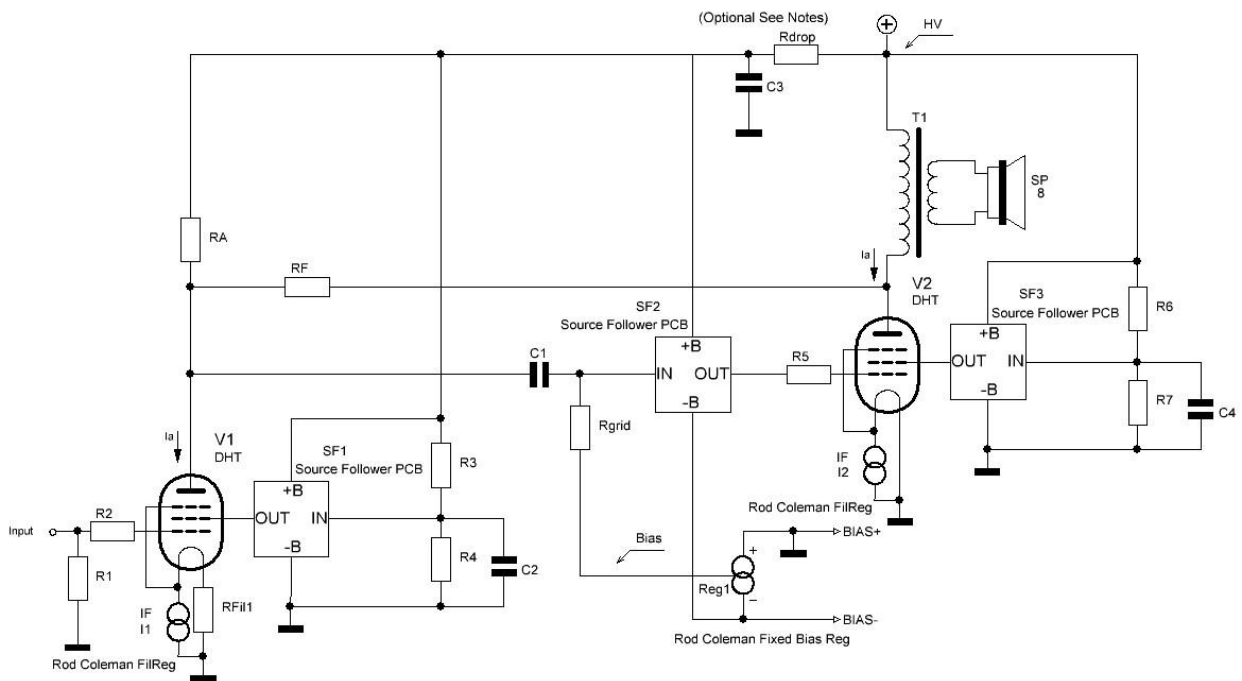


Figure 16-Local feedback SE Amplifier with Pentode driver and output stage (DHPs)

The SF boards SF1 and SF3 are used to regulate the screen voltage. R3/F4 and R6/R7 provide the screen voltage (including the VGS(th) of T1 in each case). C2 and C4 are smoothing caps and work to form cap multiplier smoothing circuit in conjunction with the SF board. SF2 serves for 2 purposes.

Similarly, you can use a variety of IHP on the previous circuit. Here is how it would look:

