DDDAC1794MK3 DAC Board

Index & links:

A.	**Introduction**	**Page 1**
B.	**Features and Specification**	**Page 1**
C.	**Board Layout and Dimensions**	**Page 2**
D.	**Completing the PCB with the delivered through hole parts**	**Page 3**
E.	**Connecting the PCB to the outside world**	**Page 9**
F.	**Bias adjustment and testing the DAC**	**Page 11**
G.	**Possible options and tweaks to improve sound quality**	**Page 13**
H.	**Disclaimer Brand and Copyrights DDDAC**	**Page 14**

A. Introduction

This manual gives background, specifications, recommendations and act as a construction guide for assembling the PCB board and installing the DAC.

Why a new revision? The previous DAC model, the DDDAC1794PBT, was 10 years old. During lifetime, I received a lot of feedback from the DIY community with tweaks, ideas for use or implementation of other parts etc. Also, during that time, I gained a better understanding of PCB design, digital signal routing and analog power supply impedance and noise generation. Time to start thinking on a new version…I built several prototypes and did a lot of testing and listening, before I believed the final version was ready for DIY 😊

Please enjoy building this DAC, it will give several hours of construction pleasure and even more music enjoyment!

B. Features and Specification

The new MK3 design is featuring:

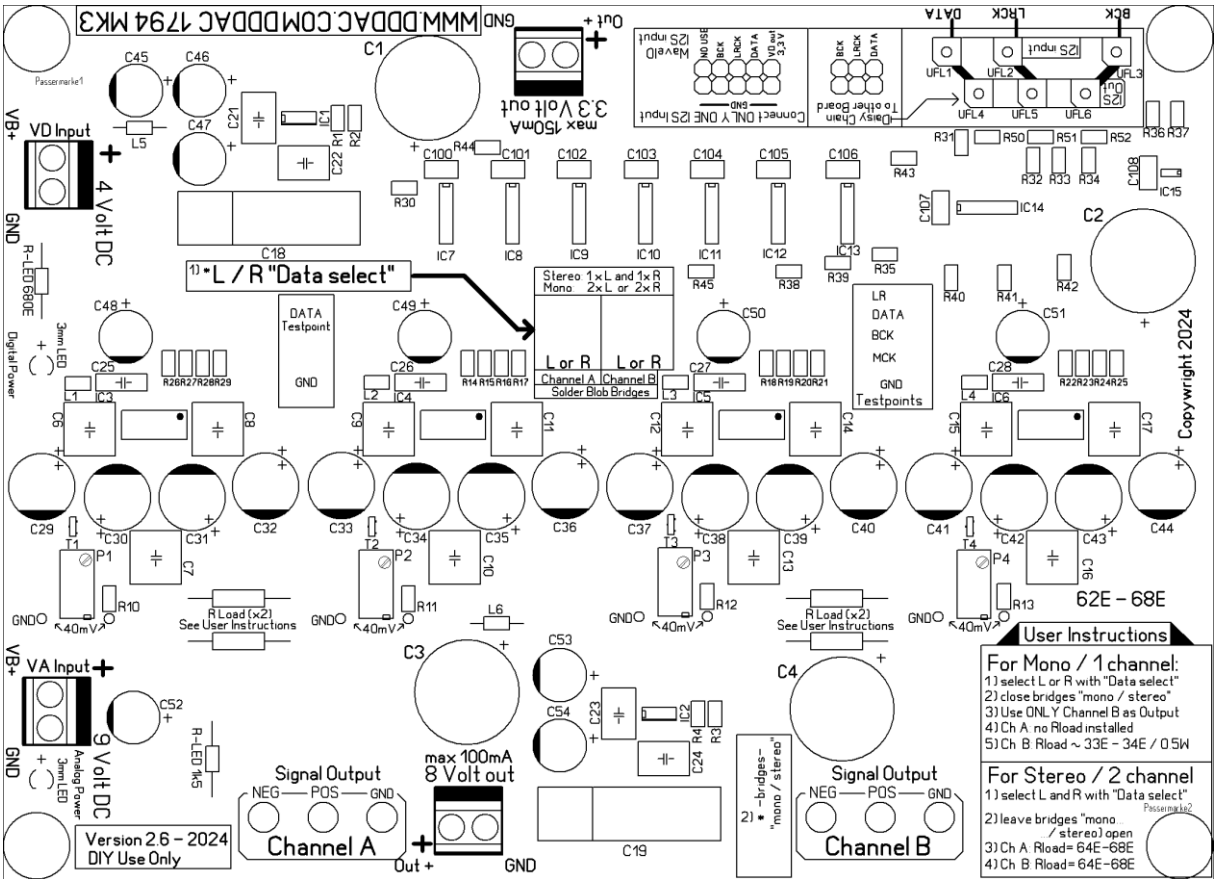
- Fully based on the previous DDDAC1794 concept & design (No digital Filtering etc.)
- 4 Layer Gold Plated PCB for optimum signal routing and low impedance power / ground planes.
- Digital section with modern high-speed CMOS Logic.
- Independent MCK digital signal line (still BCK signal) with a one gate Buffer for optimum and low jitter clocking of the PCM1794 ICs.
- Design with 4x PCM1794 DACs on one single board – equivalent to old style two-decks.
- Selected capacitors for every specific function, based research and long year experience and feedback from the user community of the previous DDDAC1794 series.
- Two channels A&B with both 2 DAC ICs in parallel mode (like previous 2-deck DDDAC1794).
- Each channel can be assigned to be a Left and or Right analogue signal.
- The two channels on the board can be paralleled again for “one channel – Mono” mode (when using two mono boards you get like the old 4-deck equivalent)
- Pin-Header and UF/L inputs and outputs (for daisy chaining multiple boards).
- On-board low 1/f noise voltage regulation, separated for digital and analog section.
- Output terminals of 3.3V and 8V regulated power for external applications or tests.
- More on board SMD parts for better performance and easier DIY construction.

