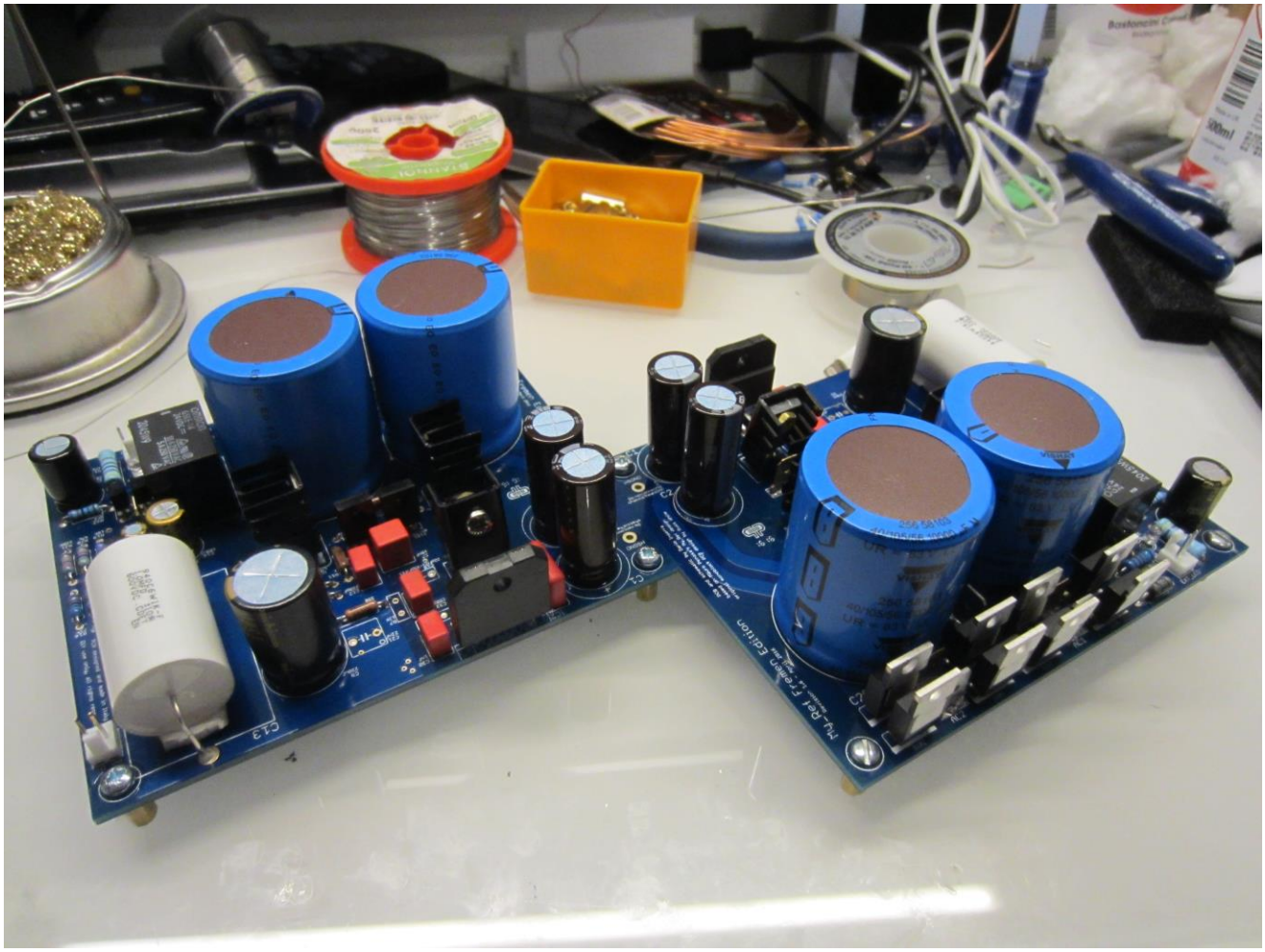
My_Ref Fremen Edition

Build Guide



CONTENTS

1 REVISIONS TABLE

1.1 Document's history

Version	Released	Author	Approved by	Comments and Notes
1.0	13/4/2013	D.Inserra		First revision
1.6	10/7/2016	D. Inserra		Update to include 1.5 and 1.6 boards
1.61	11/7/2016	D. Inserra		Some small fixes
1.62	25/8/2016	D. Inserra		Replaced a pair of pics with better ones, some small fixes. Added fixes for 1.10 and 1.7 boards.
1.63	30/8/2016	D. Inserra		Added LM317s heatsinks notes
1.7	21/8/2017	D. Inserra		Grounding, HSGND, Safety, grub screws

1.2 Associated documents

Version	Released	Author	Origin	Document's Title

2 INTRODUCTION

The My_Ref Fremen Edition (My_Ref FE or FE from now on) is an unofficial variation of Mauro Penasa's My_Ref and it's distributed under Mauro's kind permission.

The big part of info available on the My_Ref applies also to the FE, which has the following goals:

- a more stable and performing amp using SMD parts
- higher LM318's PS voltage thanks to a voltage limiter
- better (regulated) shunt power supply for LM318

I've designed a brand new PCB with these features:

- bigger caps (16mm diameter with both 7 and 5 mm pads) for C1, C2, C9
- large use of SMD parts
- quite all small caps and some critical resistors are still through-hole
- very small paths thanks to SMD
- ground planes/microstrip design for higher signal integrity
- zener limiter based on schematic 5b from this link (which seems pretty similar to My_Evo one)
- CCS shunt PS for the LM318
- An alternate C9/R10 arrangement
- Different grounding
- Different compensation
- A double diode bridge like in most Gainclones

You can find references on development, beta and release candidate here:

My_Ref Fremen Edition - need help on PCB evaluation - diyAudio
My_Ref Fremen Edition - Beta build/Fine tuning - diyAudio
My_Ref Fremen Edition RC - Build thread - diyAudio

You can find references to the original My_Ref here:

My "Audiophile" LM3886 Approach
Circuit Description

In this document we will illustrate how to build the My_Ref Fremen Edition using the final boards and the recent major revision ones (from v1.5 and up).

2.1 Revision 1.5 boards

Rev. 1.5 boards introduced some important changes:

- Thanks to an improved layout noise radiated from diode bridges is greatly reduced so distortion is reduced from (circa) 0.001% to 0.0001%
- No more carbon films (well... apart R2)
- Only one Caddock left (R3)
- Mouser BOM costs less and sounds better
- More options for through hole resistors, Z-foils pads
- Smoothing caps are now PMG-SI from Vishay/BC, much more balanced than old Panny TS-HA

The new BOM leads to a greater level of detail, sounds more balanced/neutral, very clean and musical.

Mouser BOM is a tiny bit laid back, timbre with upgrade to Z-Foils will be perfectly transparent and further detail is gained.

Older boards can be easily upgraded to almost the same level (apart distortion) using the updated BOM for pre v1.5 boards.

2.2 My_Evo Rev. A Compensation

Both boards families can apply My_Evo Rev. A compensation.

My Evo is the not public evolution of the My_Ref made by the original designer Mauro Penasa.

This updated compensation will give:

- Improved stability on capacitive loads (the system now accepts the presence of any capacity directly at the output terminals without stability limits)
- Changing the decay of the damping factor versus frequency. The Rev A introduces more damping at medium and low frequencies.
- Change of the DC operating point active network integration. The high-pass third-order granted (Q> 0.7) is replaced with a previous "double-first order", or by an active integrator that works with the more classic high pass time constants next to the DC (> 0.1Hz) represented by the input coupling capacitor

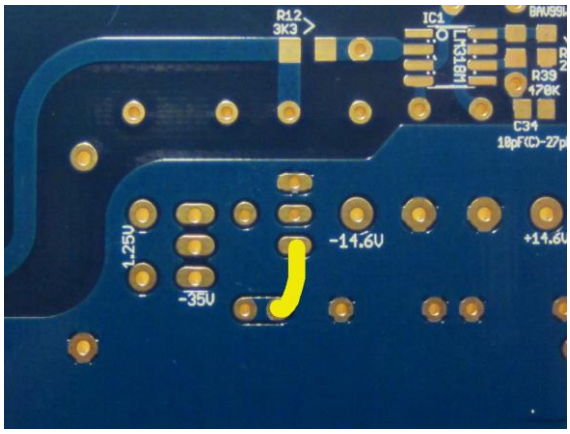
Note for whom bought boards with SMD parts already soldered

Those boards came with FE compensation; you must remove R39 (470K) to use the My_Evo Rev. A one.

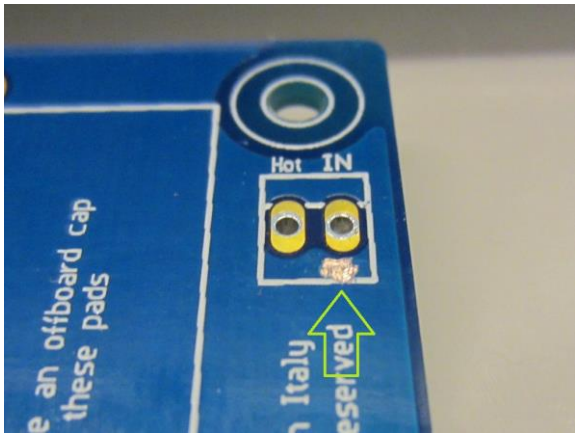
2.3 Boards with problems

Two boards revisions (1.10 and 1.7) had minor problems easily fixed soldering small jumper wires.

