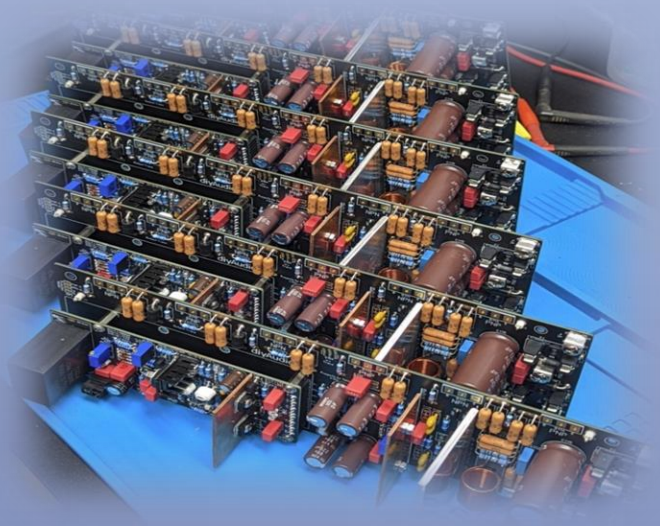


***Including latest VBE modifications*

WOLVERINE Son of Badger IPS & PRECISION EF3-X BUILD GUIDE

07/10/2023

Revision 41

A precision class ab amplifier designed by fanatics for fanatics

YouTube Build Link

Wolverine Build Album

1st Group buy PCB's
Schematic Version 4.0
Wolverine IPS, PCB Version V3.7
Precision EF3, PCB Version V3.9

2nd Group buy PCB's
Schematic Version 4.3
Wolverine IPS, PCB Version V3.8
Precision EF3, PCB Version V4.0

Table of Contents

1. Introduction

Several years ago, Pete Vogel (Ostripper) created a high-performance class A-B amplifier for the DIY crowd. It was eventually given the name "Honey Badger". The excitement it created among enthusiasts around the world ultimately generated even greater interest. The requests and "what if's" of many of those enthusiasts eventually evolved into the design and development of a follow-up amplifier. The idea was to create an amplifier having stellar distortion performance characteristics capable of driving difficult loads.

The conceptual thinking went further than that. The debate over sound quality and IPS topology resulted in various IPS designs being put forward by fellow DiyAudio members in order to figure out which one is "best". That debate was never settled and likely never will, but the situation presented an awesome opportunity. The decision was made to design the amplifier with a modular IPS board, allowing the builder to easily switch to a different IPS topology without changing anything else…. Wolverine, "Son of Badger" is the result and it's the first of many IPS boards planned for the Precision EF3-3 (3 output pairs) and EF3-4 (4 output pairs) output stages.

1.1 Specifications

Note: The Specification are still be finalized and will be updated once they are completed.

Class	AB
Rated Output (Power Supply Dependent)	240W + 240W (8Ω) Rail Voltage 71Vdc 430W + 430W (4Ω) Rail Voltage 71Vdc 200W + 200W (8Ω) Rail Voltage 64Vdc 360W + 360W (4Ω) Rail Voltage 64Vdc 160W + 160W (8Ω) Rail Voltage 57Vdc 280W + 280W (4Ω) Rail Voltage 57Vdc 120W + 120W (8Ω) Rail Voltage 50Vdc 210W + 210W (4Ω) Rail Voltage 50Vdc
Input Impedance	Unbalanced 10KΩ or 22KΩ
Input Sensitivity	1.8V/ 10KΩ/ 200W (8Ω) 1.7V/ 22KΩ/ 200W (8Ω)
Gain	24.0dB @10 KΩ 24.5dB @22KΩ
Frequency Response	20Hz to 20kHz (+0, -0.1dB) 5Hz to 180kHz (+0.1, -3.0dB)
Total Harmonic Distortion	0.000009016% (-140.9dB) @1kHz/ 80W/ 8Ω 0.0000950% (-120.4dB) @20kHz/ 80W/ 8Ω
IMD	0.0001096% (-119.2db) [f1 = 19Khz and f2 = 20Khz] 0.0001122% (-119.0dB) [f1 = 1Khz and f2 = 5.5Khz] 0.0002818% (-111.0dB) [f1 = 250hz and f2 = 8Khz] 0.0002818% (-111.0dB) [f1 = 60hz and f2 = 7Khz]
S/N Ratio (IHF-A)	117dB
Slew Rate	67 V/µs
No Output Devices (Per Channel)	3 or 4
Damping Factor	110 @ 20Hz (4 output devices)