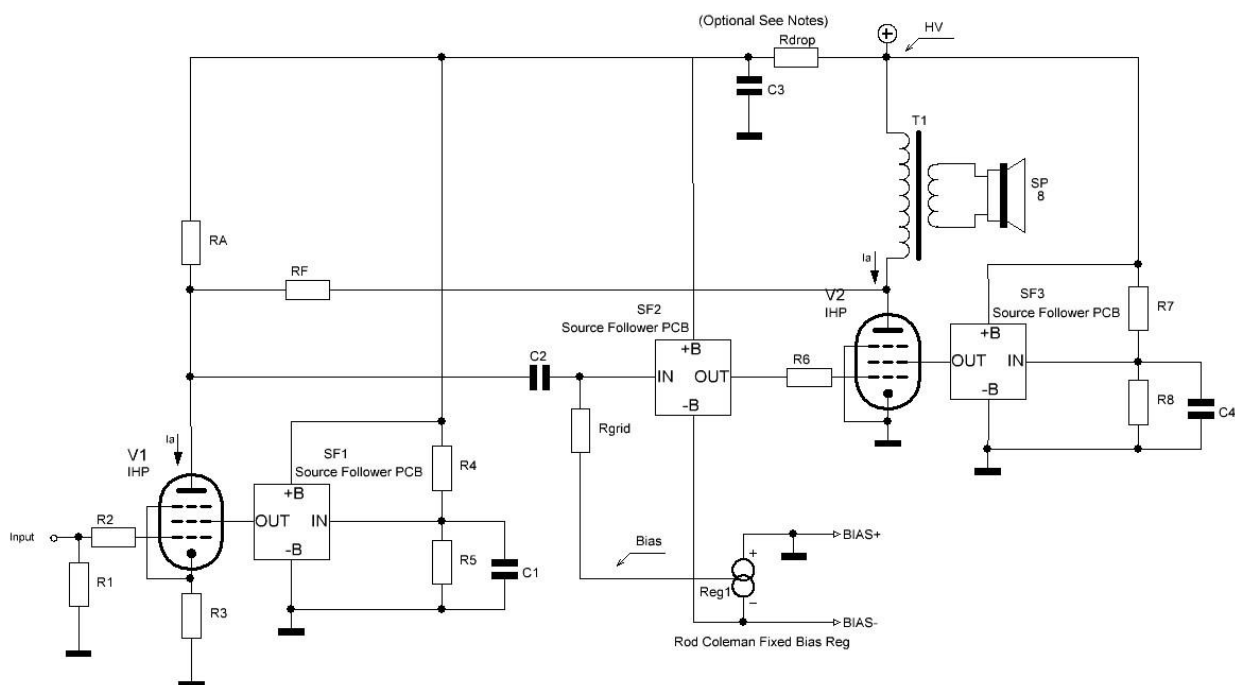


Figure 17-Local feedback SE Amplifier with Pentode driver and output stage

Keep an eye on my blog ([www.bartola.co.uk/valves](http://www.bartola.co.uk/valves)) as I will hopefully continue to post further examples in the future.

# MOSFET OPTIONS

Below are some examples of the MOSFETs to be considered. The below list is based on my own experience and should be consider as restrictive. MOSFET transconductance (Gfs) is estimated at 10mA from the datasheet values and should provide an indication of which MOSFETs may perform better than others when bootstrapping Ciss. Crss is also highlighted as we want the minimum possible for best high-frequency response. The values quoted are average and in normal conditions as specified by the manufactures. Capacitances assume sufficient VDS level. Check datasheets for details:

	C3M0280090D	FQPF2N60C	TP3NK60Z/FP	TF7N60M2	STF13N60DM2	STP4N150
Type	SiC <sup>2</sup> NMOS	NMOS	NMOS	NMOS	NMOS	NMOS
Package	TO-247-3	TO-220	TO-220	TO-220	TO-220	TO-220
Ptot(W)	54	23	20	20	25	160
VDS <sub>max</sub>	900	600	600	650	600	1500
VGS <sub>thr</sub>	2.1	4	3.75	3	4	
ID <sub>max</sub> (A)	11.5	2	2.4	5	11	4
Gfs(mS)(*)	3600	5000	1800			3500
IdGfs(mA)	7500	1000	1200			2000
Id(mA)	10	10	10			10
Gfs(ms)(**)	131	500	164			247
Ciss(pF)	150	180	311	271	730	1300
Crss(pF)	2	4	8	1	1	12
Coss(pF)	20	20	43	15.7	70	120
Rsource(kohm)	10000	10000	10000			10000
Cin(pF)	2	4	8			12
	AOT1N60	DPF5N50NZ	IRF820	RFB812PbF	IXTP08N100D	DN2540N5-G
Type	NMOS	NMOS	NMOS	NMOS	Depletion <sup>3</sup> NMOS	Depletion <sup>3</sup> NMOS
Package	TO-220	TO-220	TO-220	TO-220	TO-220	TO220
Ptot(W)	41.7		50	78	60	15
VDS <sub>max</sub>	600		500	500	1000	400
VGS <sub>thr</sub>	4.1			4		
ID <sub>max</sub> (A)	1.3		2.5	3.6		
Gfs(mS)(*)	900	3540	1500	7600	560	325
IdGfs(mA)	650	2250	1500	2200	400	100
Id(mA)	10	10	10	10	10	10
Gfs(ms)(**)	112	236	122	512	89	103
Ciss(pF)	130	330	360	810	325	200
Crss(pF)	1.8	8	92	7.3	6.5	1
Coss(pF)	14.5	50	37	47	24	12
Rsource(kohm)	10000	10000	10000	10000	10000	10000
Cin(pF)	1.80	8.00	92.00	7.30	6.50	1.00