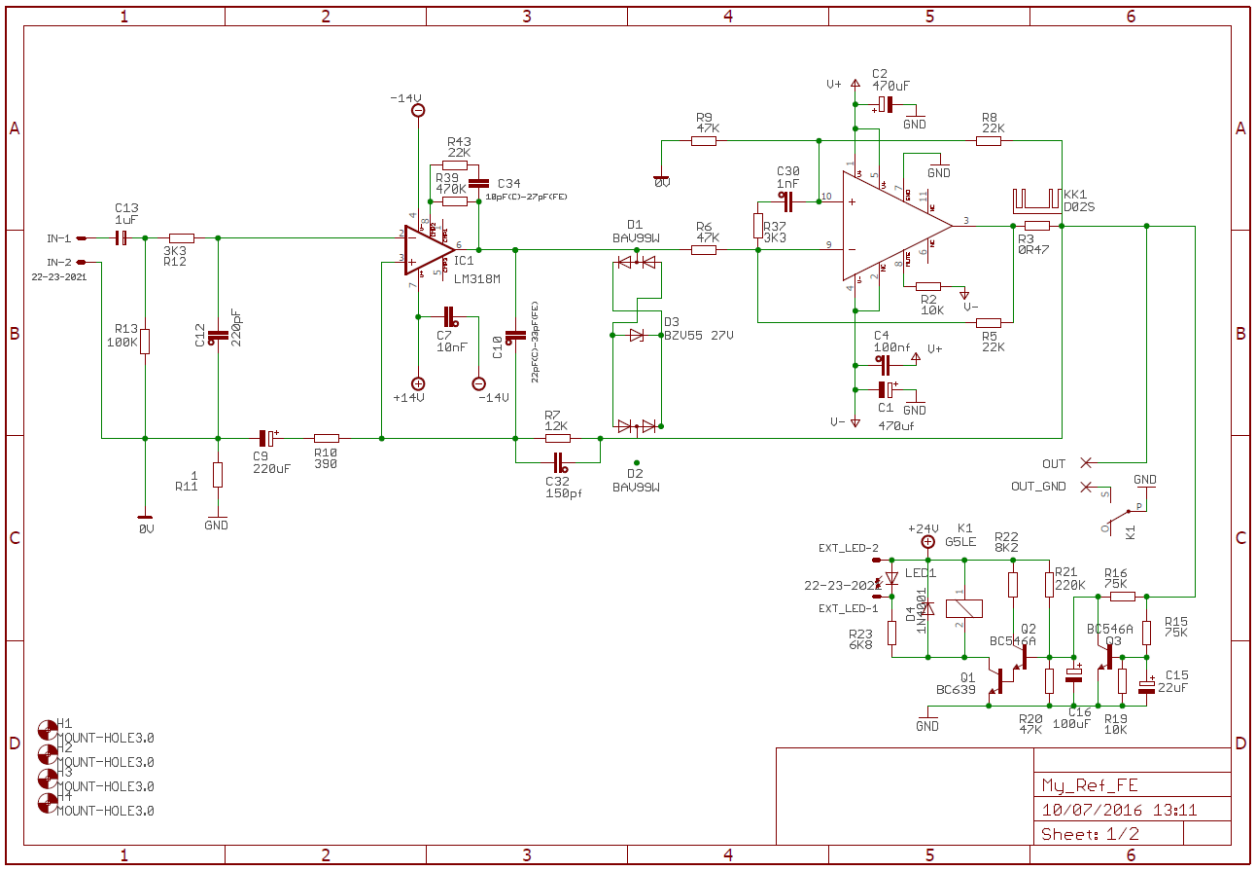
V1.10 boards

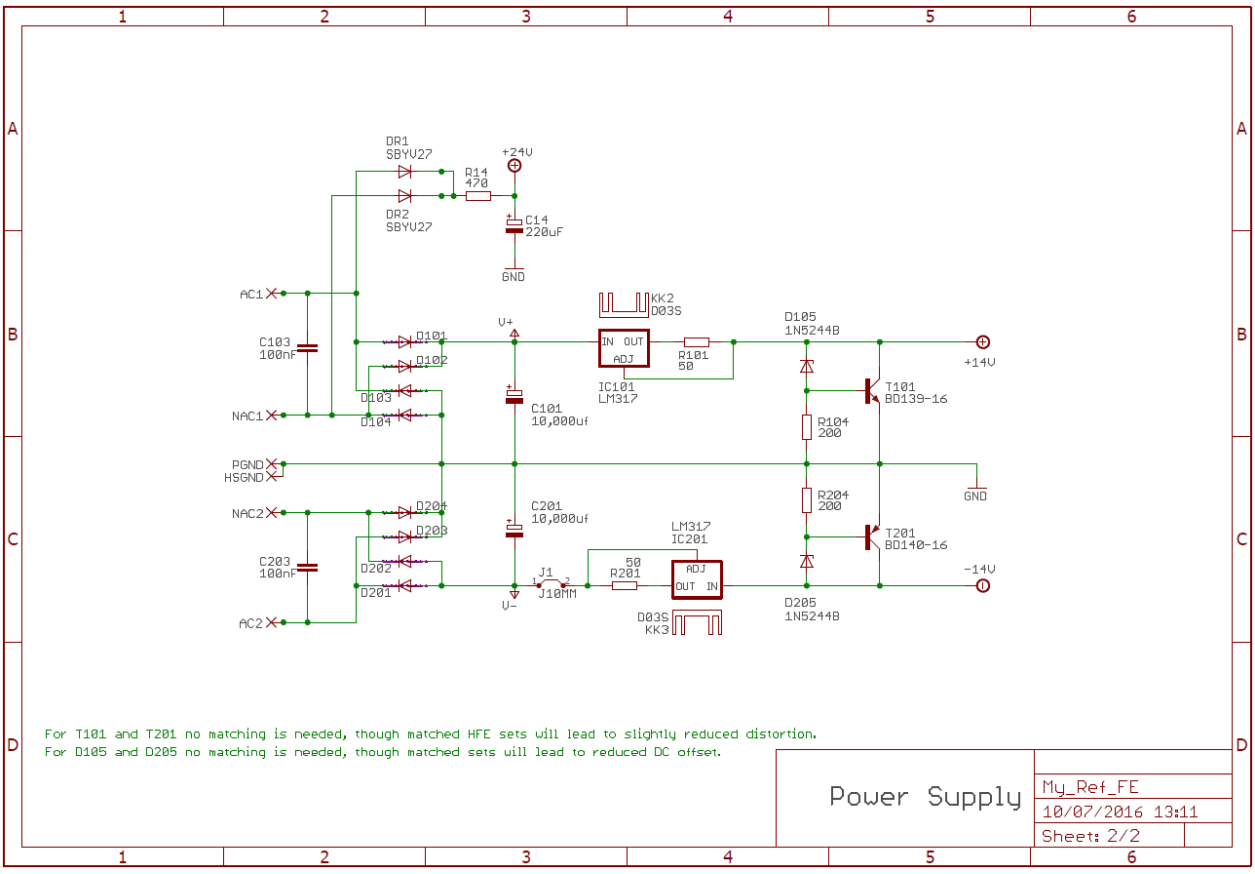


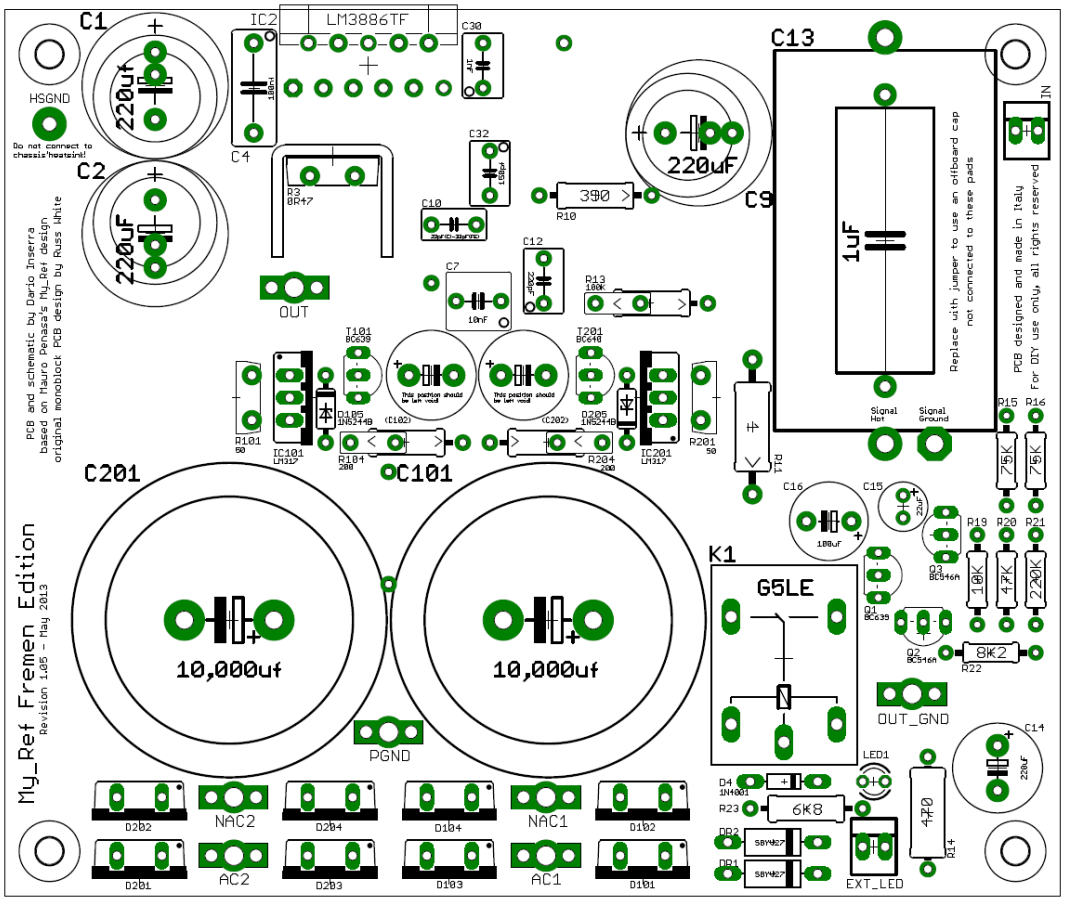
V1.7 boards

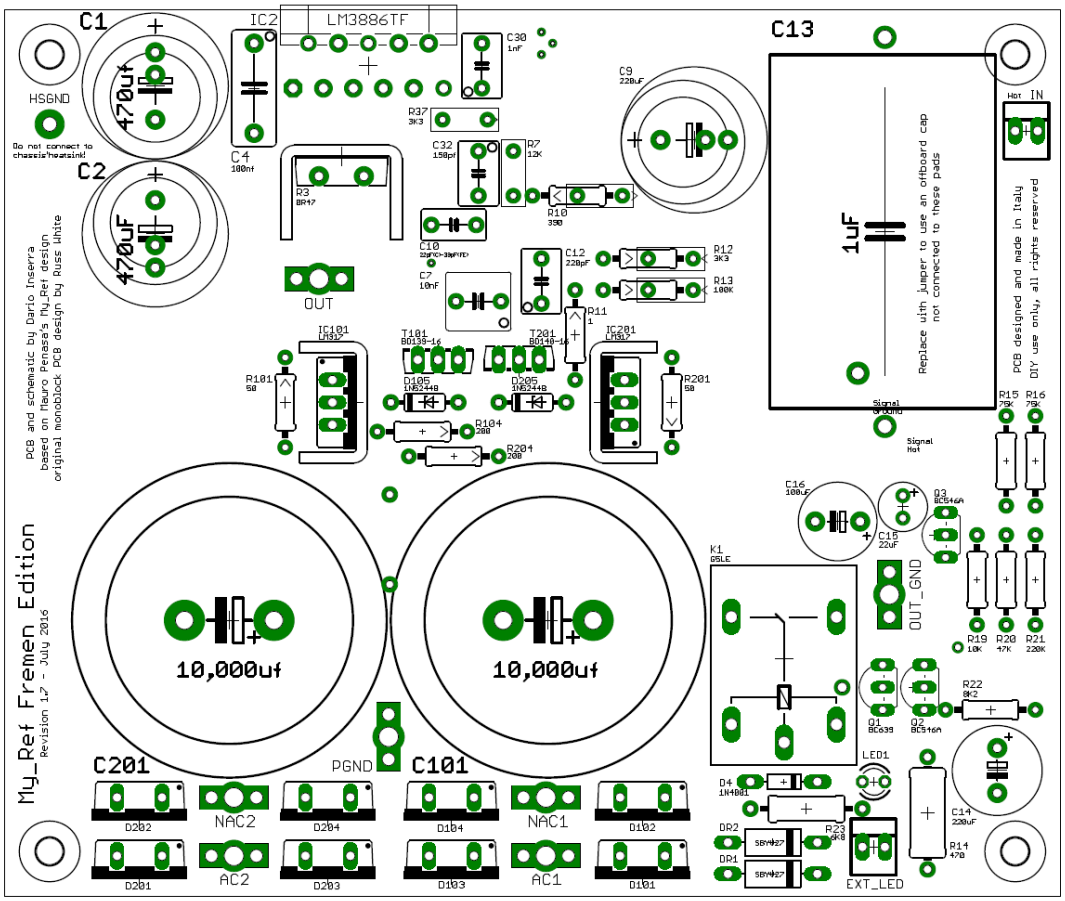


3 SCHEMATIC AND SILKSCREEN



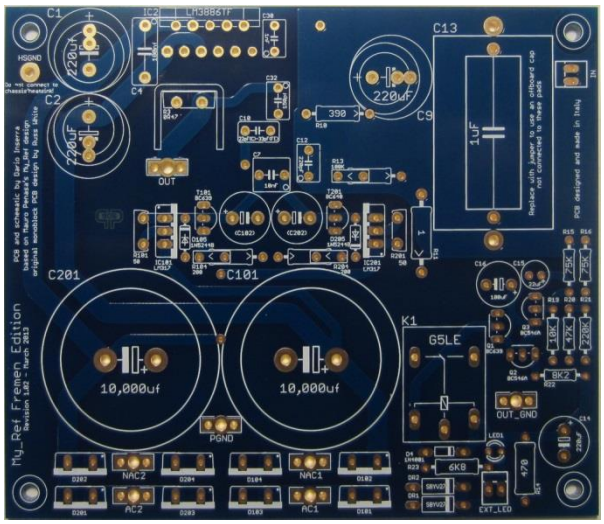


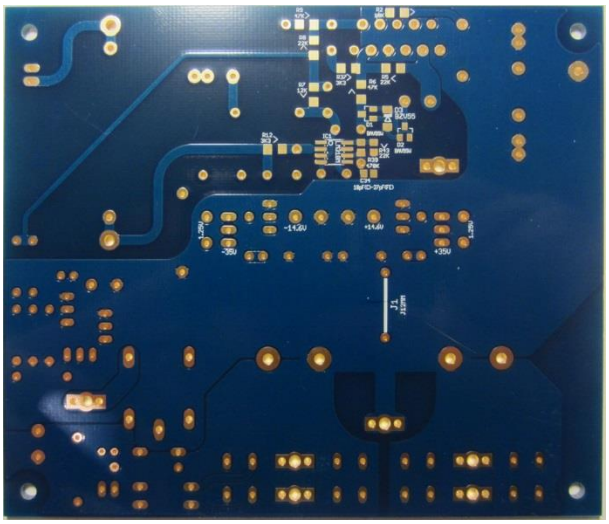

C1
220uF
C2
220uF
HSGND
Do not connect to chassis/heatsink!
IC2
LM3886TF
C30
C4
100nF
C32
150pF
R3
8R47
OUT
C10
C7
10nF
C12
220pF
R10
390
220uF
C9
C13
1uF
IN
Replace with jumper to use an offboard cap not connected to these pads
Signal Hot
Signal Ground
PCB designed and made in Italy
For DIY use only, all rights reserved
PCB and schematic by Dario Inserra
based on Mauro Penasa's My_Ref design
original monoblock PCB design by Russ White
My_Ref Fremen Edition
Revision 1.05 - May 2013
T101
BC639
T201
BC640
D105
1N5244B
D205
1N5244B
R101
50
R104
200
R204
200
R201
50
IC101
LM317
IC201
LM317
R1
C201
C101
10,000uf
10,000uf
PGND
K1
G5LE
C16
100uF
C15
2u2
R15
75K
R16
75K
R19
10K
R20
47K
R21
220K
Q3
BC546A
Q1
BC639
Q2
BC546A
8K2
R22
OUT_GND
C14
220uF
LED1
D4
1N4001
R23
6K8
DR2
SBYW27
DR1
SBYW27
470
R14
EXT_LED
D202
NAC2
D204
D104
NAC1
D102
D201
AC2
D203
D103
AC1
D101


C1
470uF
C2
470uF
HSGND
Do not connect to chassis/heatsink!
IC2
LM3886TF
C30
1nF
C4
100nF
R37
3K3
C32
150pF
R7
12K
R3
8R47
R10
390
C9
220uF
OUT
C10
C7
10nF
C12
220pF
R12
3K3
R13
100K
R11
1
C13
1uF
Hot IN
Replace with jumper to use an offboard cap not connected to these pads
Signal Ground
Signal Hot
PCB designed and made in Italy
DIY use only, all rights reserved
PCB and schematic by Dario Inserra
based on Mauro Penasa's My_Ref design
original monoblock PCB design by Russ White
My_Ref Fremen Edition
Revision 1.7 - July 2016
IC101
LM317
T101
BD139-16
T201
BD140-16
IC201
LM317
D105
1N5244B
D205
1N5244B
R101
50
R104
200
R204
200
R201
50
C201
C101
10,000uf
10,000uf
PGND
C16
100uF
C15
22uF
Q3
BC546A
K1
G5LE
OUT_GND
R19
10K
R20
47K
R21
220K
Q1
BC639
Q2
BC546A
R22
8K2
C14
220uF
LED1
D4
1N4001
DR2
SBYW27
DR1
SBYW27
R23
4K8
R14
470
EXT_LED
D202
NAC2
D204
D104
NAC1
D102
D201
AC2
D203
D103
AC1
D101

4 YOU WILL NEED

Apart My_Ref FE boards





- soldering iron
- thin solder (0.5mm euthectic silver solder)
- tacky flux
- tweezers
- a good sight or a magnifier

I suggest to use a eutectic solder alloy (63sn/37pb or better 62sn/36pb/2ag) for optimal results (it goes directly from solid to fluid).

For leaded works I use 0.8mm WBT-0820 silver solder 4% for TH and 0.5mm Stannol 62pb/36sn/2ag for SMD.

For lead free works the ROHS counterparts will do.

Good quality lead free solder usually will ‘sound’ a bit more transparent and apparently thinner, in reality it’s leaded one than will add some ‘weight’ to the sound.

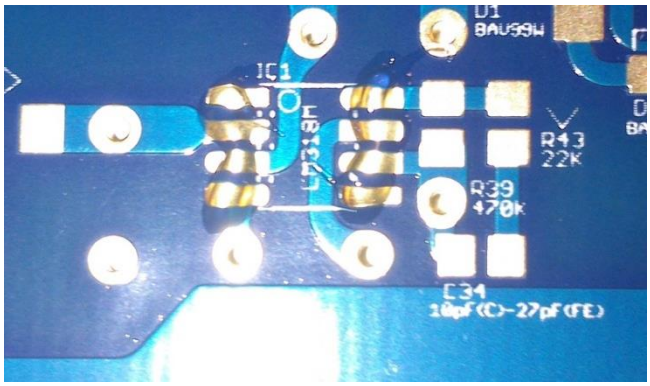
Oyaide SS-47 is an excellent lead free choice for through hole works.

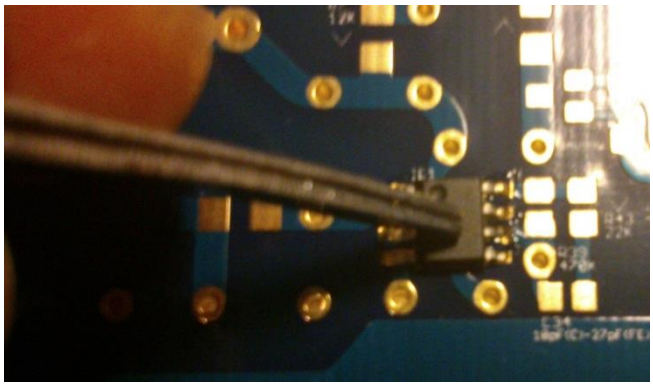
5 POPULATING BOARDS

5.1 SMD Parts

Start cleaning boards with alcohol.

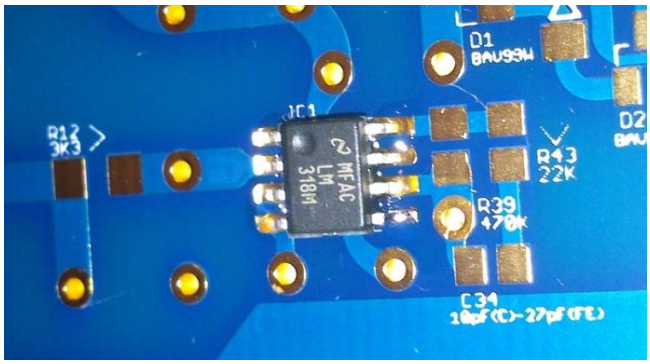
The first component we'll put in place is the LM318, apply flux to pads and then position the LM318 (take care to pins to pad alignment) and keep it in place with tweezers:

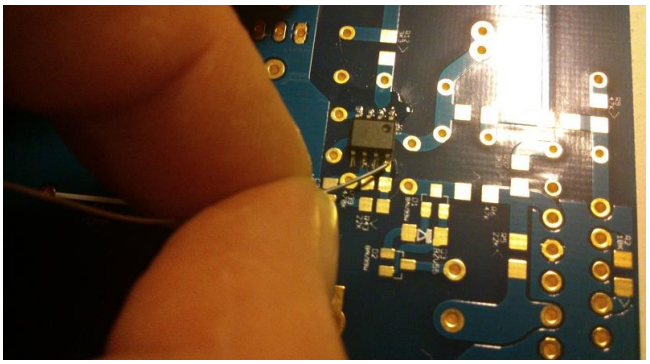




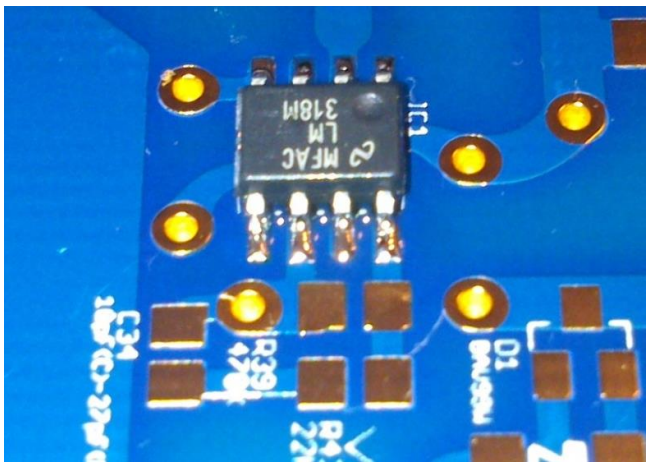
Apply the wet solder tip to one of the pins (the bottom one on the right is NC so it's a good one to use), now that the LM318 is kept in place by the soldered pin we solder all the others.

Just apply the solder and solder tip for 1-2 seconds



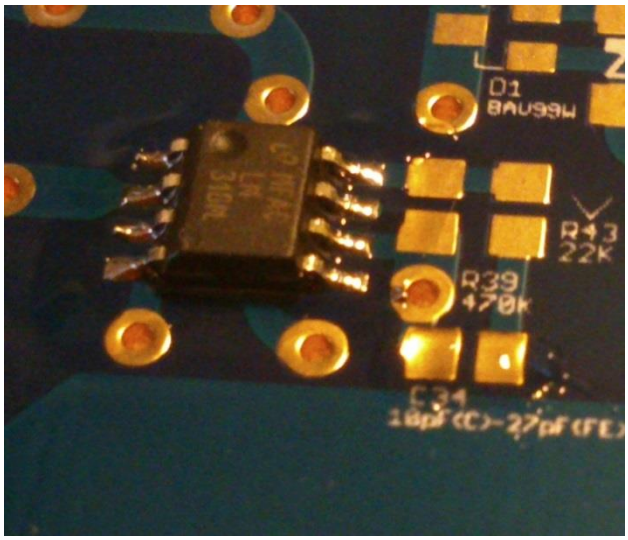


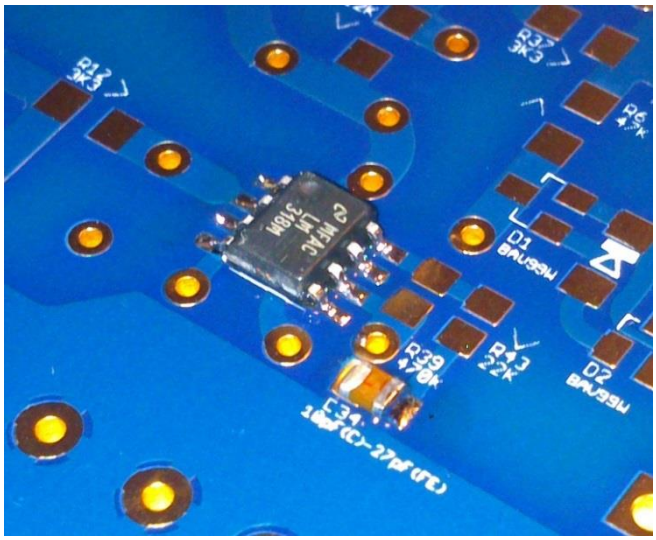
this should be the result:



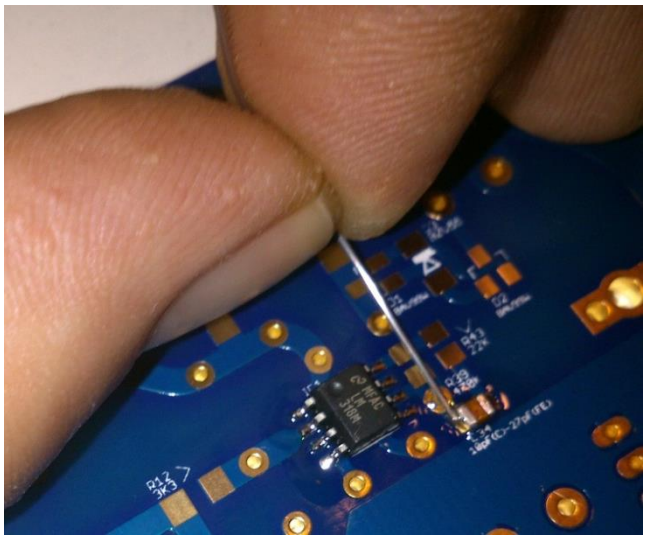
We then proceed with 0805 compensation resistors and cap, starting from C34.