2. Amplifier Build: Key Component Selection

Where to start when approaching a build like this…

Let's face it, there's always a certain level of excitement when first starting a project like this. That said, we all know that the best results are achieved by going slow and taking your time to do it correctly. We highly recommend that the builder take the time to carefully read through this entire build guide and understand all the steps before proceeding. If something isn't making sense to you or more explanation is needed, don't be shy about reaching out to the forum members.

It's important to understand that there are decisions that you must make before starting the assembly of your PCB's. These decisions have to do with the selection of certain components and their values based on your specific system needs and configuration. Before buying parts, it's necessary to understand the loads your speakers will impose on the amplifier output stage. This is a necessary step because it defines the operating conditions that the amplifier must endure without compromising sound quality and without failure. Having this information will allow the proper selection of the power supply transformer and the number of output transistors. As well, there are additional selections to be made based on your rail voltage; these are all listed in the bill of materials. Further information to help with the selection of the power supply transformer, driver, and output transistors can be found on Sheet 2 of the bill of materials.

Note: if you are new to this hobby and do not have a power supply (reservoir capacitors, bridge rectifier, etc.) If you plan to build one suitable for this amplifier, it's highly recommended that you research this topic to fully understand the requirements. There are numerous resources available, and it is critical to understand what is necessary to safely power this amplifier. Rod Elliott of ESP sound products has written numerous articles on this topic and it's a great place to start. Rod Elliott - Power Supplies

An excellent book for the less experienced DIY'er is "High-Power Amplifier Construction Manual" by G. Randy Sloane, "Designing Audio Power Amplifiers" by Bob Cordell, "Audio Power Amplifier Design" by Douglas Self, just few of many books authored by subject matter experts.

2.0 Transformer Selection

There are two types of transformers typically used to power audio amplifiers.

One configuration uses a center tap. Which is the midpoint connection of two secondary windings and typically cannot be separated.

Another configuration has two or four completely separate secondary windings, depending on the number of power supplies used in your chassis. These are the preferred secondary winding options which you should consider when choosing your transformer.

These configuration examples are shown on sheet 2 of the BOM.

The choice between dual secondary windings and four secondary windings depends the number of power supplies being used in your chassis. Dual secondary windings are preferred for a single power supply, while four secondary windings are preferred where two power supplies are used.

2.1 Input Impedance Selection

The input impedance can be chosen before building the amplifier. 10k is the standard option but the design allows 20k with some additional changes. Modern hi-fi sources should be able to easily drive 10k, but tube preamps, older devices, portable sources or consumer gear may perform better with the higher 20k input impedance. However, this comes at a cost of 3db higher noise, so is not the default.

2.2 Inspect & Clean

I. *Take some time to perform a visual inspection of the PCB's. Check the surface quality, and look for the existence of dents, scratches, pinholes, and other defects on printing traces or pads.*

II. *Isopropyl alcohol is a great PCB cleaner because it is inexpensive and evaporates quickly. Wipe over the boards with a small cotton cloth. This will ensure any oily residue will be removed from the pads prior to soldering.*