Electrical Specifications:

- Digital Input — I2S only. Inputs are 3.3V - 5V logic level compliant
- Digital Input — I2S up to 384kHz / 24bits
- Analog Output — ~1,2Vrms single ended. ~2.4Vrms balanced (FS0dB)
- Output impedance — ~68 Ohm in Stereo Mode and ~34 Ohm in Mono mode.
- Digital Power Input — 4.0 Volt (~10-50mA)
- Analog Power Input — 9.0 Volt (~ 200mA)
- Digital Power output — 3.3 Volt (recommended < 150mA)
- Analog Power output — 8.0 Volt (recommended < 100mA)

C. Board Layout and Dimensions

Size 130 x 180 mm
Mounting holes 120mm x 170mm - Dia. 3,5mm



D. Completing the PCB with the delivered through hole parts

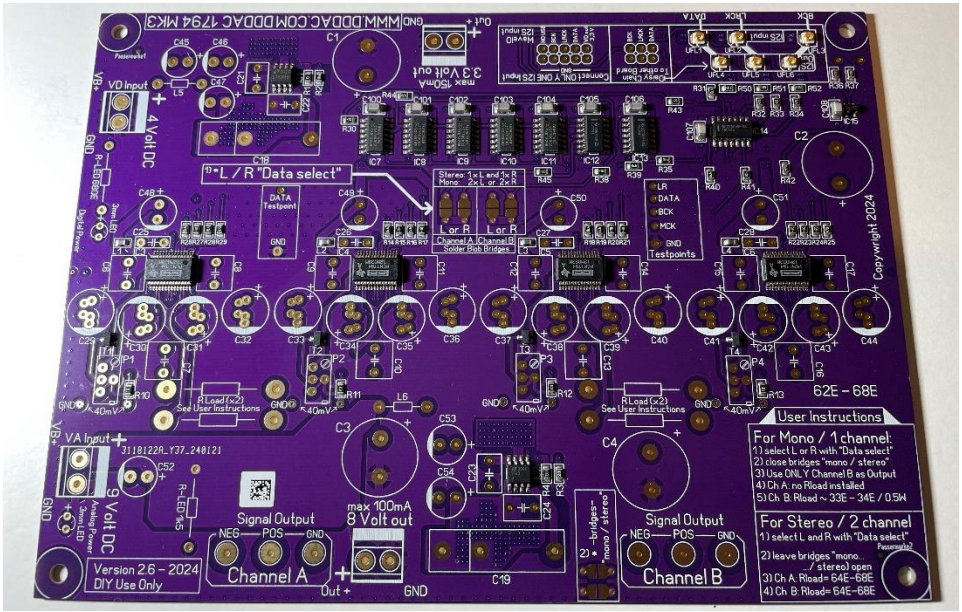
Unboxing

In the Kit you will find:

- PCB with all SMD Parts pre-assembled
- Plastic bag with all the through hole parts and other assembly material
- For the actual kit the corresponding part list – (the part list may differ in futures versions, depending on availability of certain parts. Mostly the capacitors. The parts will be the same sort and family but might have a different voltage range or where it does not matter a different value – For example for the film cap decoupling the input of the regulator, it does not matter if there is 1uF or 1.5uF or 63V or 100V version

Chronological order of building the board

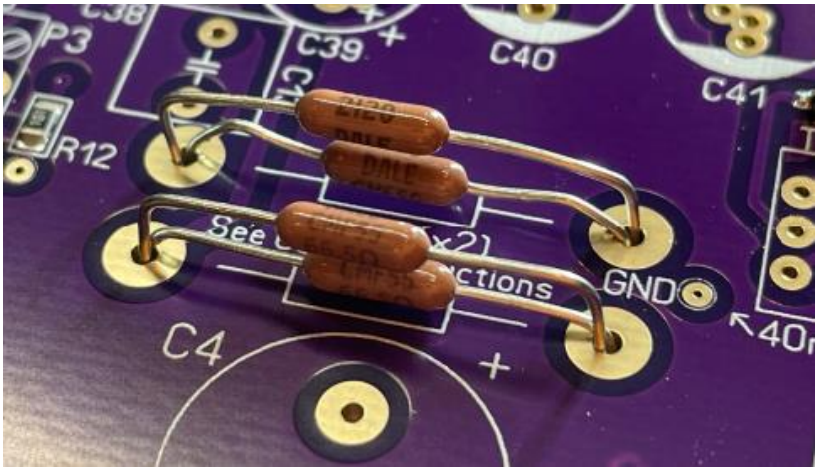
This is the board with the pre-assembled SMD Parts, we start from here.