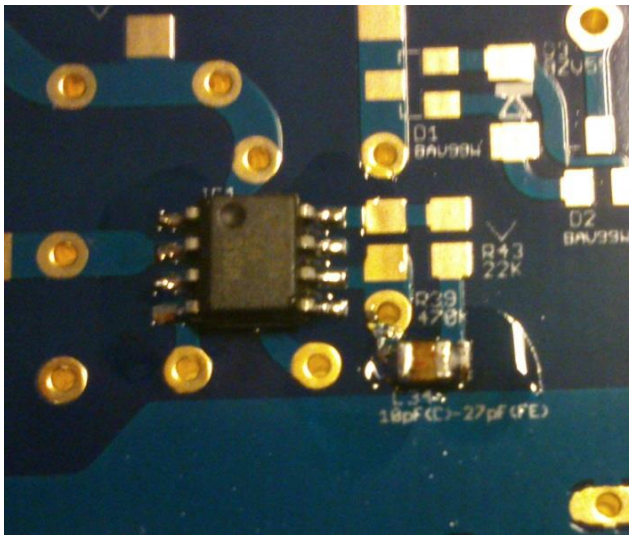
Apply flux, position the component and while keeping it in place with tweezers apply the wet tip to one side.





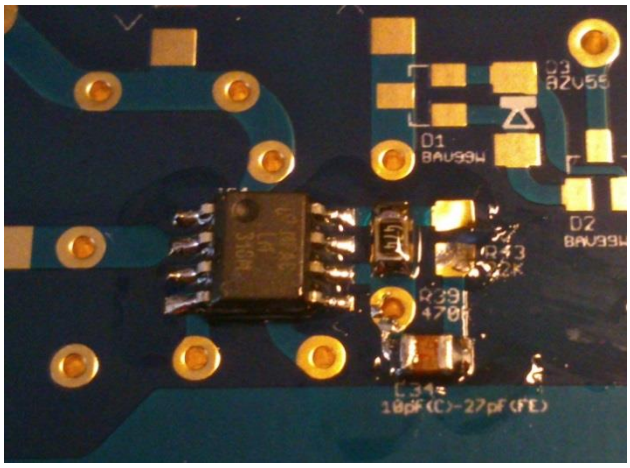
Then apply solder and tip to the other end

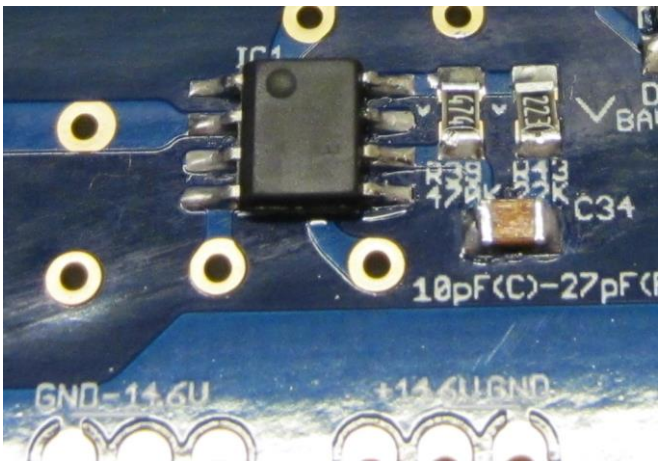




Now it's resistors turn, please note that resistor's markings should follow the tiny arrow marked on PCB.

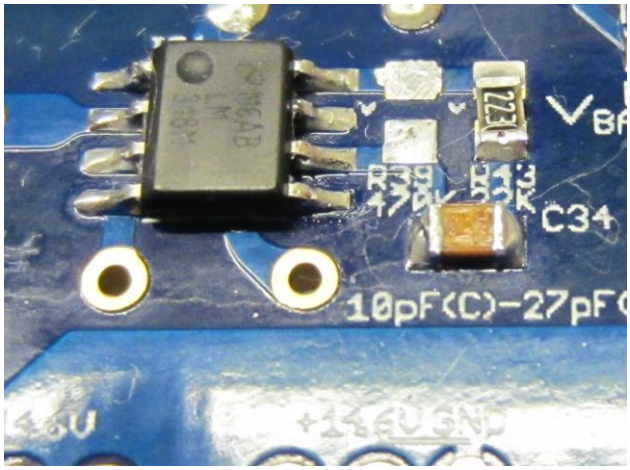
After applying flux to R39 pads we solder it like we did with C34, same technique for R43.



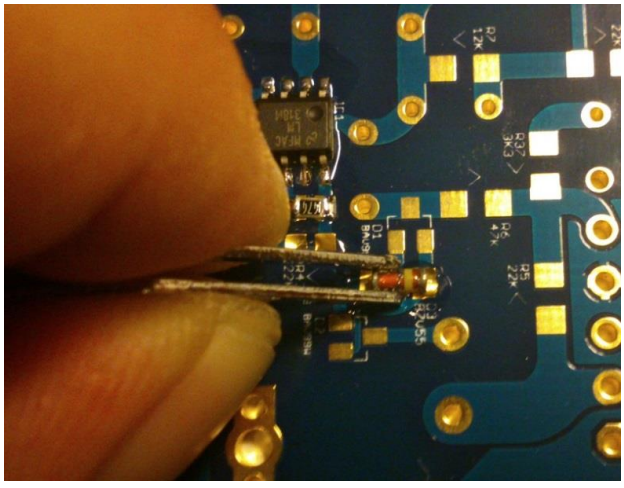


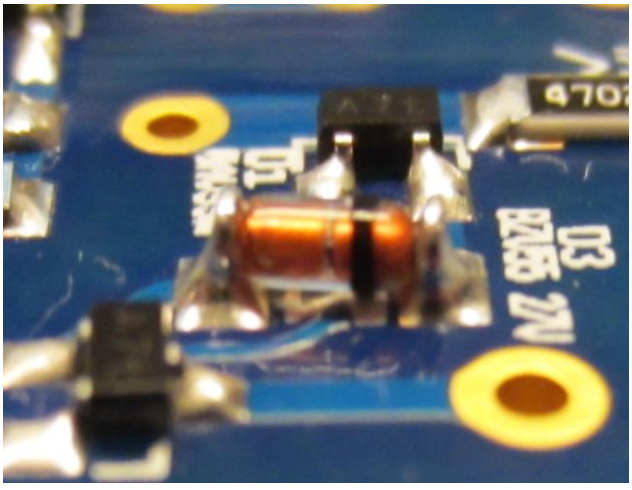
NOTE

If you're going to use My Evo Rev. A compensation R39 should be left unpopulated

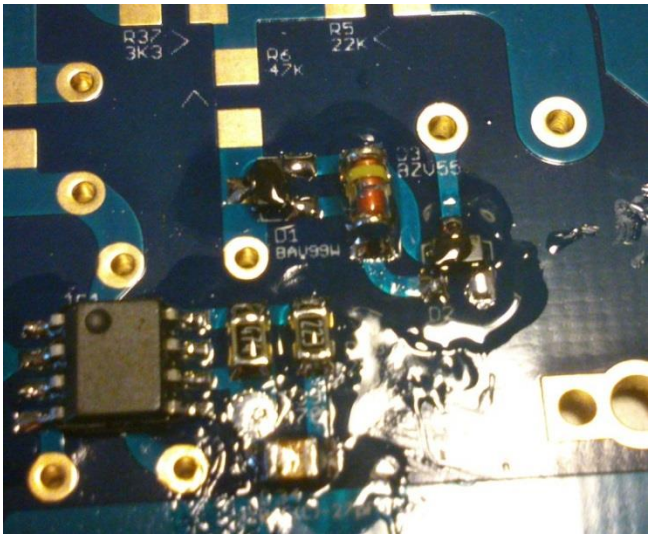


The next component will be the MELF diode D3; as usual apply flux, keep it in place using tweezers and apply the wet tip to one end (for MELF you will need a well wet tip). Apply solder and tip to the other end.

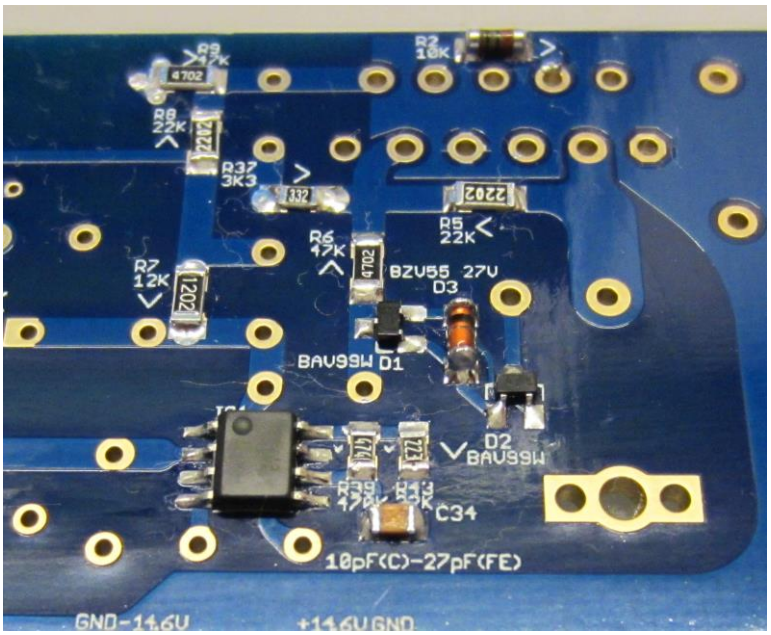


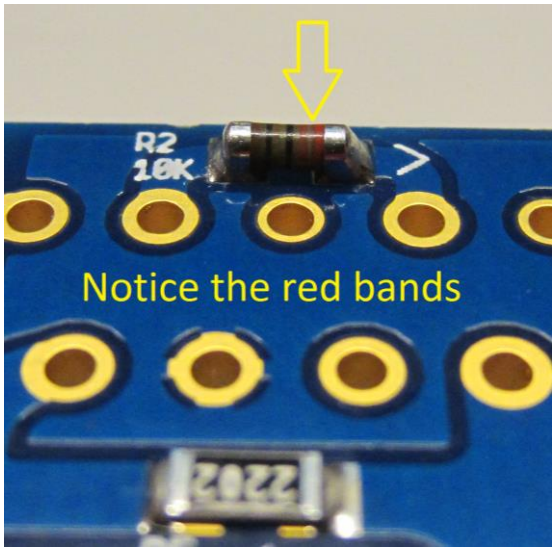


Like we did for LM318 we now proceed with D1 and D2 (flux, solder one pin while keeping the part in place with tweezers, solder the other pins)



Now the we know how to solder SMD parts and made some experience, we proceed with all other resistors (pay attention to orientation)





Note that 3K3 resistors are 0805 ones soldered on 1206 pads since Mouser do not stock 1206 3K3 Susumus so in BOM 0805 ones are indicated, later BOMs could include 1206 parts.

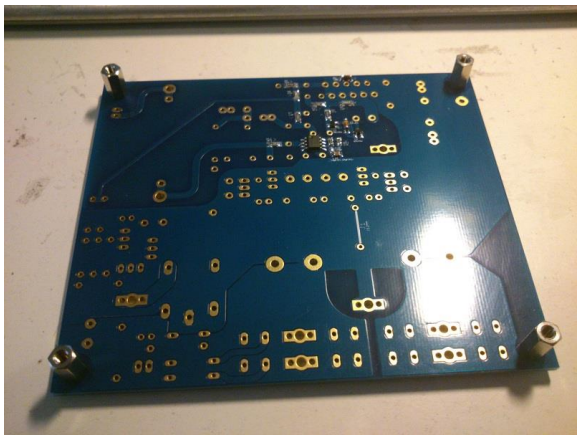
Note:

Since we used a lot of flux, it is a good idea to do an intermediate cleaning with isopropyl alcohol or a gentle solvent and a brush to remove it.

5.2 Through hole parts

5.2.1 Low profile

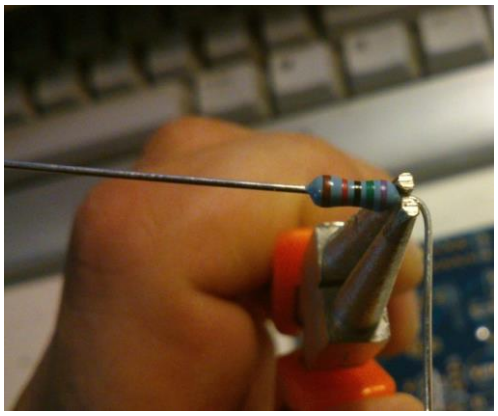
I suggest mounting standoffs before starting this phase; screws will stabilize board when it is reversed for soldering.



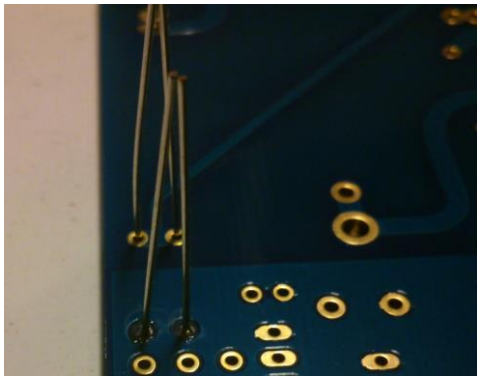
The golden rule is starting from lower profile parts so we will start from DC protection resistors.

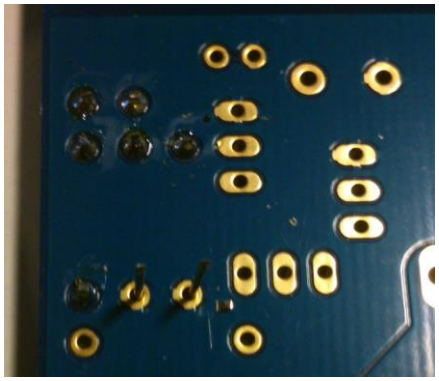
Using pliers to bend leads helps to obtain a nice looking build.

Solder parts on both boards, alternating them; it would be easier and let parts cool.



After bending leads mount the resistors in position and turn the board, and then solder first one side of both, then proceed with other side after the second board is done.

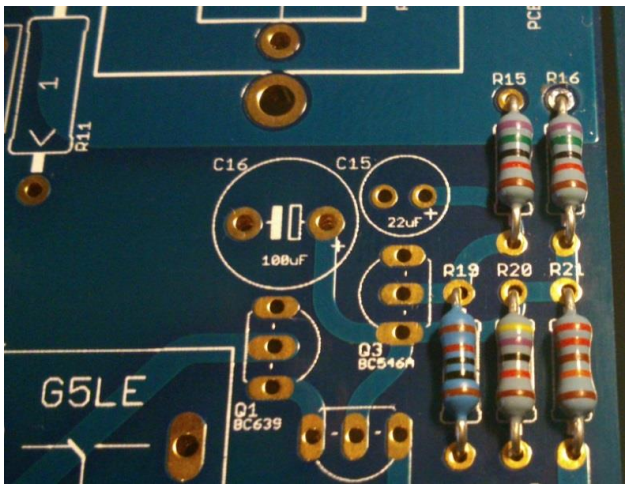




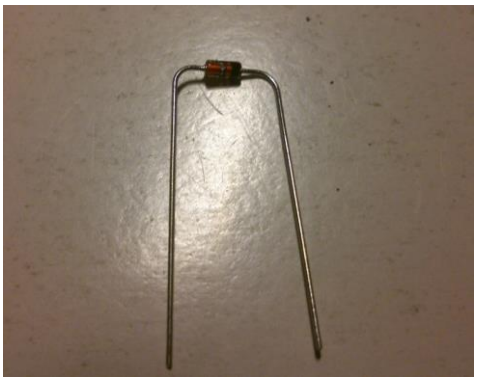
Proceed with other 1/4W resistors of the DC protection circuit.

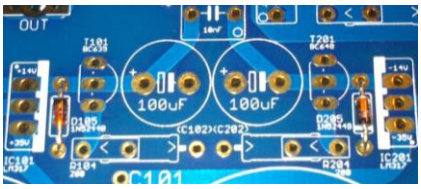
Note that I have left unsoldered thermals pads; we will do the same for all thermals so we can solder them later with a higher soldering temperature.

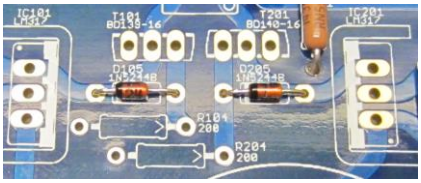
If possible keep same orientation, it will look better.



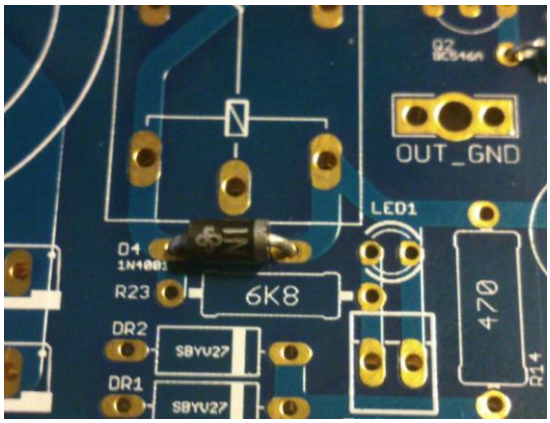
Next lower profile components are the 14V zeners, bend leads first, use same technique used for resistors (mount parts, turn the board, solder one lead for each part, do the same on the other board, solder the remaining leads).