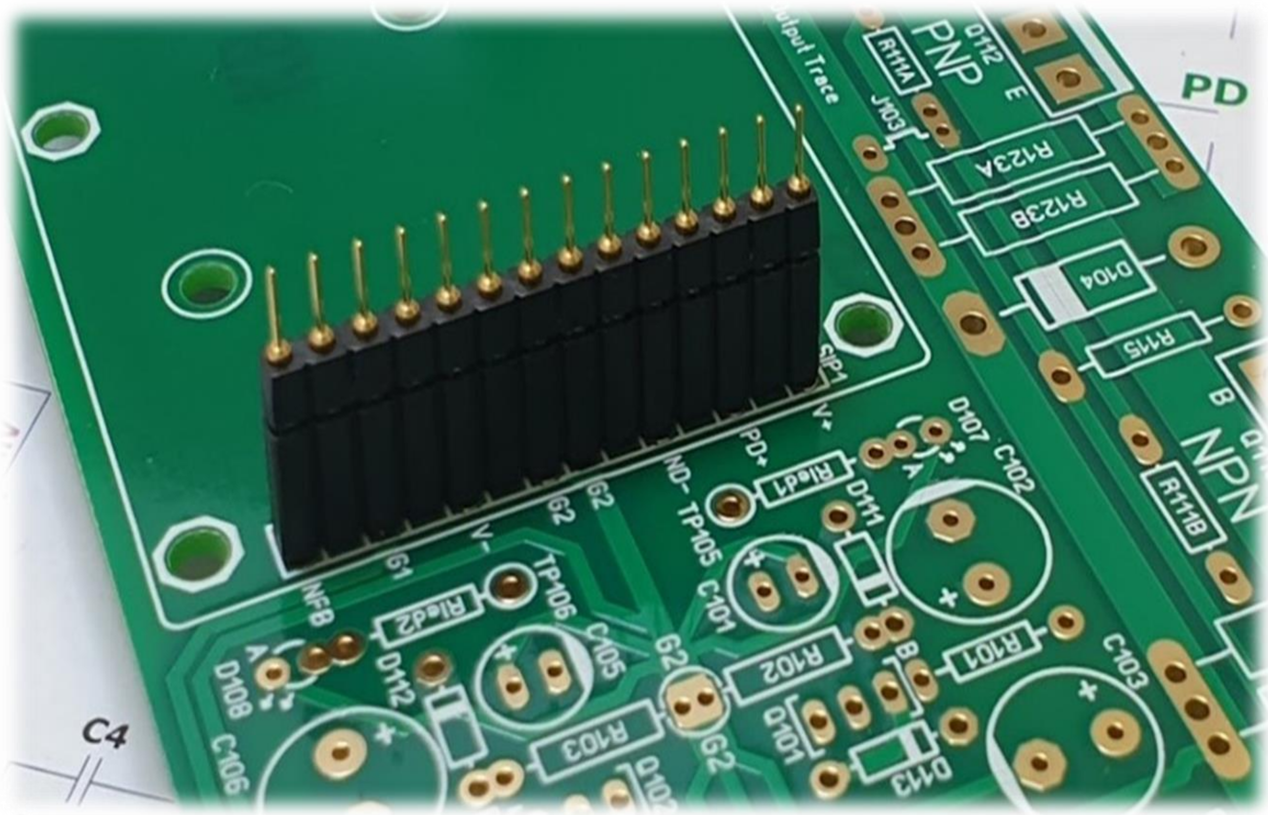
3. Starting the Build-Inspect & Trial-Fit Hardware

Note: For reference, the component layout silk screen is provided at the end of the build guide. Once components are installed and their component ID's are covered this reference will certainly come in handy.