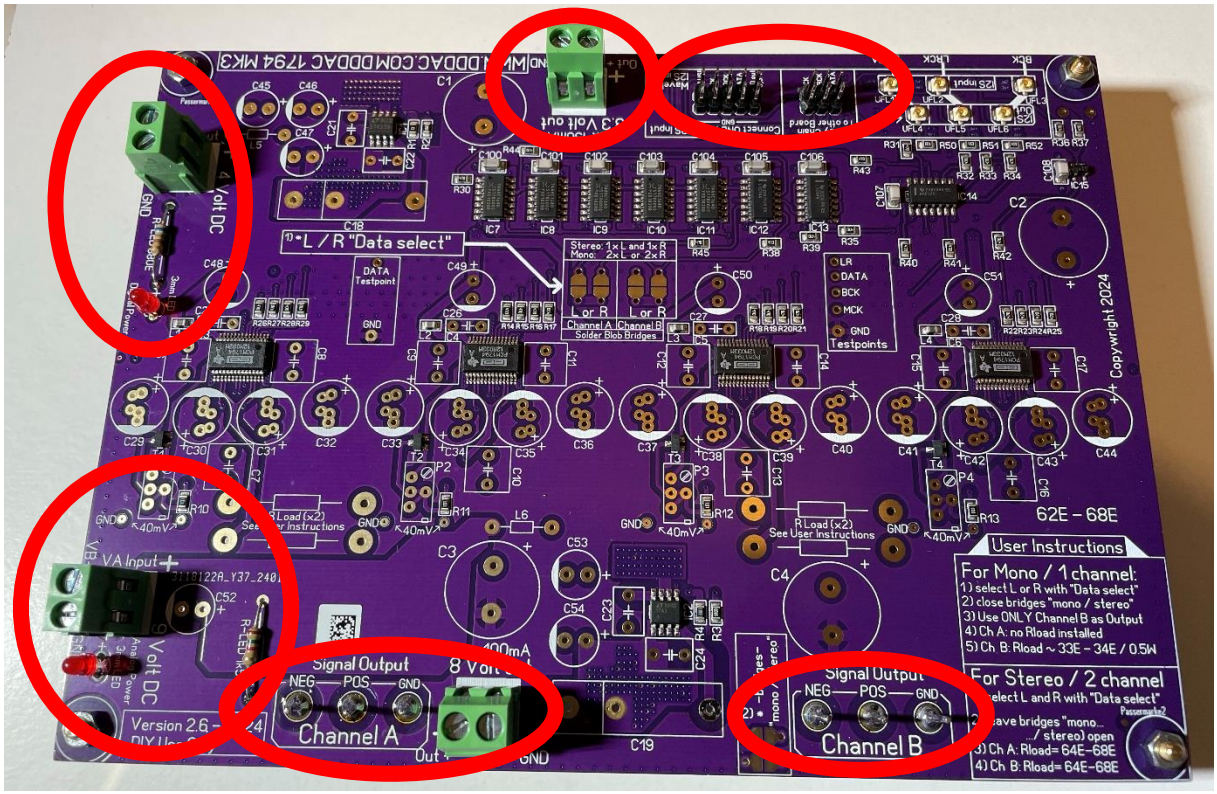


1. Mount the 4x 20mm of the 8 standoffs with the external thread trough the PCB – keep 4x M3 screws aside for later assembly in a chassis. Use the other 4 to mount two boards on top of each other. The space between boards should be 40mm.
2. Solder the 4x 2-pole “PCB Power Connectors”
3. Solder the 6 “Pins” in the analog Outputs (NEG / POS / GND) – It is intentional, that the holes are tight. Apply a bit force with needle nosed plyers to force the pins in the holes

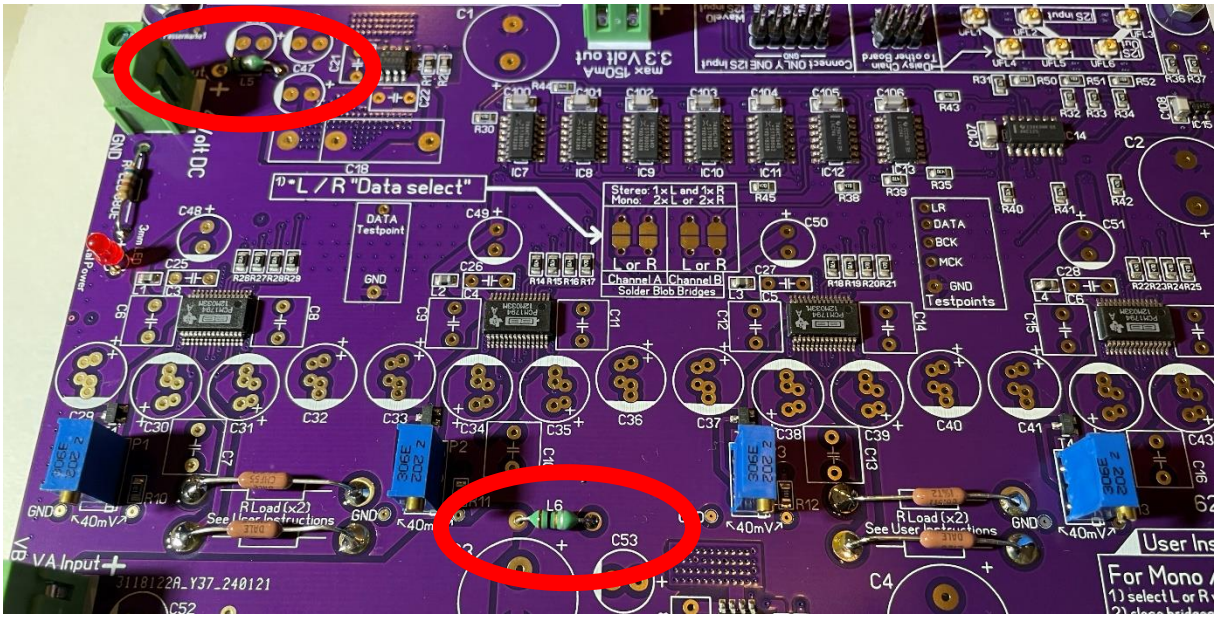
4. Solder the Two RED “3mm LED” next to the DC power terminals. Alternatively, you can move the LEDs to the front panel and just solder wires in this position leading to the front panel LEDs. Feel free to change the color of the LEDs
5. Solder the Two “R-LED” resistors next to the DC power terminals. Note the value 680 / 1k5.
6. Solder the 10p and 6p header at the I2S input at the side of the PCB. Use the 20p header from the kit and cut it into the appropriate size.
7. Install the R-Load resistors. The position and value differ, depending on the purpose of the Board: Stereo or Mono? Read the “User Instructions” at the right down side of the PCB. For 33/34 Ohm it is also possible to parallel the 66-68 Resistors from the kit. The solder pads are large enough to accommodate two resistors at the spot of “R-Load” There is more information in the upcoming section: “Selecting L/R and MONO or STEREO Function of the board”

In case you are building a “mono” board, the B Channel must have like 33 / 34 Ohm installed. Below is an example how do this with the Rloads from the kit. The holes are wide enough for two wires. Of course, you could also install a single 33 Ohm resistor (not in the kit)