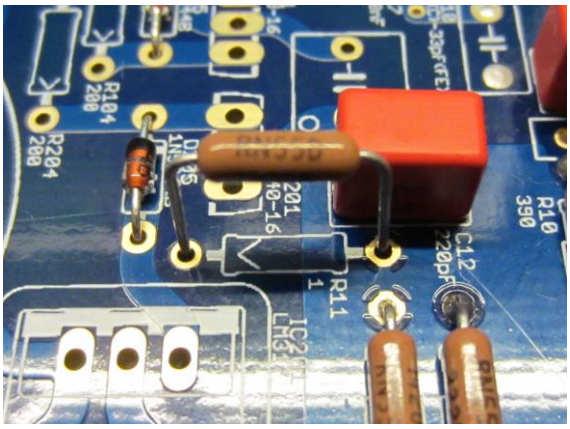


Then solder the 1N4001

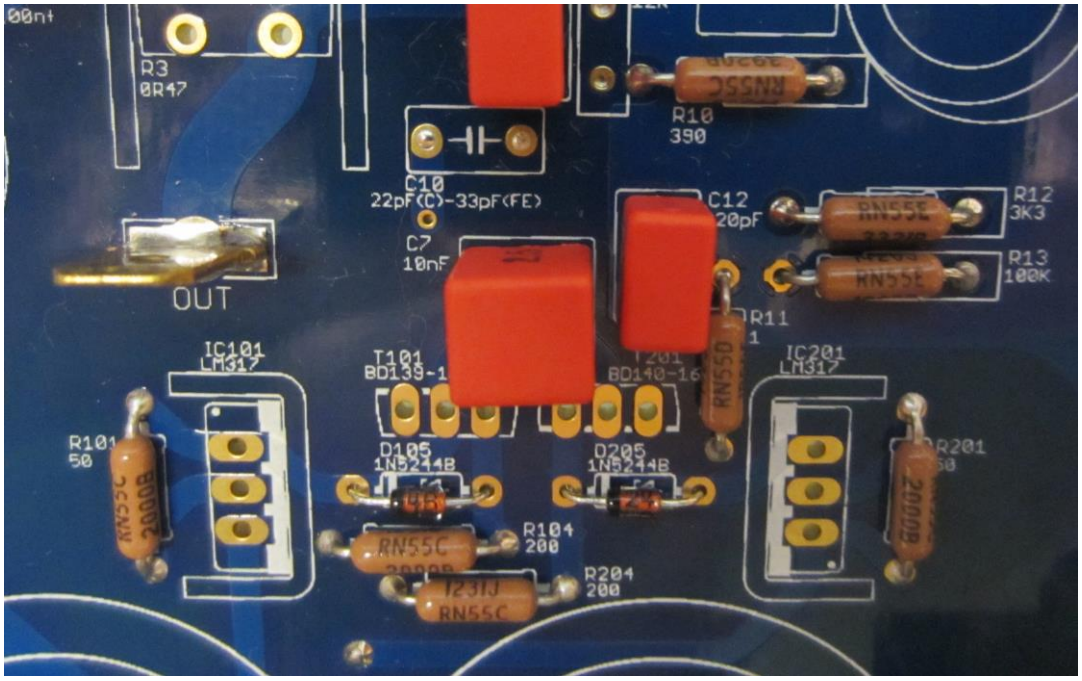


We can start with signal resistors, using same technique.

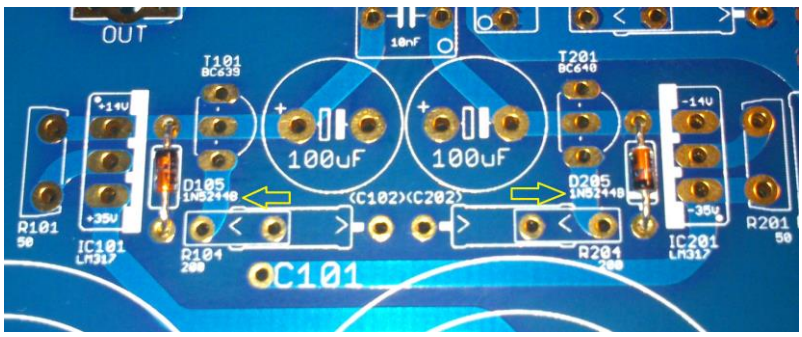
Take care to orientation, in general markings (or tolerance band) should be read in the direction of the arrow marked on board, v1.5 and v1.6 boards have a wrong direction on board for R11, use this pic as a reference:



This should be the final result (ignore other components for now):

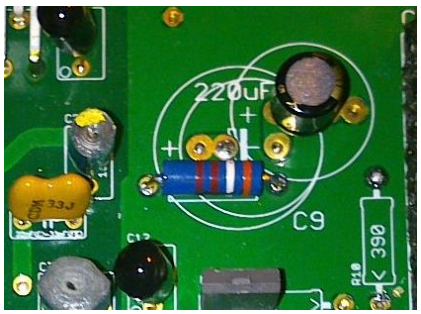


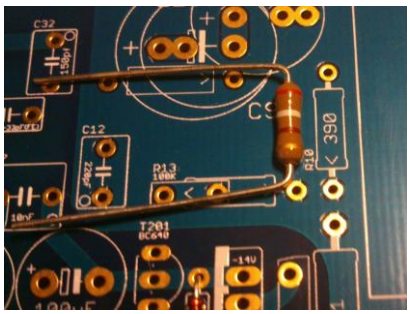
The pic is from a 1.6 board but orientation is the same for older boards (resistors will be bigger) with the exception of R104 and R204:

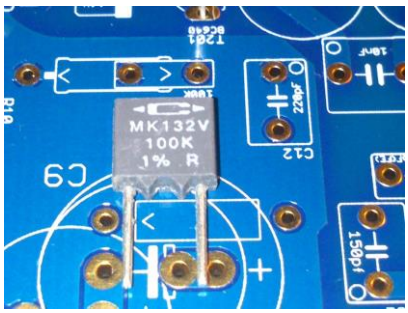


In this case follow the yellow arrows on pic.

If you have an old board with parts from the old BOM some reference pics:





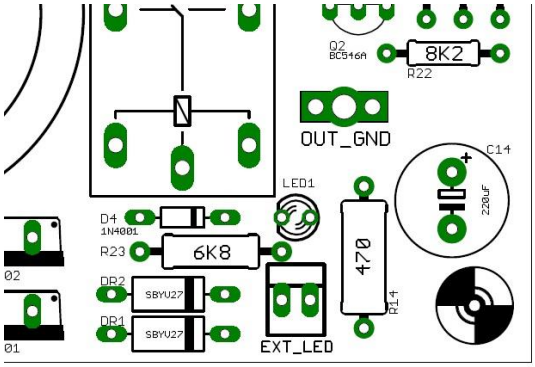


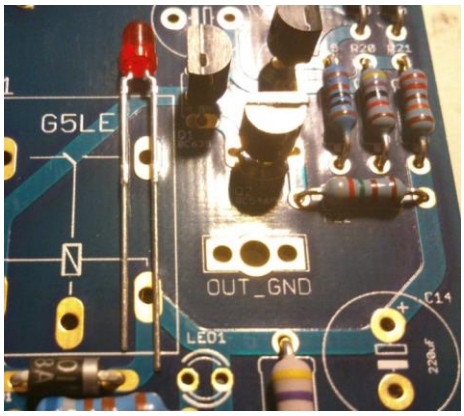
Note that a Caddock MK132 in R204 should be mounted reversed, not as in pic.

5.2.2 LED

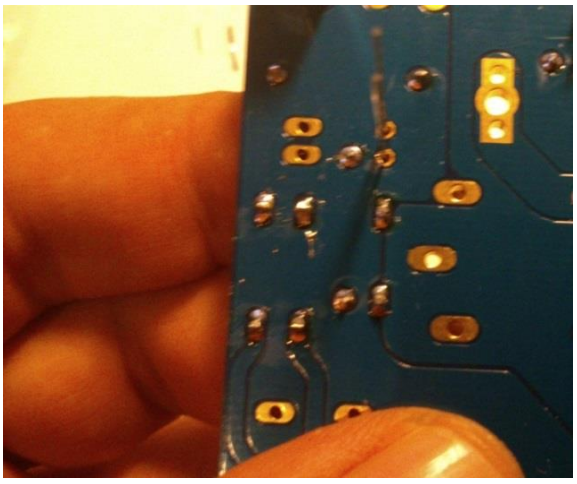
Next part to solder is the led, silkscreen it's not perfectly clear on led orientation, the silkscreen file help us.

Sadly the led in BOM doesn't have the dent so we should use leads length to identify anode and cathode, the right orientation is this one.





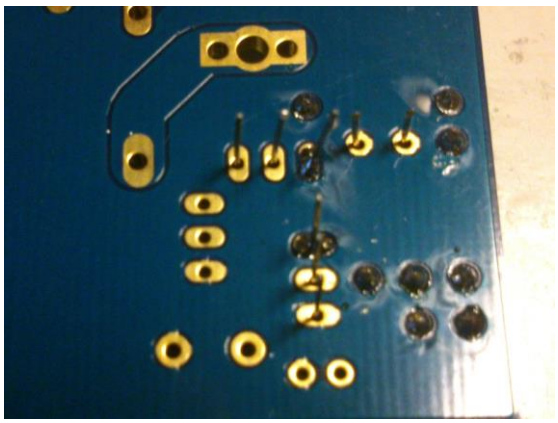
To solder it keep it in place using fingers:

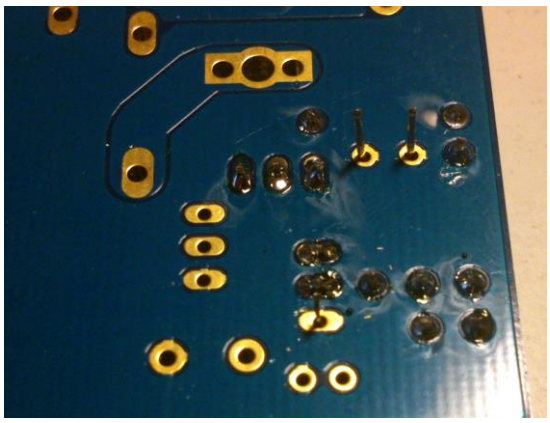


Further note on LED, traces of LED's pads and external LED connectors are parallel so for those that want to use the connector use same orientation and omit the LED on PCB, they will not work both.

5.2.3 Transistors

Start from DC protection ones, when soldering transistors solder one single lead for each

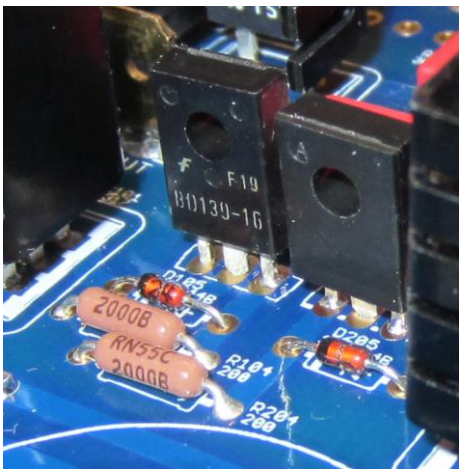




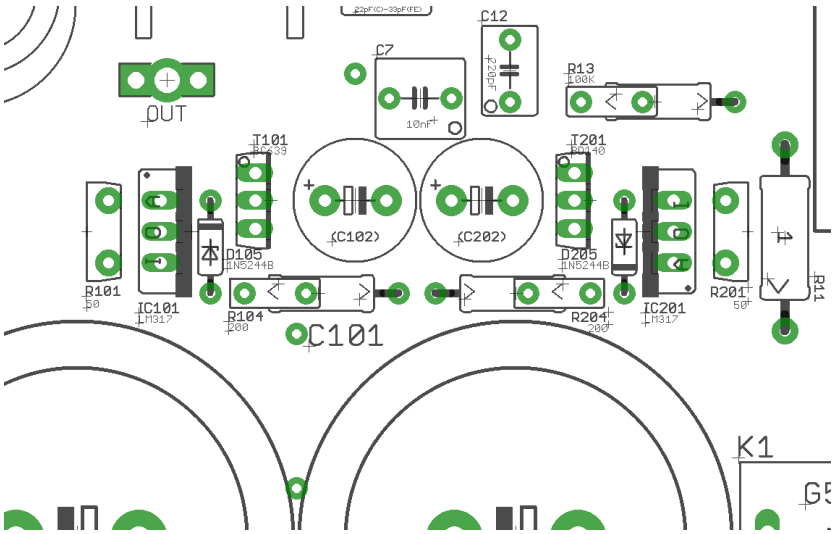
then solder other leads alternating transistors so to not heat too much parts.

As usual, leave thermal pads unsoldered to solder them later at a higher temperature.

Then we can proceed with regulator transistors using the same technique.



Older boards used BC639-16 and BC640-16 for the shunt regulator; those parts are no longer available but can be replaced with BD139-16 and BD140-16 (same part in different case), use this pic as reference:

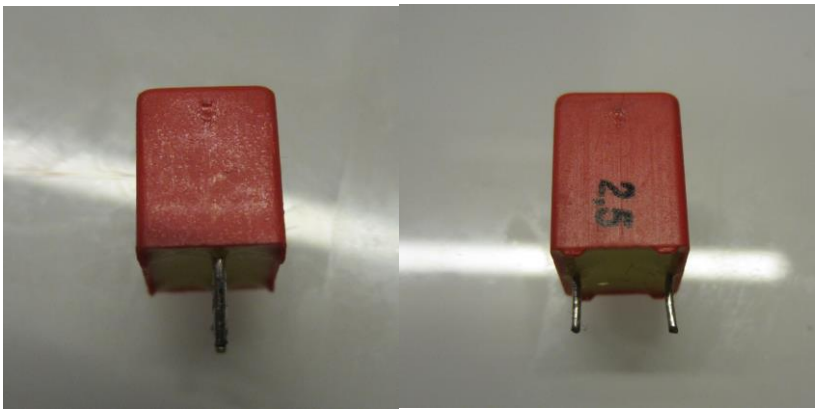


Newer boards use directly the BD139/BD140 transistors.

5.2.4 Film caps

As it could be noted, all markings on boards are oriented so that marked parts could be easily oriented.

Wima FKP2 in BOM can have right or reversed markings, they can be identified by a small sign on side of front:

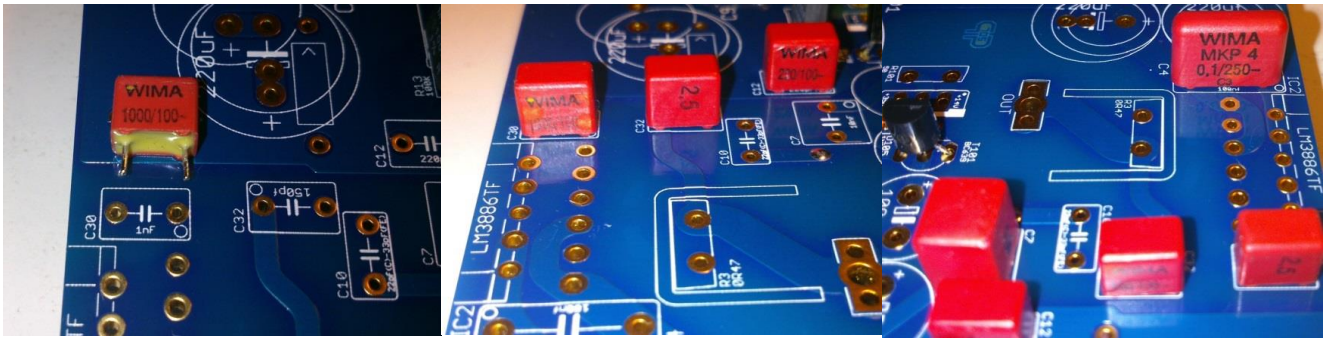


That sign should be on the left side (small) or front (big), otherwise markings are reversed.

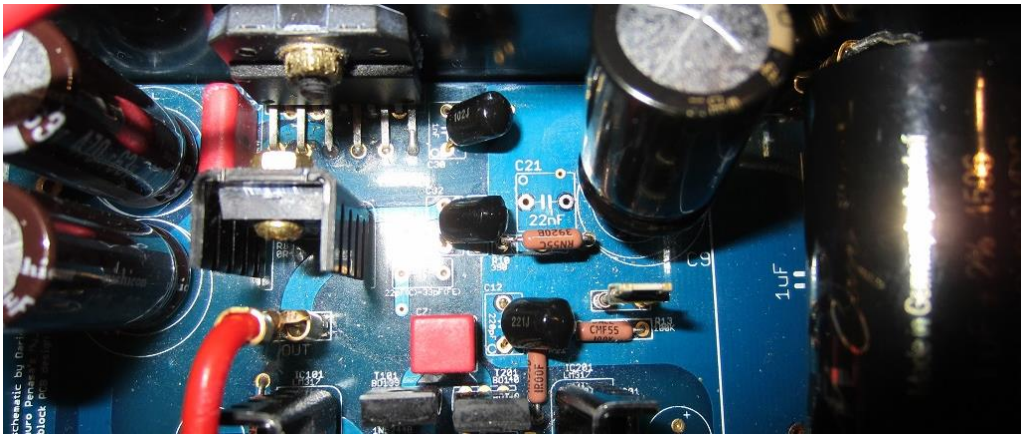
So the ones in the pics are reversed.

Amtrans AMCH are more consistent and their direction is the one of markings.

See for FKP2s (straights)

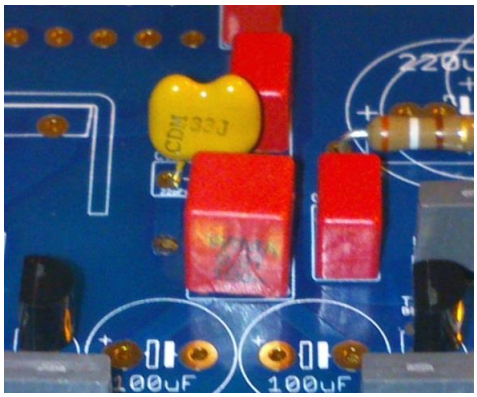


⚠ All boards up to 1.6 have an error on C32 direction, it should be reversed so referring to the first picture they all go from the left to the right. On the second you can see C32 in the WRONG direction.



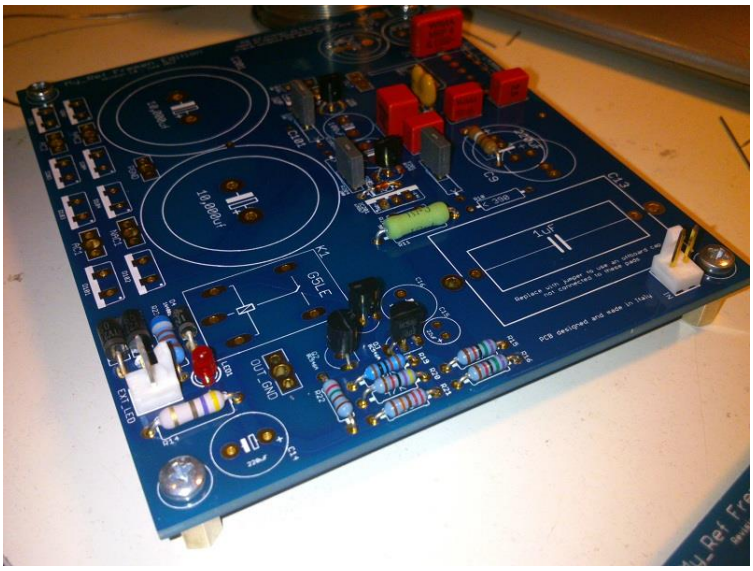
Amtrans example, note that also in this picture C32 is in WRONG direction.

