INTRODUCTION

After 6 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)
  - DHT output stage drivers (e.g. 300B, 45, 10Y, 211/845, 813, etc.)
  - Push-pull drivers in LTP configuration
  - Phono output stages
  - Parafeed stages
- 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 or similar
- Various options in the PCB to fit low noise and capacitance JFET like either
J309/J310, 2SK170/LSK170 or BF862 / MMBJ310 SMD devices. This will provide a wide range of anode currents with great HF response.

- Anode current test point across a 10R resistor
- PCB characteristics:
  - 70x65mm board on a high-quality 1.6mm thick FR4 board
  - ENIG finish, 2 oz. copper.
- Rev 06 update:
  - Additional back to back Zener protection for the lower JFET allowing depletion and also enhancement (MOSFET) devices to be used
  - There are additional pads for a protection Zener to be placed between drain and source

Overview

I’m not going to cover in detail this well known 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
- 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 cascaded pair J1 and M3 through R6. D1, D2 and D3 protect J1. 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.

The low frequency response is primarily driven by the RC pair R6 and C1. Typically, I’d use the following combinations: 4.7M 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 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 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 500μA, so the value of R4 is roughly the output voltage divided this reference current. For simplicity, the following guidance is provided:

<table>
<thead>
<tr>
<th>R4</th>
<th>Anode Voltage (Va)</th>
</tr>
</thead>
<tbody>
<tr>
<td>220K</td>
<td>100-120V</td>
</tr>
<tr>
<td>330K</td>
<td>120-160V</td>
</tr>
<tr>
<td>390K</td>
<td>160-200V</td>
</tr>
</tbody>
</table>

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.
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’s hard to know its value. From my measurements and simulations, I came up with a simple table which will help you:

<table>
<thead>
<tr>
<th>J1</th>
<th>Rmu</th>
</tr>
</thead>
<tbody>
<tr>
<td>2SK170</td>
<td>1K5</td>
</tr>
<tr>
<td>BF862</td>
<td>330-470R</td>
</tr>
<tr>
<td>J310</td>
<td>1K-1K5</td>
</tr>
</tbody>
</table>

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!

Here is the generic DHT preamp circuit model. See the “Circuit Examples” section for practical examples:
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 version in which case you should look at the end of this section) looking like this:
The first step in building the gyrator is to 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 on 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.

Once you have completed the resistors you can add the protection Zener diodes D1, D2 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 3 - Build step 2: Protection Zener diodes D1, D2 and D3

Figure 4 - Build step 2: Trimmer P1

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 3: LND150 CCS devices

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’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 like is shown in the picture above.

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

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](image)

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

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!
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**

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. Use a 22k / 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.

Once this 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 preference as well as input resistor R1
- The gyrator board and you’re done!

Here’s the latest circuit implementation:

![DHT Preamp (w/ gyrator)](image)

*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:

![Diagram of 26 DHT Preamp with Gyrator Load](image)

Figure 14 – The 26 DHT preamp with gyrator load

4P1L DHT Preamp

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 just 30mA:
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 15 – The famous Siberian preamp (4P1L)

Figure 16 – Adding and output buffer stage
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:

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

Bear in mind that most of these FETs are SMD and don’t fit with the BF862 pin-out. So you will have to solder them at 45 degrees. This isn’t easy at all! You’ve been warned!

ORDERING

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

All schematics and designs are copyright Alejandro Moglia 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!

©2012-2017 Bartola Valves, London
www.bartola.co.uk/valves  email: valves@bartola.co.uk