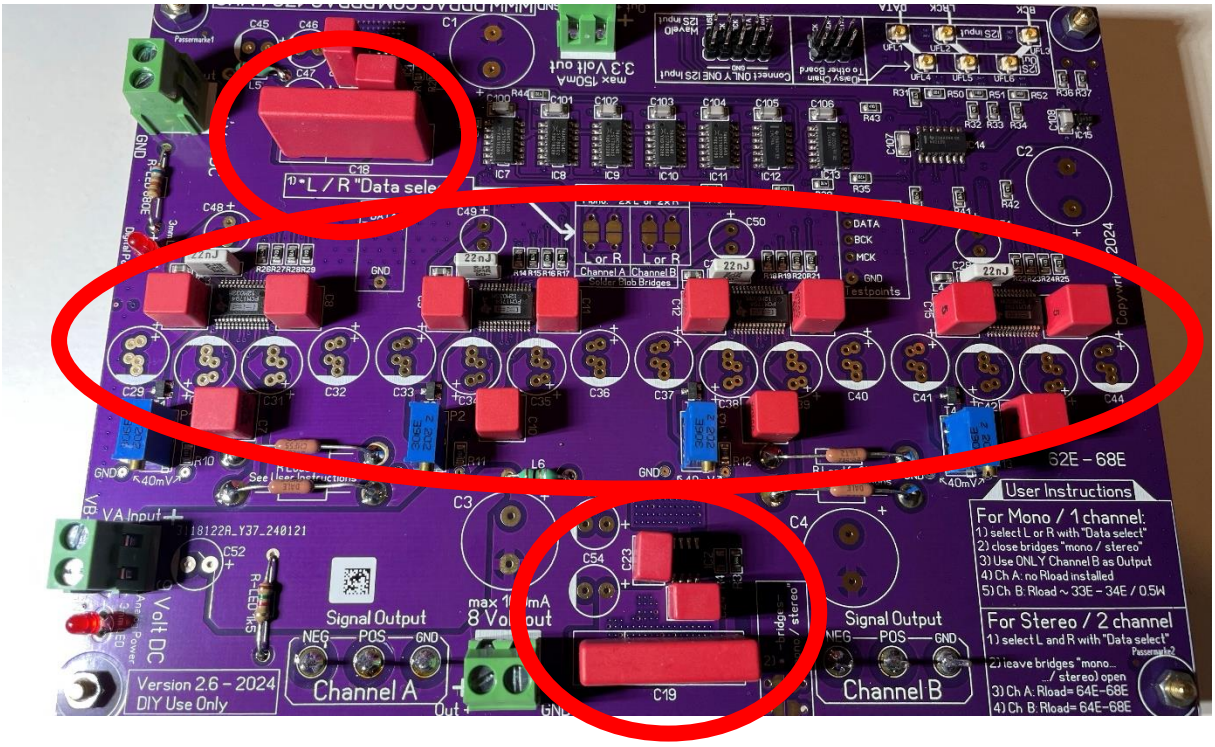


8. Solder the 4x trimming potentiometers P1-P4 (The blue ones in the image)
9. Solder the two inductors L5-L6 (the look like resistors!)

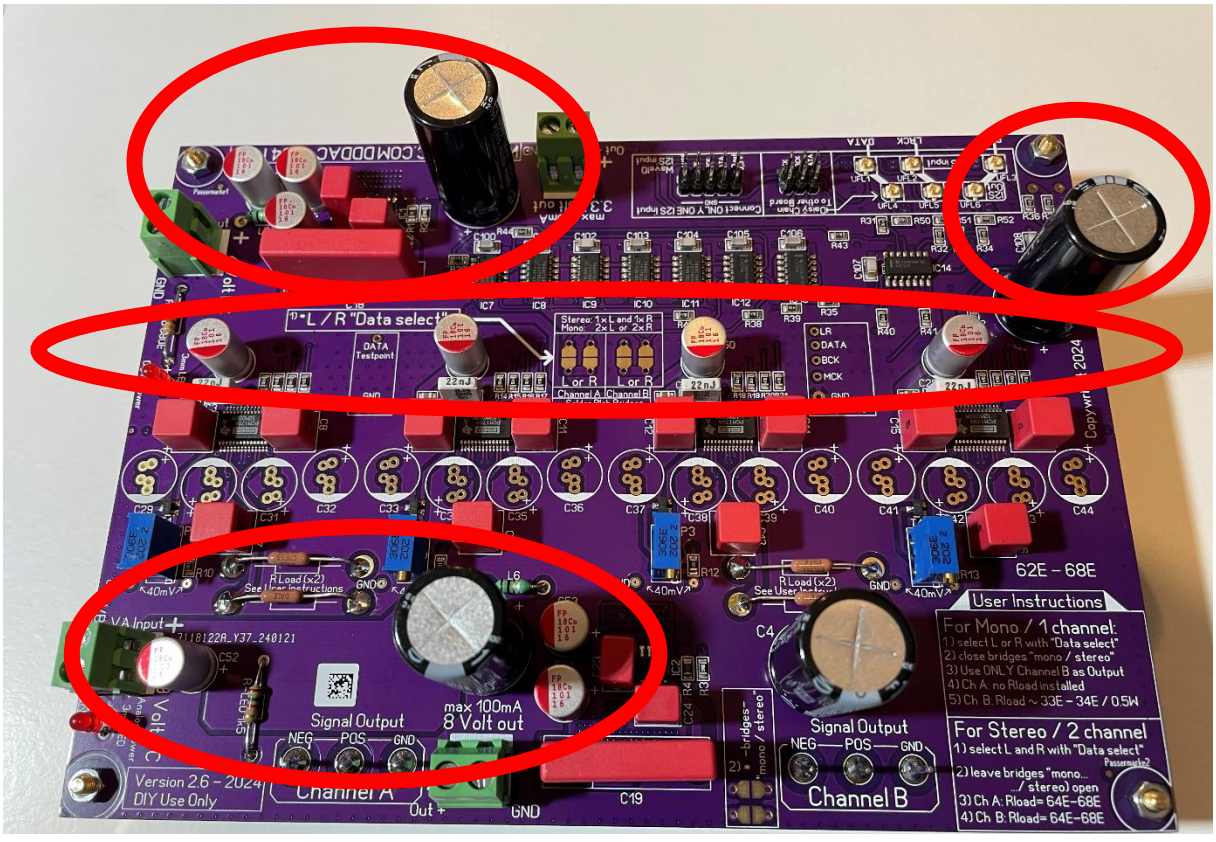


10. Solder the 4x SMR film capacitors C25-C28 – directly next to the DAC ICs (PCM1794)

11. Solder the 12x FKP2 film capacitors C6-C17 – positioned around the DAC ICs (PCM1794)
12. Solder the 4x MKS2 film capacitors C21-C24 – positioned around the Regulator ICs
13. Solder the 2x MKS4 film capacitors C21-C24 – positioned around the Regulator ICs