Then solder C10, unless you're using My_Evo Rev. A compensations, in that case this component will be omitted.

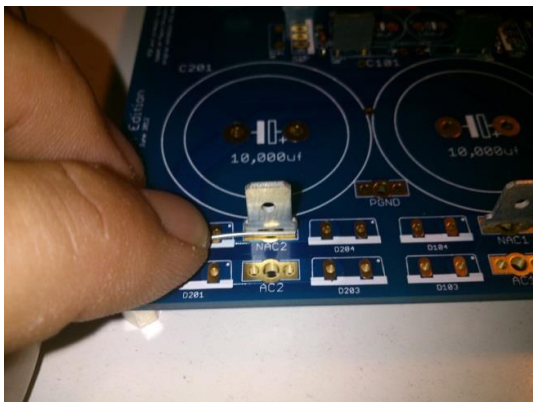


5.2.5 Connectors

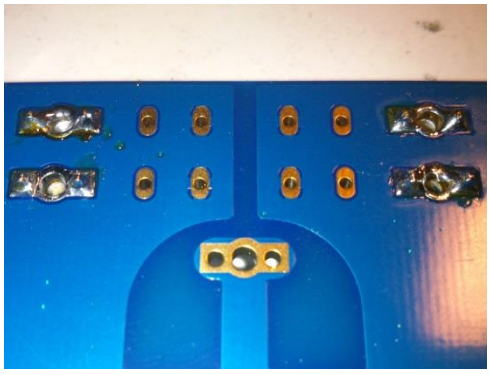
⚠There are two types of Molex included in BOM tin plated (EXT LED) and gold plated (IN)



Put in place faston tabs and solder on the upper side one end



To obtain this result tabs must be heated with the soldering iron to be soldered, then reverse board and solder all the tabs:

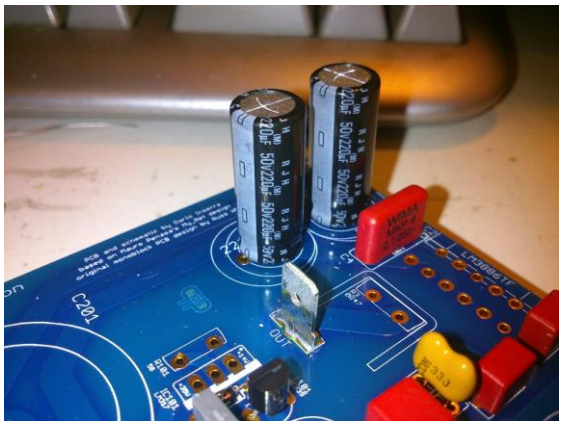


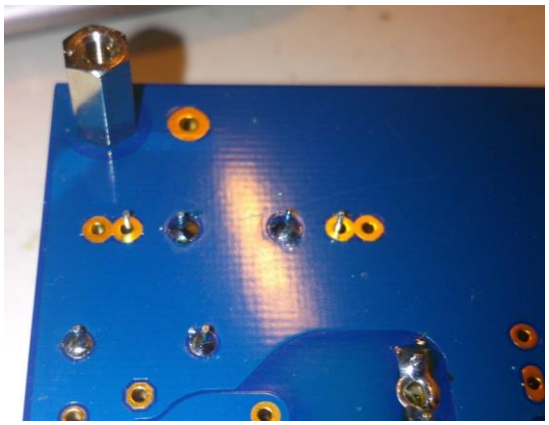
A further note on faston tabs.

If the soldering temperature it is not enough to obtain results depicted in previous post defer their installation to when we will raise soldering temperature to solder thermal pads.

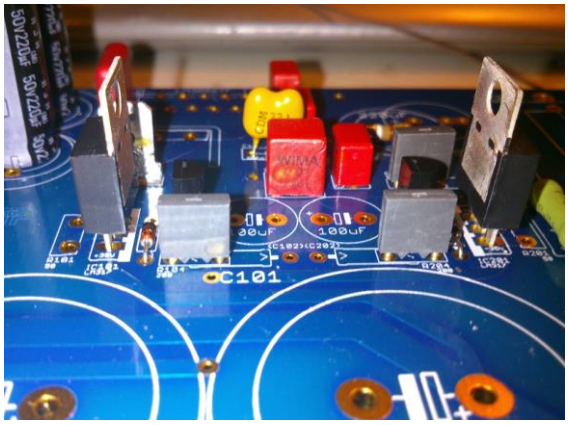
5.2.6 Higher profile parts

Now we solder C1 and C2, as usual we'll leave thermal pads for later (we'll raise soldering temperature to solder all of them):

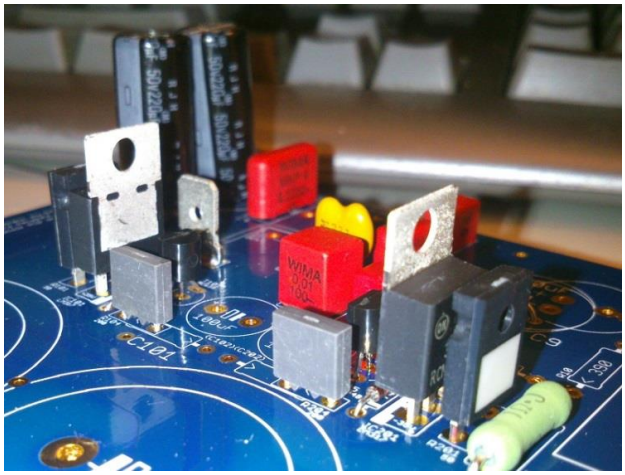




It's LM317s turn, mount heatsinks before soldering (not depicted) using the mounting kit recently included in BOM. Note that the supplied mica insulator will not fit the heatsink, it must be trimmed cutting around 1-2 mm from each side.



Now MP915s for older boards, RN55 for newer boards should have already been soldered since they're low profile parts.



Note that ceramic pads side goes on the outside for both.

Now an essential component that could be easily forgot resulting in a not working amp or worse...

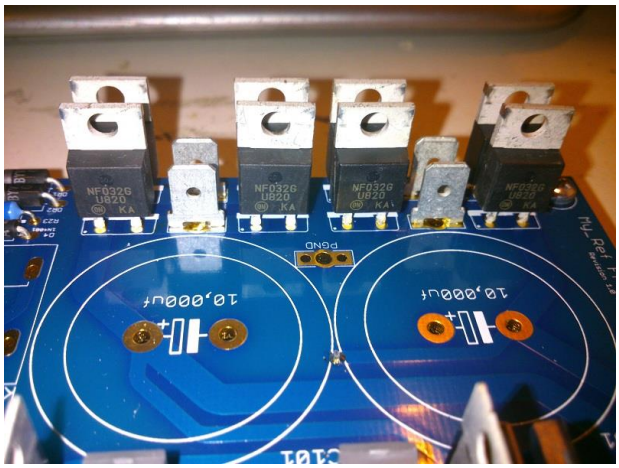
The unique jumper in the whole board (J1) which carries V-

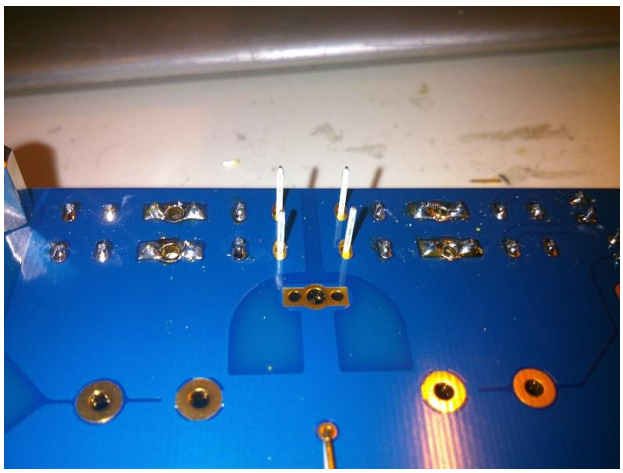
Using pliers make a 12mm jumper:

If you bought Rikens you can use their gold plated leads after trimming.

Note that J1 jumper is installed in the bottom side and soldered on the upper side.

Now it's rectifiers diodes turn

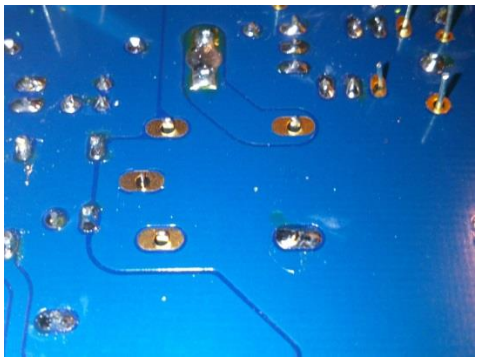


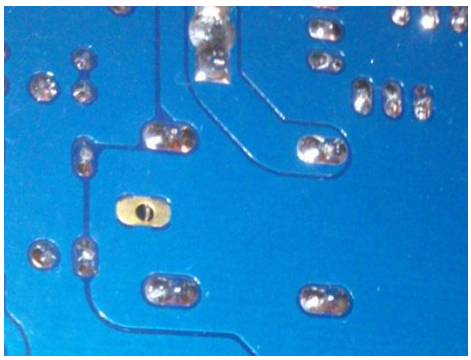


As usual thermal pads are left unsoldered.

Proceed soldering the relay, start from one or two pads while keeping it in place with fingers

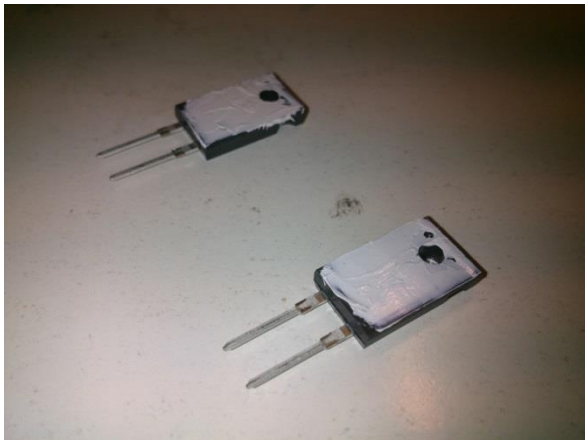
Proceed with other pads leaving out the thermal one.

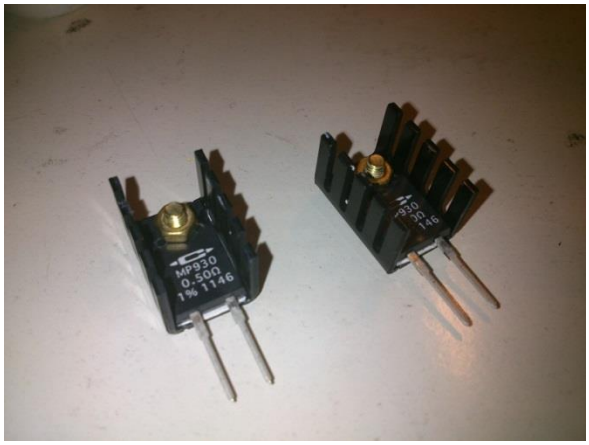




Prepare R3s with thermal compound (a thin layer).

And fix them to heat sinks using M3 screws and bolts

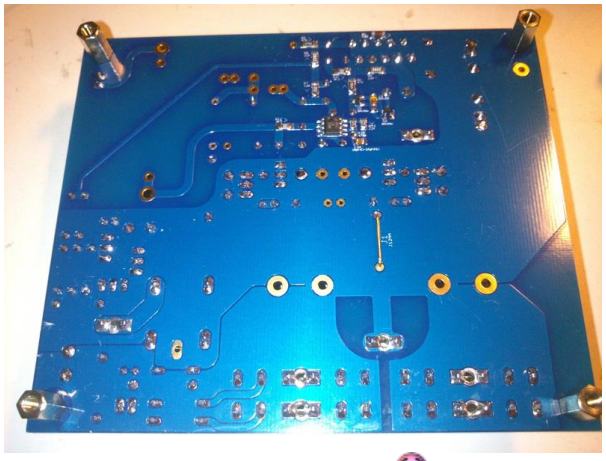




⚠M3 screws and bolts are not indicated in BOM but can be easily sourced from any hardware store. Prefer nonmagnetic materials like brass.

Solder R3 in place.

Now it is time to raise soldering iron temperature (min 350°C, 380°C for lead free), solder all thermal pads we left in previous steps and trim remaining leads.

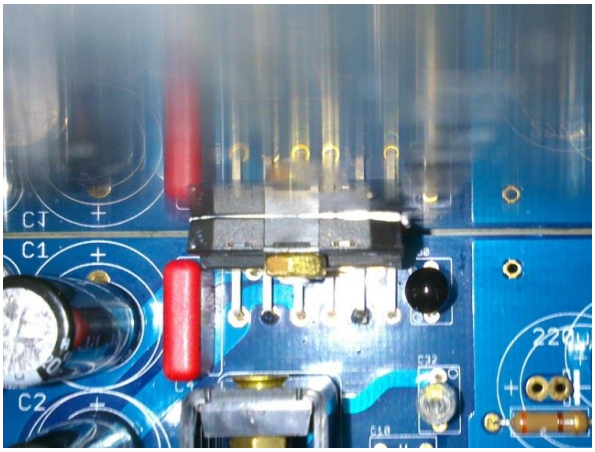


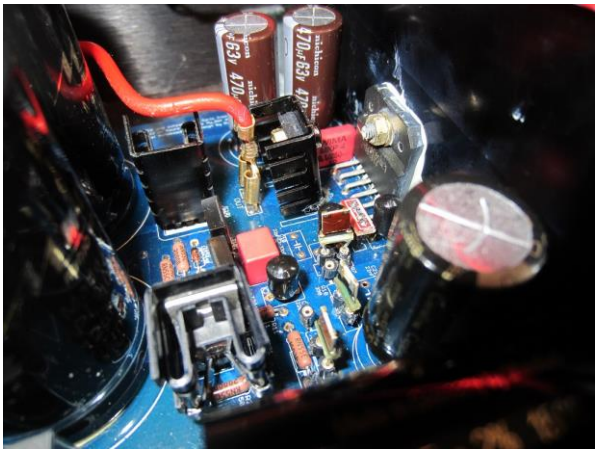
Note that in the previous picture I forgot to solder relay's pad... To solder LM3886TF take measures and drill heatsinks then fix the opamp to heatsinks.

The best way to fix the opamp to the heatsink is via an M3 grub screw and a M3 nut, there's no clearance to use a normal screw.

Then insert LM3886, together the heatsink, in place on board and solder a pair of pins on the upper side to keep chip in place.

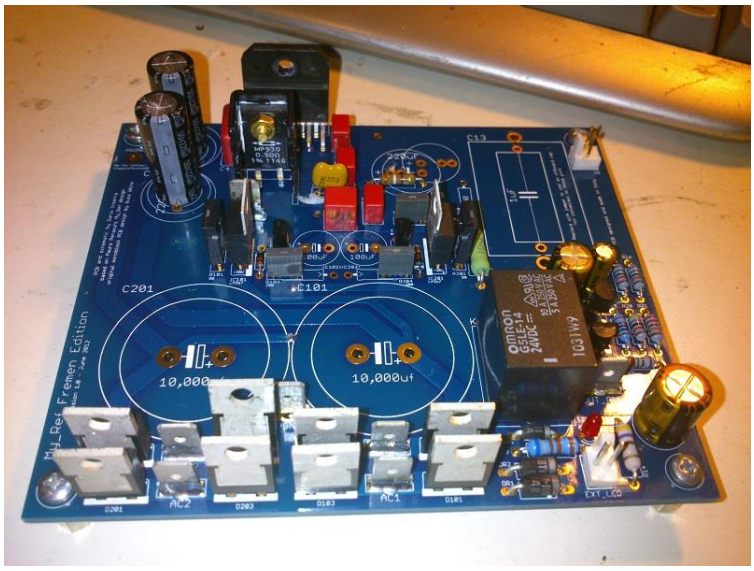
Unmount LM3886 from heatsink, reverse board and solder all pins.





Note that, unlike previously available My_Ref boards there's some clearance between heatsink and PCB...

The nearly ready boards



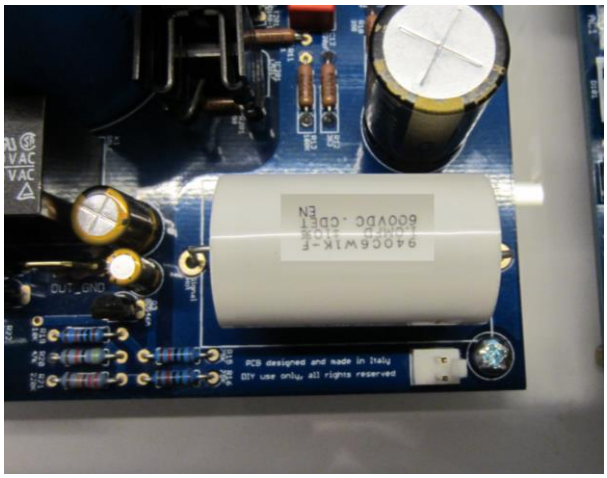
(Older boards)

(newer boards)

the only remaining components, not covered here, are the big smoothing caps.

For the input caps I suggest using something like ‘Blue Tack’ to keep them in place while soldering and add some vibration damping.

Also for those some care is needed regarding their orientation (outer foil).



5.2.7 Audiophile upgrades (best suggested parts)

The full BOM include several audiophile parts not available from Mouser.

This guide focused on building using the Mouser base BOM.

Let's start with upgrades priorities:

1. C9 (Cerafine)
2. R12 (Z-Foil)
3. C30, C32, C12 (Amtrans AMCH)
4. C13 (Mundorf Supreme)
5. R7 and R10 (Z-Foil)
6. C101, C201 (Mundorf MLythic AG)
7. R37 (Z-Foil)

Upgrades 1-4 should be considered the Audiophile Base BOM, while 5-7 for a cost no object build.