Notes:

(\*) Device transconductance at stated drain current (Id Gfs) from the datasheet

(\*\*) Calculated transconductance at operating drain current (Id)

I have provided the equivalent bootstrapped Cin (pF) for a given transconductance (gfs) at a specific drain current (Id) and with an equivalent source resistance. This will provide an indication of the


capacitive load presented to the driver by the MOSFET in a similar operating condition. Obviously the higher the transconductance and the lower the reverse capacitance is, the great bootstrapping of the parasitic capacitances leading to a lower total input capacitance ( $C_{in}$ ).

$$C_{in} = C_{rss} + \frac{C_{iss}}{g_{fs} \cdot R_{source}}$$

## Rev03 update

After experimenting with few MOSFETs and due to EOL of some devices I have settled for the following exceptional MOSFET which has 3 key advantages:

1. TO220 plastic package, so the tab is isolated from drain!
2. Has protection diodes built in, so no need to buy and add 4 extra Zener diodes (D1 to D4)
3.  $C_{rss}$  is extremely low at 0.1pF!



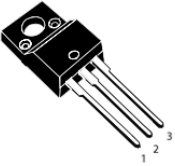
# STF3LN80K5

---

N-channel 800 V, 2.75  $\Omega$  typ., 2 A MDmesh™ K5  
Power MOSFET in a TO-220FP package

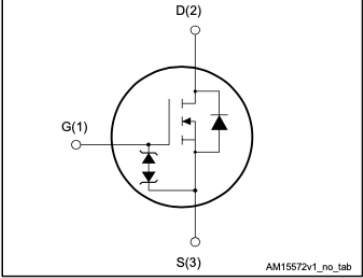
---

Datasheet - production data



TO-220FP

Figure 1: Internal schematic diagram



AM15572v1\_no\_tab

### Features

Order code	$V_{DS}$	$R_{DS(on)}$ max	$I_D$
STF3LN80K5	800 V	3.25 $\Omega$	2 A

- Industry's lowest  $R_{DS(on)}$  x area
- Industry's best FoM (figure of merit)
- Ultra-low gate charge
- 100% avalanche tested
- Zener-protected

### Applications

- Switching applications

### Description

This very high voltage N-channel Power MOSFET is designed using MDmesh™ K5 technology based on an innovative proprietary vertical structure. The result is a dramatic reduction in on-resistance and ultra-low gate charge for applications requiring superior power density and high efficiency.

**Table 1: Device summary**

Order code	Marking	Package	Packing
STF3LN80K5	3LN80K5	TO-220FP	Tube

## HEATSINKS

The below are good options to consider for MOSFET heatsink:

<https://www.digikey.co.uk/short/z4rt2d>

For higher dissipation you should consider this one:

<https://www.digikey.co.uk/product-detail/en/aavid-thermal-division-of-boyd-corporation/504222B000000G/HS104-2-ND/5833>

It can do 6.4C/W which is really good. As I mounted it backwards it fit a TO-247 as well if needed instead of a TO-220 device is used.

## ORDERING

---

Just send me an email to [valves@bartola.co.uk](mailto:valves@bartola.co.uk) or order on-line here:

<http://www.bartola.co.uk/valves/for-sale/source-follower-pcb/>

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

# *Disclaimer*

Bartola and Bartola Valves Logo are trademark of Bartola Ltd.

Bartola Ltd. is a limited company registered in England and Wales. Registered number: 10819140. Registered office: 81 Telford Avenue, London, SW2 4XN, United Kingdom.

All schematics and designs are copyright Alejandro Moglia and Bartola Ltd. unless stated otherwise. All rights reserved. You are welcome to build the circuits presented here for your own personal entertainment. You may NOT build from information on this document for commercial profit without a royalty agreement with the author in place.

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!

**©2010-2020 Bartola Ltd. London, UK.**

[www.bartola.co.uk/valves](http://www.bartola.co.uk/valves)

email: [valves@bartola.co.uk](mailto:valves@bartola.co.uk)