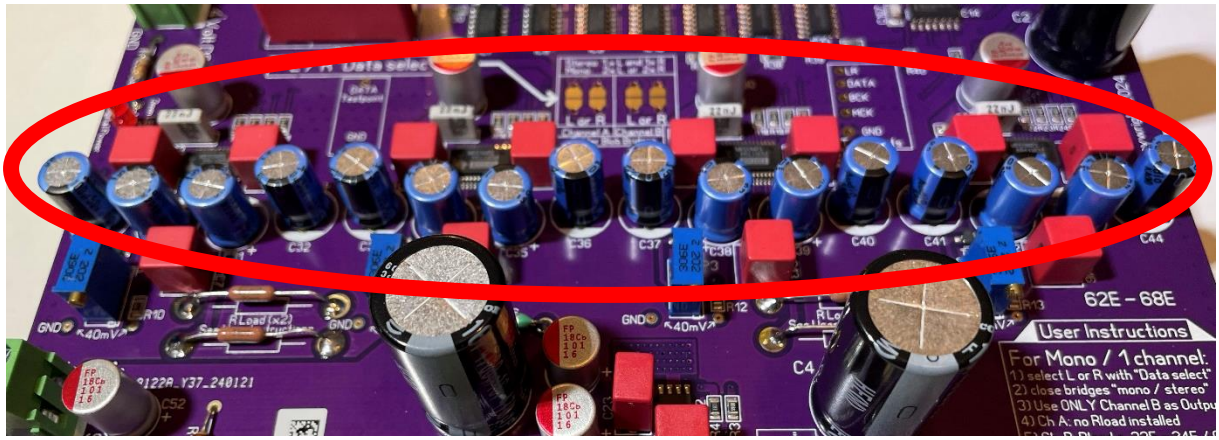


14. Solder the 10x Polymer capacitors C45-C54
15. Solder the 4x Electrolyte capacitors C1-C4



16. Solder the 16x C29-C44 Electrolyte UKA capacitor
17. READY!!

In the image below, I used the Nichicon's "Audio Grade Electrolyte" capacitors (read my Blog for more background)



Selecting L/R and MONO or STEREO Function of the board

As described already in point 7. The DAC Board can be used in stereo and mono mode.
You always have to make choices, otherwise there will be NO sound.

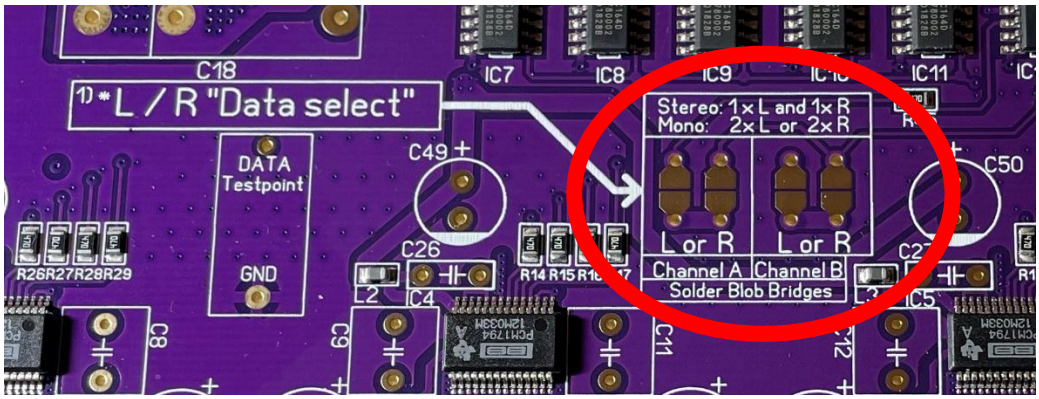
Please follow these steps to set-up the board:

1. Assign Left and or Right Data to channel A or B

   You can define any combination here:

   For stereo you choose Ch. A-L and Ch. B-R or the other way round, it does not matter.
   ONLY solder two options should get a solder blob bridge.