Each upgrade leads to a significant perceived performance increase and when all applied will raise the bar to another category of performance.

In this regards the Fremen Edition could be considered a 'scalable' amplifier, more upgrade components you will put in better it will sound.

1 - Cerafine in C9

I consider this upgrade MANDATORY!

KZ will sound good but it will not sound how a My_Ref Fremen Edition is supposed to.

What you can expect:

- A more life-like timbre
- A much more open, dynamic and detailed sound
- An higher harmonic content

Note:
The higher harmonic content means less loss, NOT added distortion!
Remember that I compare them to an absolute reference, in this case a shorted C9 (only for test purposes, it MUST be populated, DC offset will be 240mV!!!).

Black Gates PK and Standard are good alternatives, PK will sound very similar to Cerafine but slightly warmer and cleaner than life, Standard similar to PK but with less harmonic content and detail.

2 Vishay VAR/Texas Components TX2575/Charcroft Electronics CAR in R12

I strongly suggest this update

What you can expect:

- A more neutral and life-like timbre
- More detail and deeper soundstage
- An higher harmonic content
- Cleaner sound

3 - Amtrans AMCH in C30, C32, C12

Amtrans AMCH are a worthwhile improvement over Wima FKP2.

What you can expect:

- A more life-like timbre
- An higher harmonic content
- Cleaner sound

Note:
The higher harmonic content means less loss, NOT added distortion!
Remember that I compare them to an absolute reference, in this case an unpopulated C12 (only for test purposes, it MUST be populated).

Attention!
During revision I've determined that C32 orientation indicated on PCB (up to rev. 1.6) is not the optimal one, C32 should be mounted reversed.

4 - Mundorf MCAP Supreme in R13

Mundorf Supreme is a worthwhile improvement over CDE940C.
CDE 940 is an excellent base cap, though, and not a problem at all.

What you can expect:

- Bigger soundstage and contrast
- An higher harmonic content
- Cleaner sound
- More engaging sound

Both are quite transparent I would say 95% of the control (plain wire) for 940C and 98% for the Supreme.

Note:
The higher harmonic content means less loss, NOT added distortion!
Remember that I compare them to an absolute reference, in this case C13 populated with a piece of copper wire (only for test purposes, C13 MUST be populated to protect your loudpeakers from source/preamp failures).

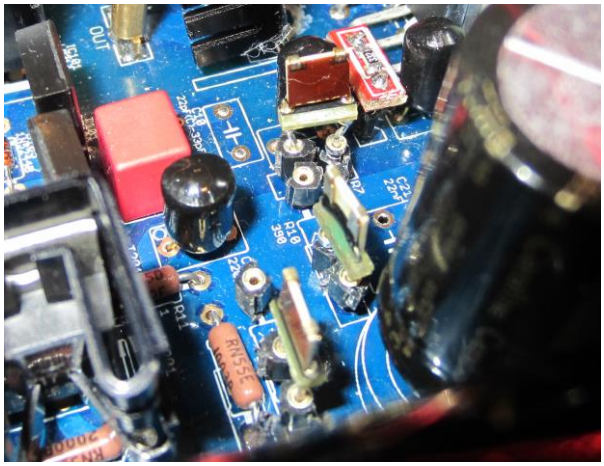
5 - Vishay VAR/Texas Components TX2575/Charcroft Electronics CAR in R7 and R10

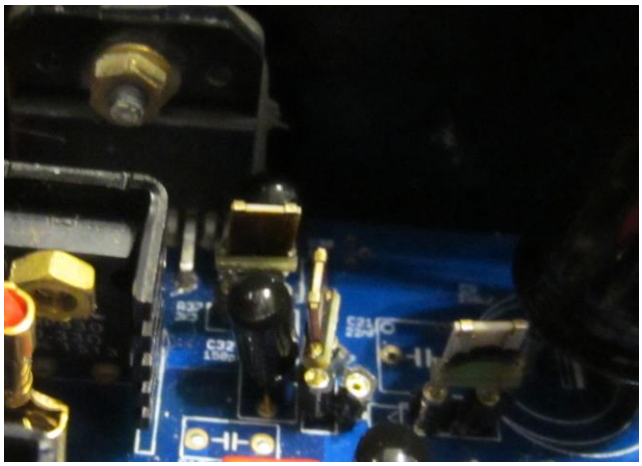
I strongly suggest to upgrade them together, if not possibile R7 is more important.

What you can expect:

- A more neutral and life-like timbre
- More detail and deeper soundstage
- An higher harmonic content
- Cleaner sound

Z-Foils advantage increases as more of them are used, see orientation here.





6 - Mundorf MLythic AG in C101, C201

Mundorf AGs are a worthwhile improvement over Vishay/BC PMG-SIs.

Vishay/BC PMG-SI is an excellent base cap, though, and not a problem at all.

What you can expect:

- Blacker background and more contrast/focus
- An higher harmonic content
- Cleaner sound
- Fuller and deeper bass

The change after the upgrade is not small, pretty evident but AGs comes at over 4x the PMG-SI cost.

Is it worth it?

IMHO, absolutely yes

7 - Vishay VAR/Texas Components TX2575/Charcroft Electronics CAR in R37

What you can expect:

- A more neutral and life-like timbre
- More detail and deeper soundstage
- An higher harmonic content
- Cleaner sound

Z-Foils advantage increases as more of them are used.

Try to keep leads as short as possible, you will hear it.

6 TRANSFORMERS WIRING

As specified each module should need a dedicated transformer to avoid hum problems but several builders used a single one without problems.

Transformers must have double secondaries (2x25V, 2x24 also fine) and 160VA minimum rating (120VA minimum for the more efficient R-Cores).

In case of single transformer, the minimum is 300VA.

You must find the label on transformer or its datasheet to wire them.

Selectronic R-Cores

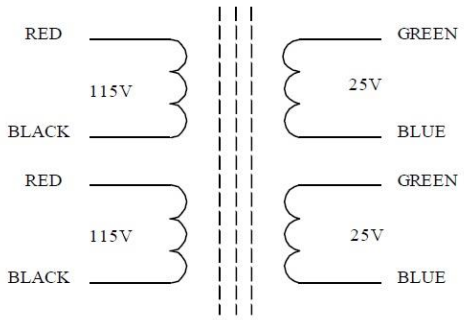
With this type of label the first color in the couple is usually hot/phase, the second neutral.

So:

AC1 Violet
NAC1 Gray
AC2 Blue
NAC2 Red

Antek AN-2225

From manufacturer datasheet:



With this type of label/wiring you can't distinguish visually each secondary from the other.

One way to do it is using a multimeter to measure continuity.

Members of the same couple will have continuity.

After you identified couples wire in this way:

AC1 Green1
NAC1 Blue 1
AC2 Green 2
NAC2 Blue 2

Others

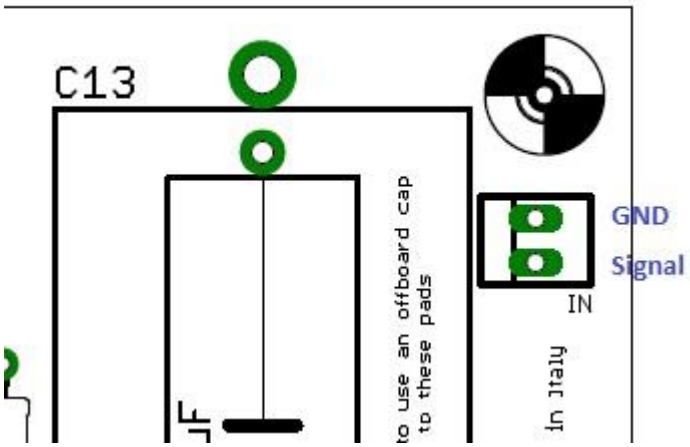
Sometimes you have no clue from label which is wire in a couple is phase/hot and which is neutral (usually in that case both wires in a couple are in a single color).

Do not worry, simply ignore the hot/neutral thing, it will work perfectly.

It is possible, though, that reversing wires in a couple sound better.

7 INPUT CONNECTOR PINOUT

Input connector pinout on more recent boards is indicated, for older ones:



8 CONNECTORS SELECTION

The My_Ref Fremen Edition is a high performance amp and using quality connectors will reward you with increased perceived performance.

Magnetic connectors are a no-no, use at least high copper content brass or even better copper ones.

Good quality audiophile brands are CMC USA, Vampire, WBT, Furutech.

I will suggest some specific make/models, they will make and audible difference.

In fact, connectors will make a bigger difference than cables.

8.1 Power inlet, fuse and fuse holder

Prefer the ones with integrated fuse holder, they will be easier and safer to use, if possible with gold plated connectors.

Furutech make an audiophile version (FI-03G) which is very well made (20€):

If you prefer to use a separated fuse holder you can use INLET G, also from Furutech (13€):

The power fuse will be slow-blow 1.25A (230V) or 2.5A (110V), the FI-03G mounts a 5x20mm fuse.

I've tried Padis audiophile fuses and found them worse than industrial ceramic ones.

8.2 Input RCA Connector

Don’t use nothing less than premium industrial parts like Neutrik/Rean/Switchcraft or similar.

Audiophile parts from PartsConnexion/HiFiCollective are usually better.

A good price/performance ratio connector is CMC-805-2.5-F from CMC USA (14€/pair):

Or its full copper counterpart CMC-805-CU-G (25€/pair).

Low mass copper connectors are the absolute best, like the NextGen WBT-0210 Cu from WBT (50-60€/pair).

8.3 Binding posts

Best price/performance ratio are CMC-858-S-G from CMC USA (27€/4 pieces)

Or their full copper counterparts CMC-858-S-CU-G from CMC USA (50-60€/4 pieces)

Low mass copper connectors are the absolute best like the NextGen WBT-0703 Cu from WBT (80-90€/4 pieces).

9 LAYOUT

Several layout and different type of case can be used but, basically, there are two main possibilities:

- Full mono-block construction
- Stereo construction

In the first case you will use one case for each channel while on the second you will put both channels in a single chassis.

As a mono-block excellent example the Davym's build:

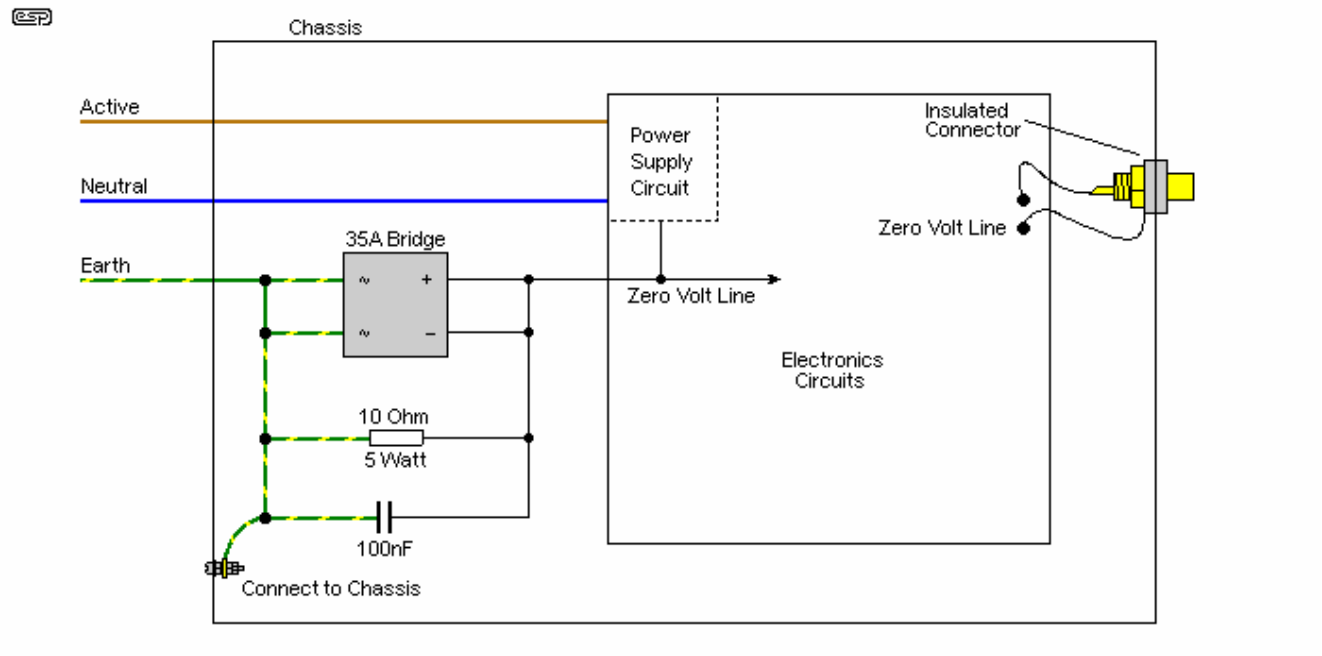
As a stereo example, my build layout:

10 GROUNDING AND SAFETY CONSIDERATIONS

The My_Ref and so the My_Ref Fremen Edition are floating designs so they're not supposed to be grounded.

While best perceived performance is achieved without any form of grounding the resulting system would be not completely safe since in case of a catastrophic failure you could have mains voltages on connectors.

While such a fail is unlikely for a safe build which retains most if not all of its perceived performance using a safety ground loop breaker connected to PGND, like in this suggested grounding layout:



(image from [Elliot Sound Products website])

10.1 Heatsink grounding

The heatsink, if not grounded, is a possible source of hum.

You will not likely hear that hum unless you have very sensitive speakers, like 94 dB and up or you put your ears right on speaker cones.

Most builds will have the heatsink already grounded since it will be part of a grounded metallic chassis.

In case the heatsink is not already grounded, like when using a plastic or wood chassis or when the heatsink is insulated from the grounded chassis the HSGND pad could be used to ground the heatsink.

The HSGND pad **must not be used** if the heatsink is already grounded by other means.

11 FIRST POWER UP

After branching transformers, you can fire up modules (be sure they're heatsinked....) with all inputs and outputs disconnected.

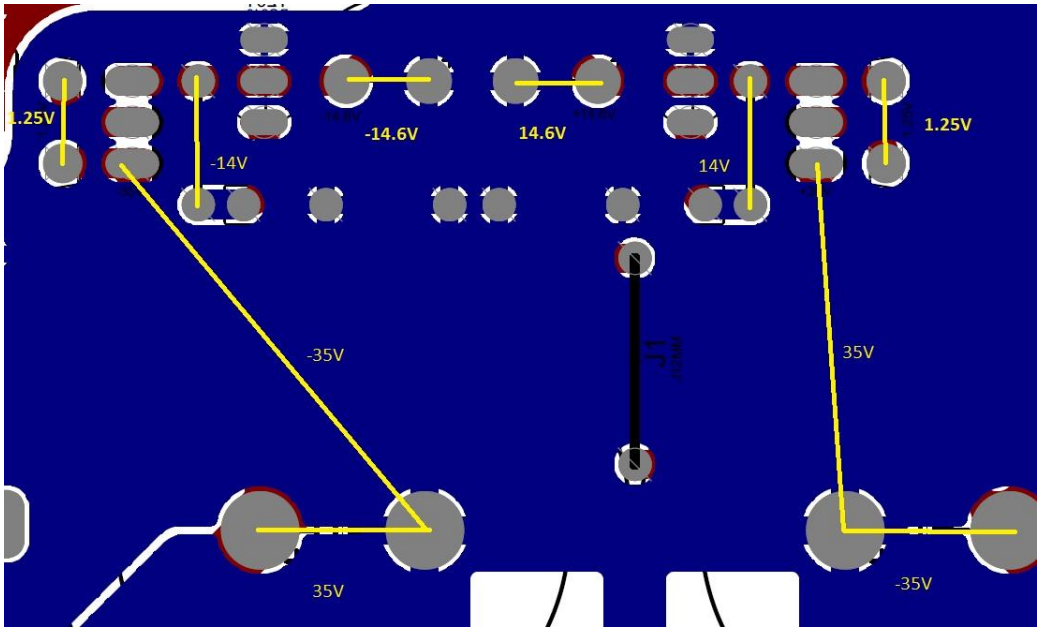
You should hear relays click, this is the first good news, it means modules are working and no serious DC is present at output.

Keeping modules powered on we then proceed to measure DC offset at output tabs.

It should measure less than 20mV and with shorted inputs less than 10mV.

If relays do not click do not panic, power off modules and ask for help on the build thread.

In such case, it would be useful if you measure first some voltages (on newer boards they’re already indicated on PCB):



Please also post macro pics of your boards (up and bottom sides).