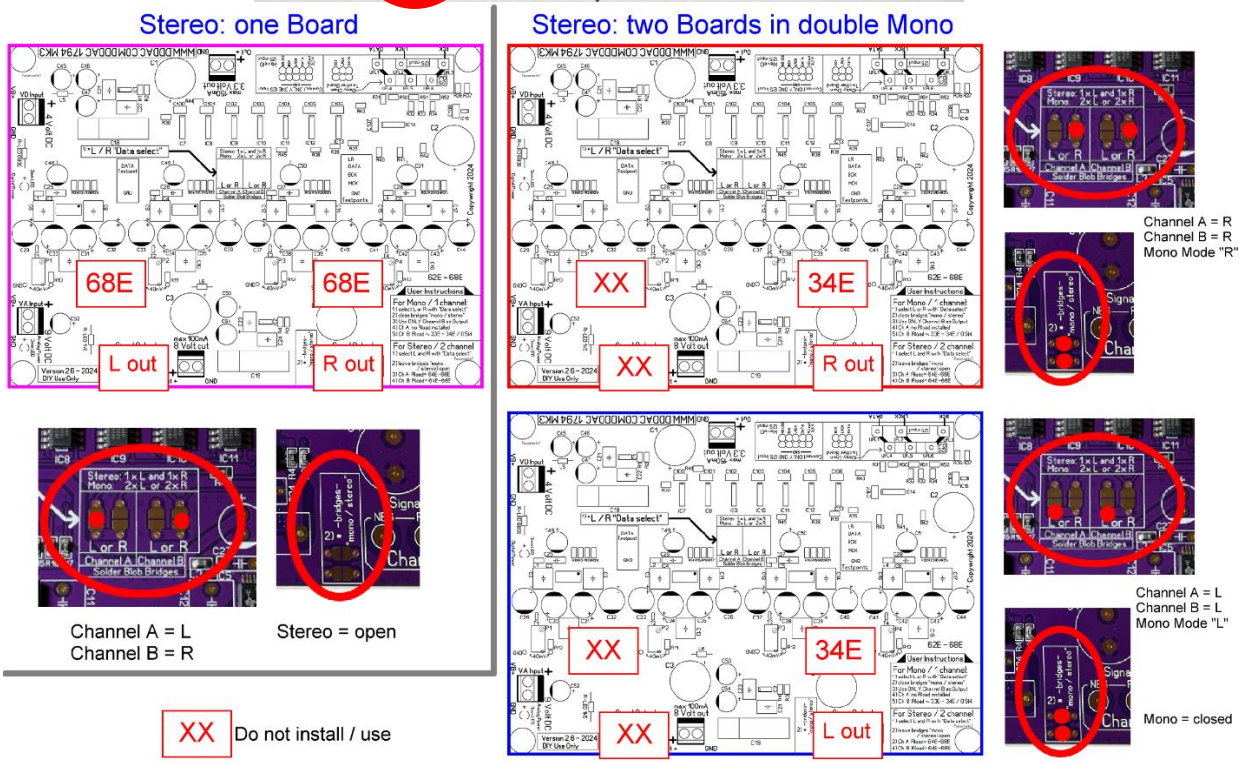
   For Mono you solder/close L or R for both Channel A and B.
   With two boards you select two mono boards: one R and one L so you have stereo again 😉



This how blobs look like:

2. Now you have to assign if the board works in mono or stereo. The point is that in mono the two channels A and B will be put in parallel (Like a 4-deck version of the "old" DDDAC1794. Hence the instruction to only fill the B channel with the Rload resistors. For stereo you obviously need two mono boards: one assigned L and one assigned R…

For mono you have to close BOTH solder pad-pairs with solder blobs. For stereo you leave them open (no paralleling of channel A and B)



That's all. You are now ready to connect the board to the outside world…

E. Connecting the DAC PCB to the outside world

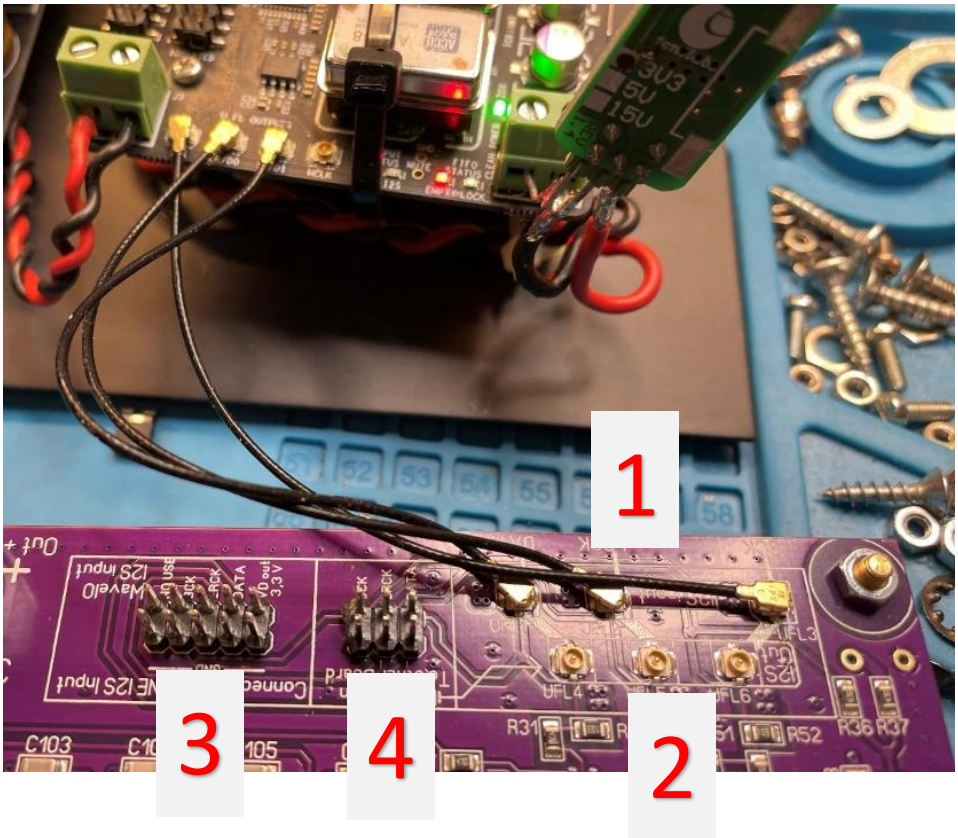
I2S input

The I2S input is 3.3V - 5V compliant.
There are four options to connect I2S signals to the board

1. Through the first row of UF/L connectors (cables are part of the kit) Input or Output
2. Through the second row of UF/L connectors (cables are part of the kit) Input or Output
3. The 5-pin header is intended as Input only from a WaveIO module (It supplies the 3.3V for the isolator on the WaveIO board…)
4. The 3-pin header can be used both as input and output – mainly to daisy-chain two boards

Option 2 and 4 are intended to "daisy-chain" two Boards with each other: leading the I2S signal from one board to the other. The first board gets it I2S signal through option 1 or 3.

The image below shows an example where the I2S data comes from an Ian Canada FiFoPi. You can use option 2 or 4 (NOT 3! To lead the I2S signal to a second DDDAC board for double mono use – for example). This would connect to second board at option 1 or 4 (NOT 3!)

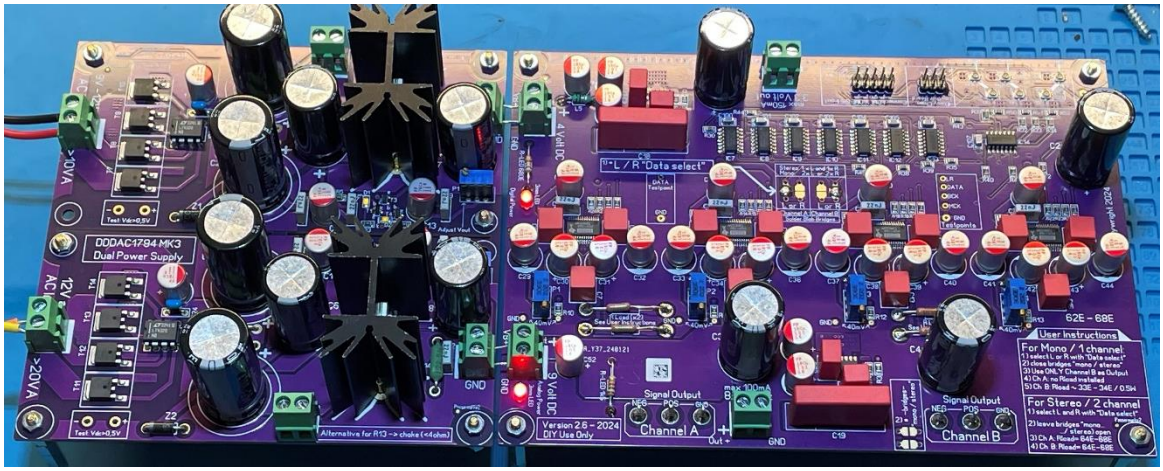


Power supply connections

The DDDAC1794MK3 needs two power supplies. One 4V and one 9V. Both values are not very critical and can be off up to 0,2V without any problem. Never the less, try to keep it at 4V and 9V

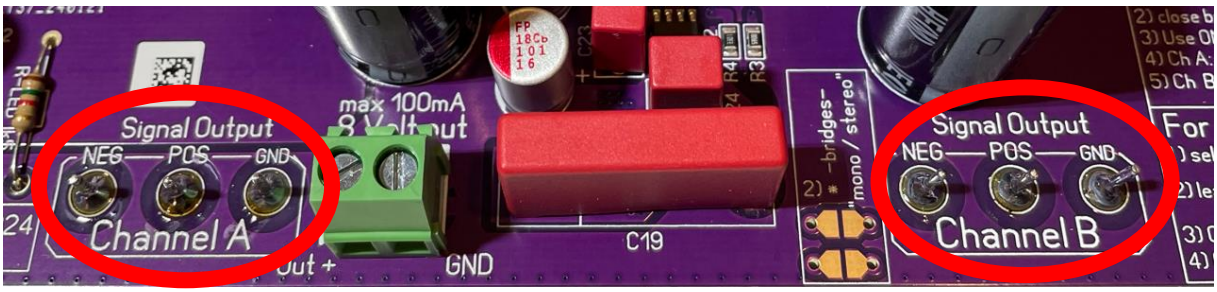
Current draw is depending on the FS of the tracks playing, but roughly:
4V ~20-50mA
9V ~200mA

For delivering the correct power, I designed the DDDAC1794MK3 PSU. It is available as kit and goes smoothly together with the DAC board, electrically and mechanically… see below image.



Analog Output

The analog outputs can be used as balanced or single ended. The signal outputs are indicated as POS and NEG for the balanced outputs. For single ended take one of both and GND, but always the same for channel A and B to avoid 180° phase shift. For Balanced output only connect POS and NEG. If XLR are used pin 1 goes to the chassis. The GND from the DAC must be connected to chassis as well. Best point is the GND from the analog power supply



The kit contains two 1uF MKP couple capacitors for the single ended output. Even though they are of respectable quality, it is advised to change those to “higher end” audiophile types of your personal liking and preference. After the capacitor, solder the 100k bleeder resistor directly at the cinch outputs.

Additional Regulated Power Output Supply Terminals

There are two terminals on the PCB, where the regulated 3.3V and 8V can be tapped. They are connected to the on-board regulators. You can use this for other applications or tests.

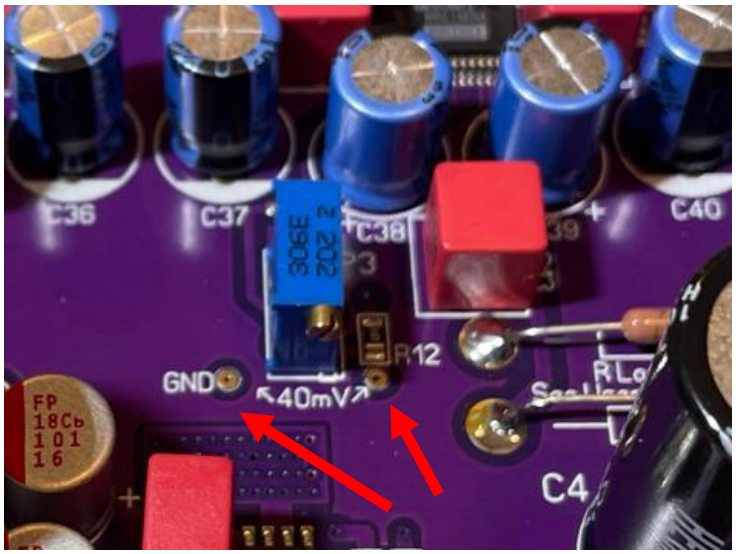
In fact, the max current is only limited by the regulator which is 500mA. As analog draws 200mA you could go up to 300mA. But for dissipation reasons, you should keep some distance to the maximum specs. My advice is like 100mA which should be enough for some external applications.