Disclaimer

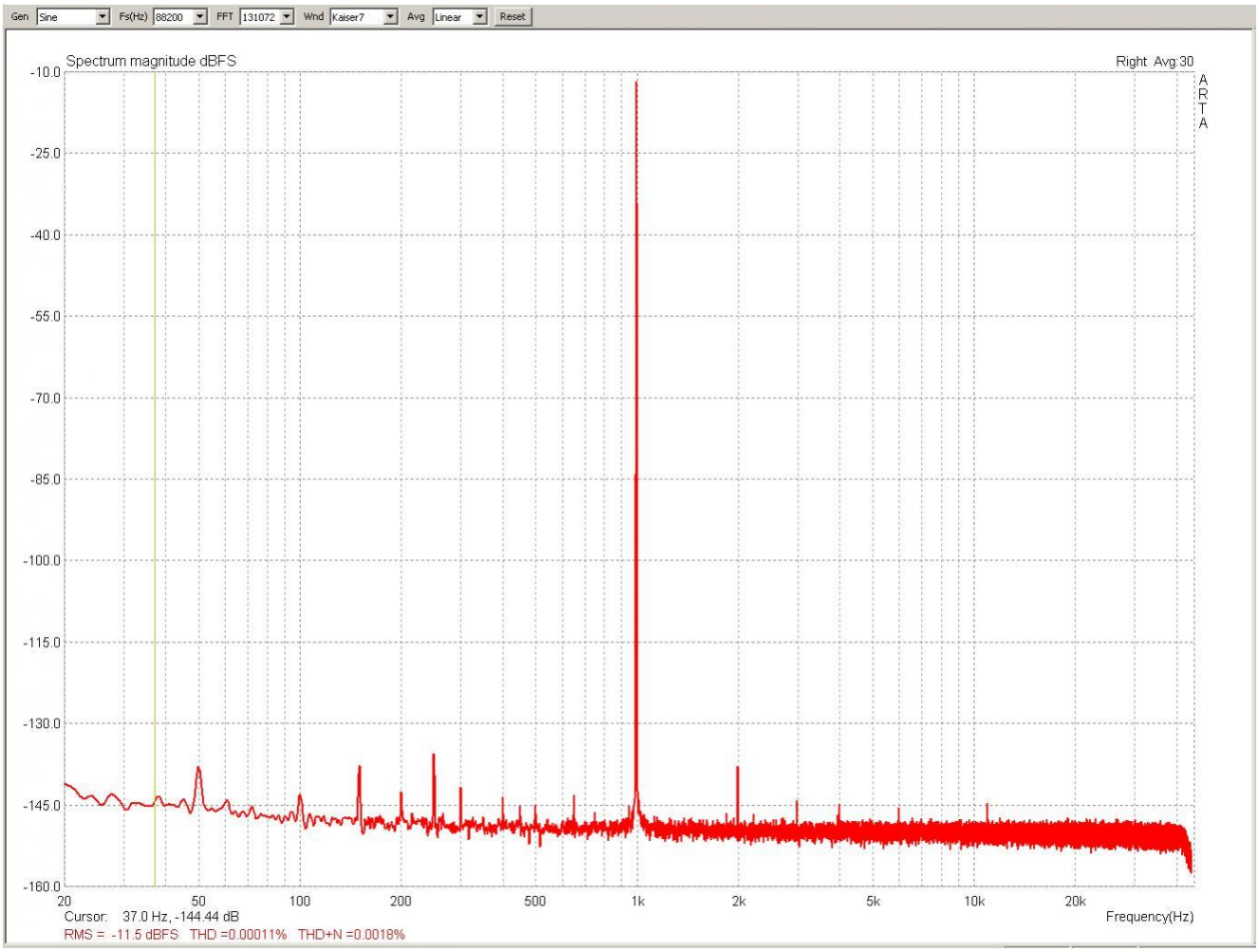
This tutorial it's not the unique way to build the amp and it's not necessarily the best one but it worked for me several times.

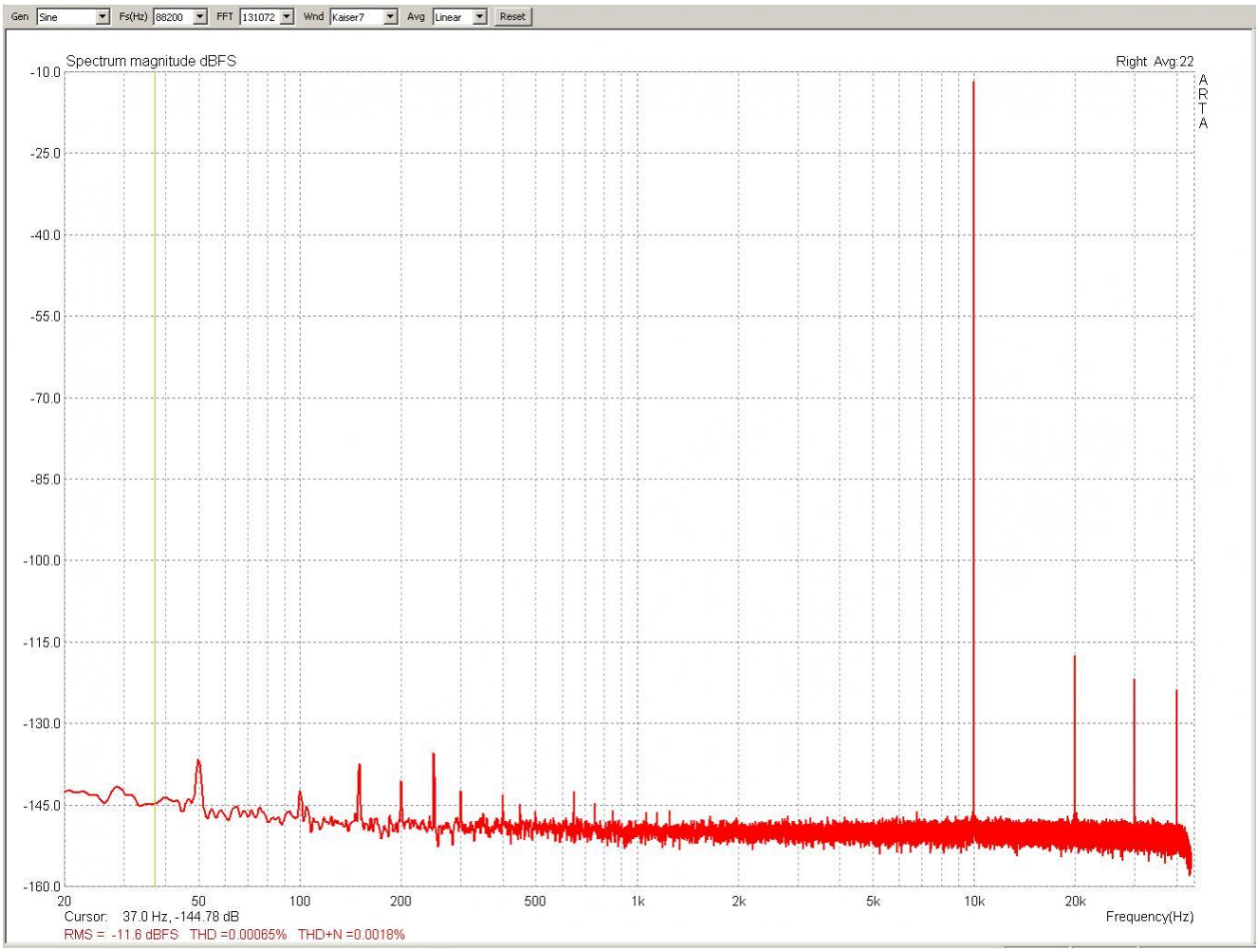
12 MEASUREMENTS

JosephK made a full measurement set, I will simply include here what he posted on the build thread.

I have made tests on the board, and the results are very nice and clean. This version of the FE board measures very close to what I had seen testing my original Penasa Myref-Evolution units.

In the pics there is the output at 5W, at 1kHz and 10kHz.

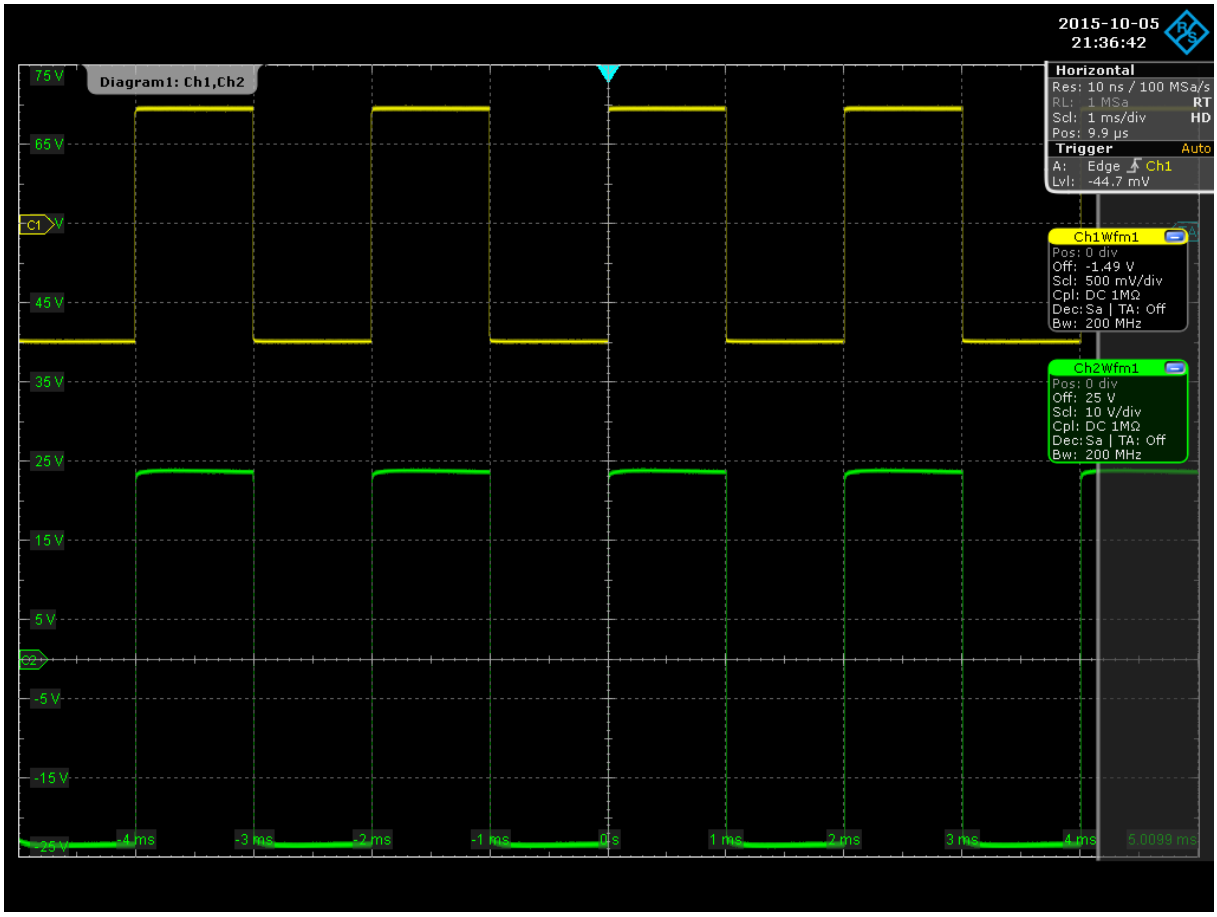


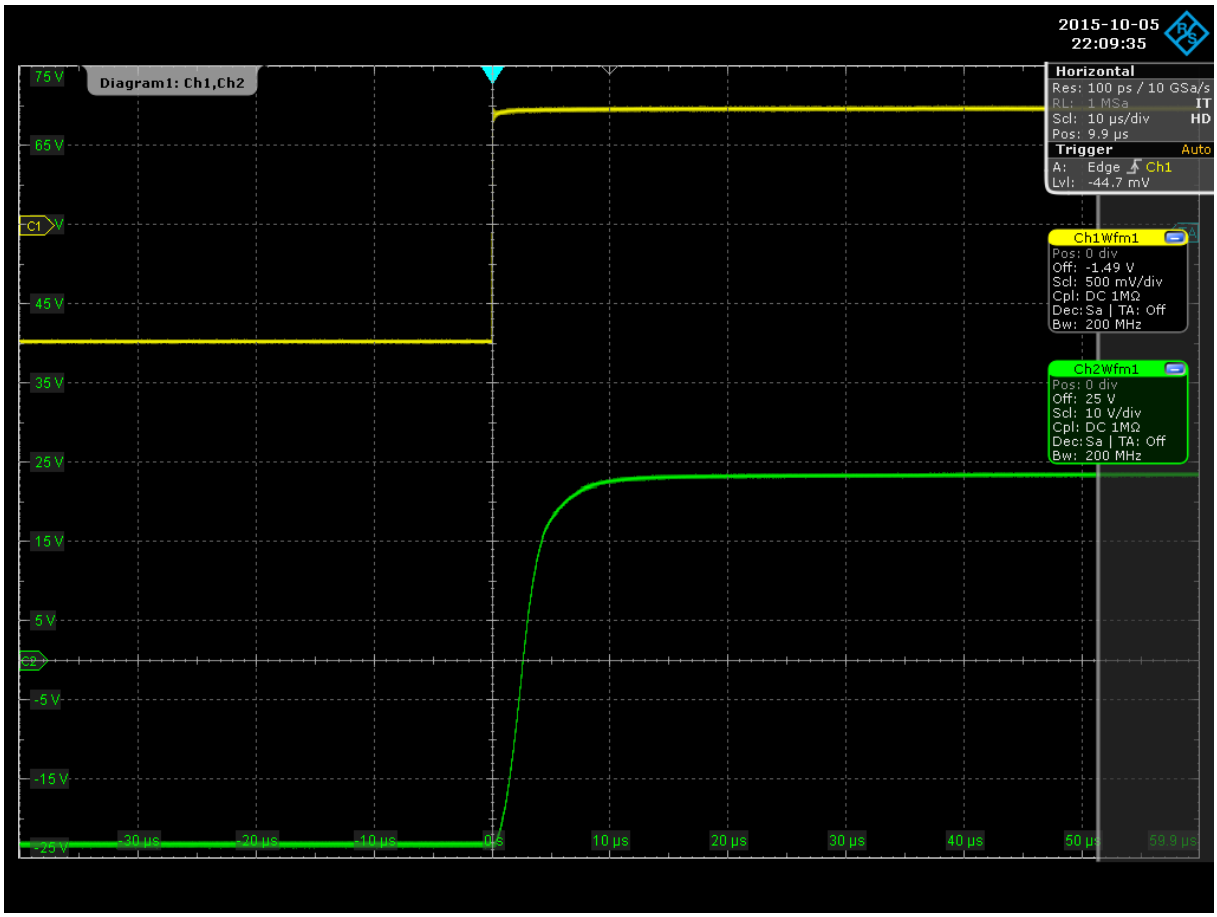


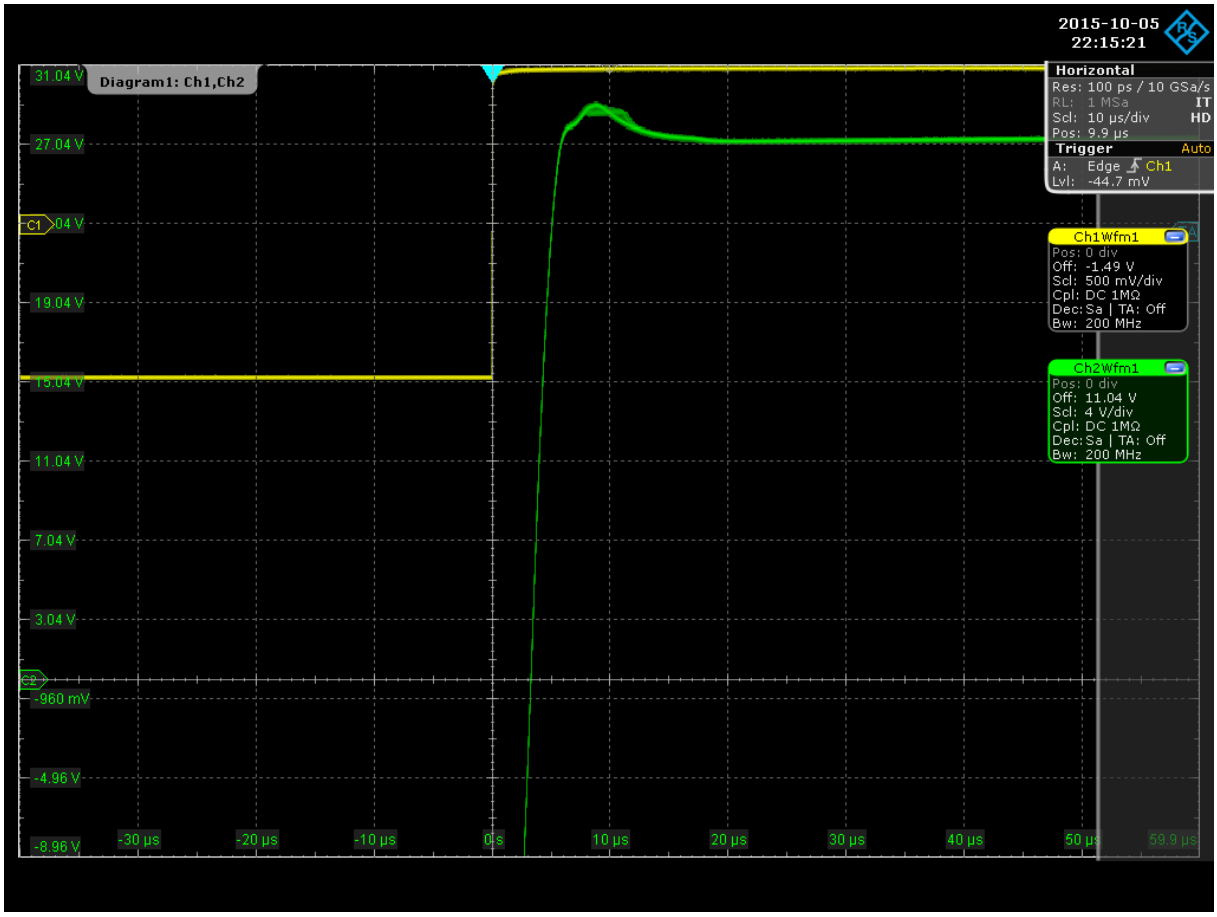
Additionally, here are some time domain tests, showing square wave response on 6.8ohm load, -yellow trace always the generator, -green is output