The digital side has more headroom, but for same reasons I suggest to keep it at kind of max 150mA

F. Bias adjustment and testing the DAC

Bias Setting

Like in the previous version, the DAC needs to gets it DC output voltage and DC Bias Current adjusted for optimum results. It can be done with the 4 trimmer potentiometers. Use a quality multimeter with mV DC range and adjust the bias after 30 minutes warm up. You need to measure at the test points. Not versus any other GND on the board. Set the bias so that the test voltage is as close as possible to 40mV. Check: The NEG and POS outputs versus GND should measure around 2.7Vdc - 2.8Vdc – this value is not critical, but should be for all readings as close as possible within ~20mV from each other. This is important for connection output transformers as the delta between POS and NEG output is a residual DC voltage across the primary of the transformer. This need to be as low as possible, preferably in the mV range. Lower is “better”. If not, check the Rload resistors, may be a bad connection or one with an out of spec value (The DC voltage ate the output depends on the Rload…) This delta can NOT be adjusted with the bias trimmer… It exists due to difference of value in the Rloads

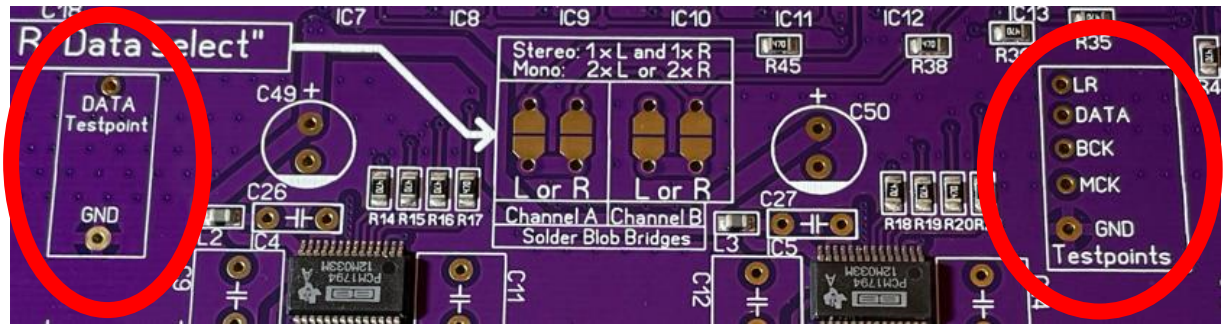


Testing signals and voltages

Voltages can be easily tested at the Power connectors and the analog output pins.

In case of the digital signals, I added a few test point-pads for ease of measurement in case you need to some checking… For Data there are two pads – one for Channel A and one for Channel B.
MCK is basically the same signal as BCK. It is extra buffered from the BCK input and goes as clock signal to the DAC ICs Master Clock (MCK)

These test-points are intended for use with an > 50Mhz oscilloscope and 1:10 Probes for proper results.



"It does not work" – what now?

If you see smoke coming from some parts, there is something fundamentally wrong. There is no real guidance how to fix, as there might be further damage to ICs for example. Better contact me…

If there is just "no sound" or "distorted sound" follow the following check steps:

- Is there 3.3V and 8V (at the power outputs?)
- What DC voltage is at pins POS and NEG versus GND?
- Are the digital signals present at the test points? You need an oscilloscope for this. A "cheap" one (like those USB ones) will do, no need for expensive test equipment. Just see if there is a signal and if the shape looks fine.
- Play a test tone sinus (0dB level – 400Hz for example) and measure the output with a scope or multimeter. BOTH Channel Signal Outputs should be ~1,2Vrms ate BOTH NEG and POS

The above will help deducting where the problem might be. Possible and typical errors:

- Cables are not connected well
- I2S Source is not working properly
- Cold solder joints making bad contacts
- Solder drops making short connections

In general, if you are stuck, please feel free to contact me for online help (Email – pls do not call me). Last resort is to agree to send the defective board to me for analysis and possible repair

G. Possible options and tweaks to improve sound quality

This could be a 500-page book of course, so it will not be a complete Guide. I will just give some hints where to look and what is worth it…

I2S Source

Very simple, the lower the jitter, the better the power supply, the better the result.
Start looking for reclocking devices like FiFos or high-end streamer or build your own.

Also, audio players and connections (cables) can make significant differences. Use good USB to I2S converters or use systems over Diretta. DIY RPI based streamers with Volumino, Gentoplayer or Ropieee are examples where sonic improvement is very well possible.

Power Supply

Buzz words here are CLC choke supply (see my blog site blog.dddac.com). Ultra-Cap Supplies for the I2S source. The DDDAC1794 MK3 PSU is a very good start. Upgrade with bigger transformers and chokes.

Other ideas to look for: Main's purifying, filters etc.

Output Sections

If you go single ended,
there is a ~2.7V DC voltage which needs to be blocked. It is almost mandatory to use couple capacitors of high quality and of your personal liking. Make sure to add the bleeder of 100k after the couple capacitor output to GND – it is part of the Kit

If you go balanced
you can go straight out if your next amplifier has XLR TRUE balanced inputs which are capable of having a common DC voltage at the input. Otherwise use couple capacitors again…

The final option is to use an output transformer.
The benefit of full balanced DAC output and the fact there is no need for a balanced input amplifier, makes the OPT such a good option.

The OPT output is single ended. For example, the Sowter DDDAC OPT which is specifically designed for this DAC. Best results are in double mono, but it can be used without any problems with a one board stereo version. The low frequency cut off is a bit higher (see my blog site for this) but for most systems good enough.

There are of course many other OPTs which work fine with the DDDAC…

H. Disclaimer Brand and Copyrights DDDAC

Just to make sure there are no misunderstanding. All my designs are intended for DIY only. My designs, or any part of it, cannot be used in any commercial enterprise or products. Even when it might have a kind of “private” character.

Anyone making their own designs based on my DDDAC designs are not allowed to infringe my copy rights and intellectual property. For example, you designed your own PCB and sell or even just give them to friends, when on the PCB or in the “sales pitch” there is any reference to DDDAC, my websites or my name. This is not allowed.

Selling anything with phrases like “based on DDDAC” or “DDDAC powered” or similar, are not allowed, unless agreed by me in writing. To be clear upfront: you pay a license for that.

Thanks for your understanding.