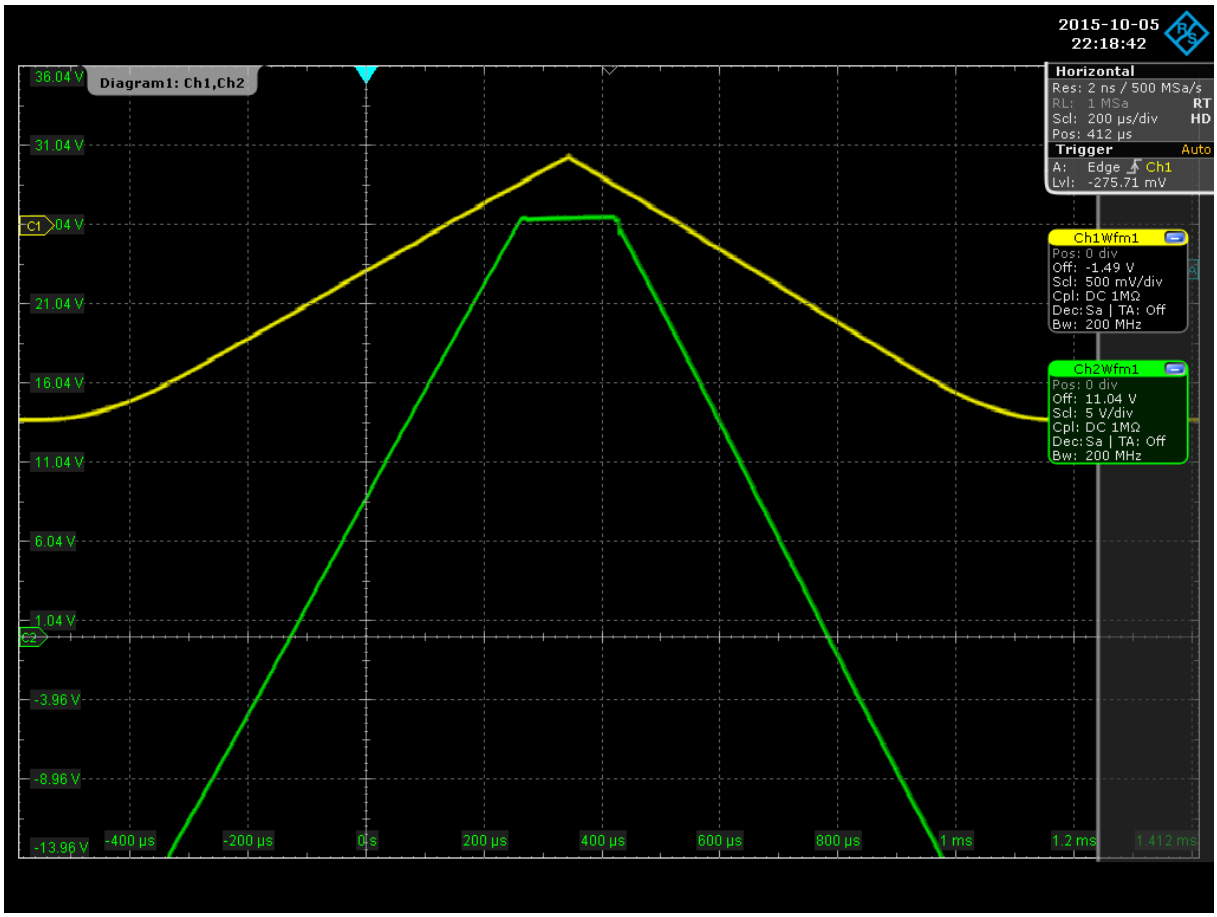
-1. 1kHz square at close but below to 50Vp-p output
-2. close-in zoom of the same
-3. clipping at 27V peak, 54Vp-p
-4. showing the behaviour when getting out of clipping, almost zero sticking, as visible

To note that during these tests the dissipated power was such that at a certain point my poor test resistor had just dis-soldered itself with a soft plop sound..
(I used to just hang it in the air below the setup)


2015-10-05
21:36:42
Diagram1: Ch1,Ch2
Horizontal
Res: 10 ns / 100 MSa/s
RL: 1 MSa
RT
Scl: 1 ms/div
HD
Pos: 9.9 µs
Trigger
Auto
A: Edge Ch1
Lvl: -44.7 mV
Ch1Wfm1
Pos: 0 div
Off: -1.49 V
Scl: 500 mV/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
Ch2Wfm1
Pos: 0 div
Off: 25 V
Scl: 10 V/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
75 V
65 V
C1
45 V
35 V
25 V
15 V
5 V
C2
-5 V
-15 V
-25 V
-4 ms
-3 ms
-2 ms
-1 ms
0 s
1 ms
2 ms
3 ms
4 ms
5.0099 ms


2015-10-05
22:09:35
Diagram1: Ch1,Ch2
Horizontal
Res: 100 ps / 10 GSa/s
RL: 1 MSa
IT
Scl: 10 µs/div
HD
Pos: 9.9 µs
Trigger
Auto
A: Edge Ch1
Lvl: -44.7 mV
Ch1Wfm1
Pos: 0 div
Off: -1.49 V
Scl: 500 mV/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
Ch2Wfm1
Pos: 0 div
Off: 25 V
Scl: 10 V/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
75 V
65 V
C1
45 V
35 V
25 V
15 V
5 V
C2
-5 V
-15 V
-25 V
-30 µs
-20 µs
-10 µs
0 s
10 µs
20 µs
30 µs
40 µs
50 µs
59.9 µs

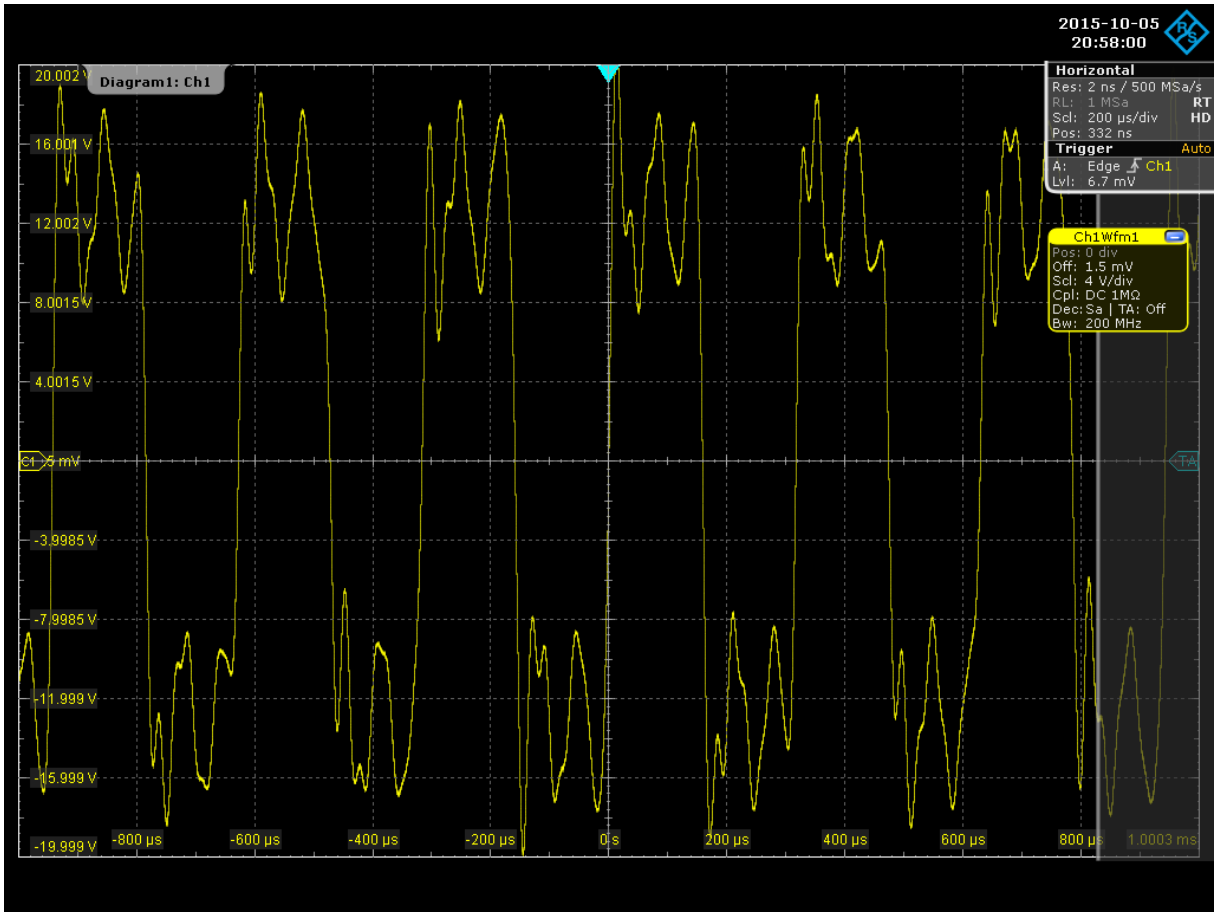

2015-10-05
22:15:21
Diagram1: Ch1,Ch2
31.04 V
27.04 V
C1
04 V
19.04 V
15.04 V
11.04 V
7.04 V
3.04 V
C2
-960 mV
-4.96 V
-8.96 V
-30 µs
-20 µs
-10 µs
0 s
10 µs
20 µs
30 µs
40 µs
50 µs
59.9 µs
Horizontal
Res: 100 ps / 10 GSa/s
RL: 1 MSa
IT
Scl: 10 µs/div
HD
Pos: 9.9 µs
Trigger
Auto
A: Edge Ch1
Lvl: -44.7 mV
Ch1Wfm1
Pos: 0 div
Off: -1.49 V
Scl: 500 mV/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
Ch2Wfm1
Pos: 0 div
Off: 11.04 V
Scl: 4 V/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz

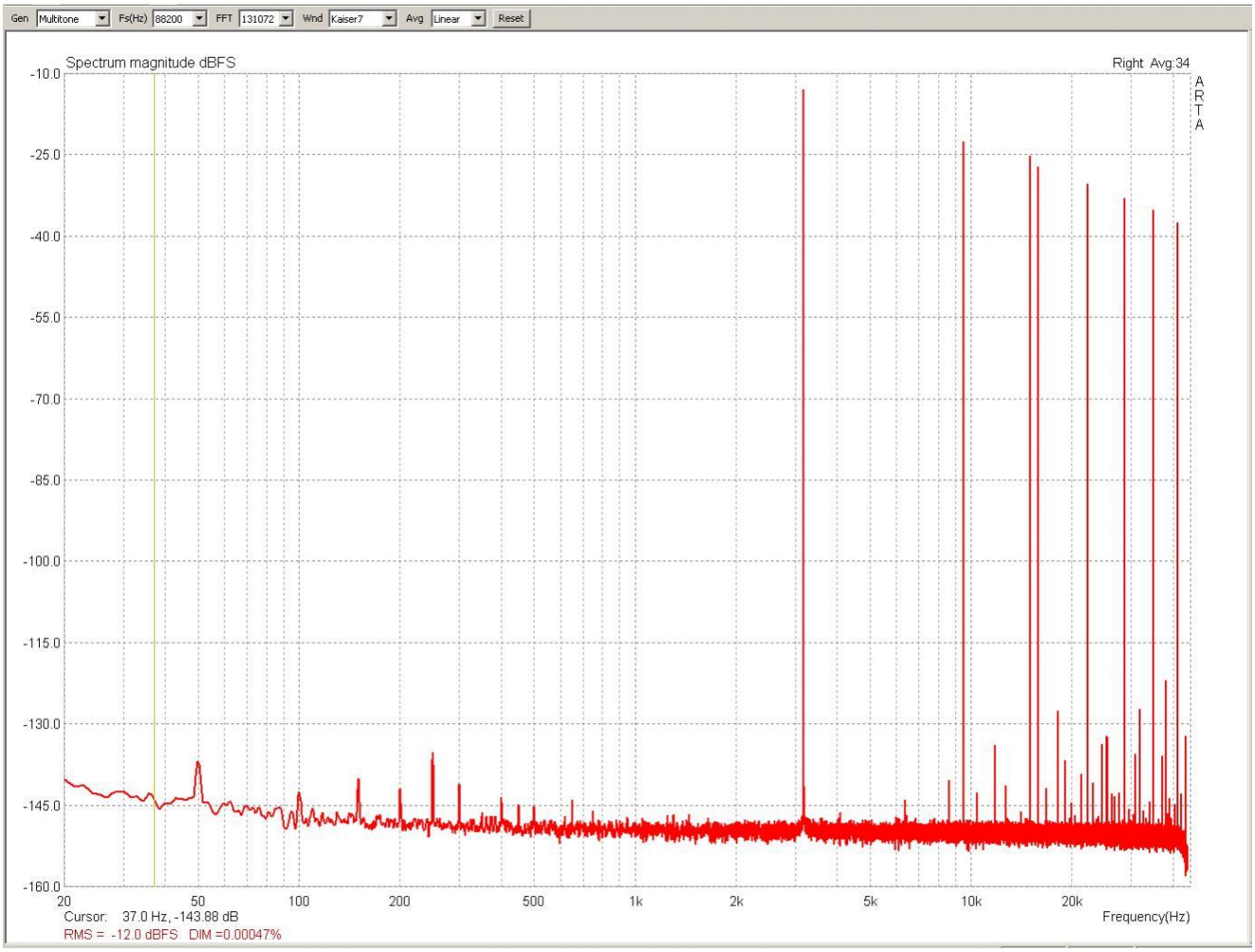

2015-10-05
22:18:42
Diagram1: Ch1,Ch2
36.04 V
31.04 V
C1
04 V
21.04 V
16.04 V
11.04 V
6.04 V
1.04 V
C2
-3.96 V
-8.96 V
-13.96 V
-400 µs
-200 µs
0 s
200 µs
400 µs
600 µs
800 µs
1 ms
1.2 ms
1.412 ms
Horizontal
Res: 2 ns / 500 MSa/s
RL: 1 MSa
RT
Scl: 200 µs/div
HD
Pos: 412 µs
Trigger
Auto
A: Edge Ch1
Lvl: -275.71 mV
Ch1Wfm1
Pos: 0 div
Off: -1.49 V
Scl: 500 mV/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz
Ch2Wfm1
Pos: 0 div
Off: 11.04 V
Scl: 5 V/div
Cpl: DC 1MΩ
Dec: Sa | TA: Off
Bw: 200 MHz

For fun, this is the Dynamic InterModulation test waveform, the one which the amplifier had managed to follow (at 5W output power) with

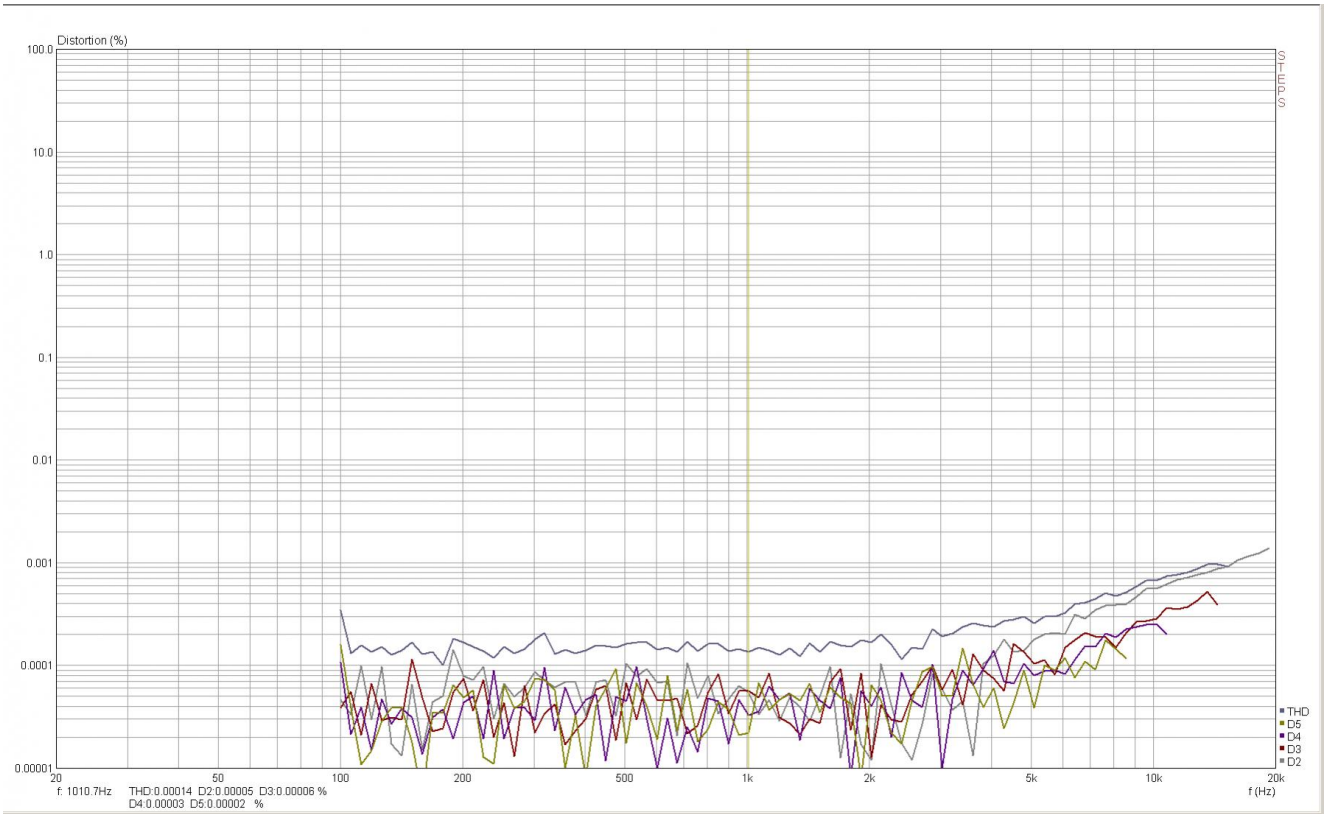
DIM=0.00047% fidelity

(3,18kHz square + 15kHz pure sine, 4:1 ratio)





The distortion vs. frequency sweep.
Measurement setup / sound card bottom line is flat around THD=0.00015%, ~up to 20kHz.
As visible, the amplifier distortion at low frequencies is dissolving in the measurement background, but at 10kHz it is very cearly standing out.



JosephK posted also a complete PDF with all tests made, [you can find it here](#).

13 SOME ADDITITIONAL NOTES FOR BUILDERS

For those that bought the ERO kit

Solder the KP1834s as the last parts, after cleaning the boards.

Polystyrene is very delicate and I cannot find a datasheet to confirm or not they can stand with solvent washing.

Soldering temperature for C101/C201

The thermal pads of both caps need a very high temperature to be well soldered, I would say around 370-380 °C.

Final inspection before powering up

Before powering up inspect all solder joints surrounded by the ground plane to insure joints are exactly on pads without making bridges with the ground plane.

It should not be a problem thanks to the solder mask but it is a cleaner and safer job.

14 GREETINGS

Mauro Penasa for his great design and kind permission.

LinuxGuru for his help on new compensation.

Luka for the LM318 PS initial design.

KSTR for the new C9, R10 arrangement.

Soongsc, Marce, Sebaastian, KSTR and Metal for help on PCB design.

BMCBob for all support, tests and reviews.

JosephK for his measurements and suggestions during v1.5 redesign