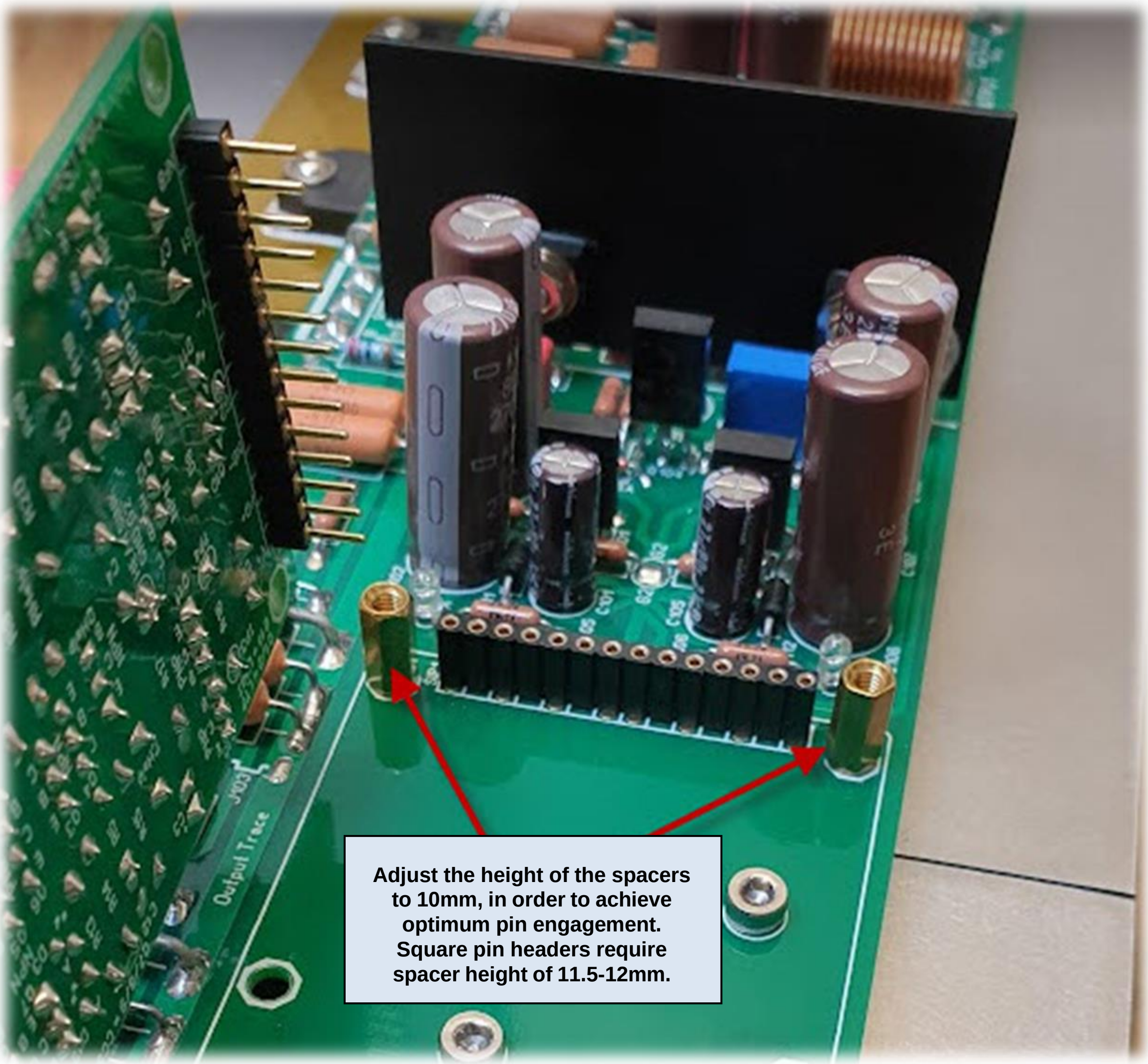
I. Start the build process by trial-fitting the IPS board with the EF3 board. It's important to ensure the proper fit of the fastening hardware, spacers and the 14-pin SIP1 header (No soldering is done at this time). Spacer/stand-off height should be adjusted as necessary to ensure that the upper and lower pin headers fully engage, but are not under compression. The objective is to have the boards fit together without putting the boards or pin header under any physical stress.

II. **Important:** The round pin headers are considered the best choice for this application. There are two length combinations listed on the BOM 10mm & 7.5mm. They have 2.54mm lead spacing and are shorter in height than the square pin headers (11mm) listed in the BOM. The Spacer height is adjusted accordingly. The ideal round header pin assembly requires a minimum spacer height of 10.0mm (0.393").

III. Care must be taken with the male header as the pins on each face are a different diameter. The larger of the two diameters is intended to engage in the female header.

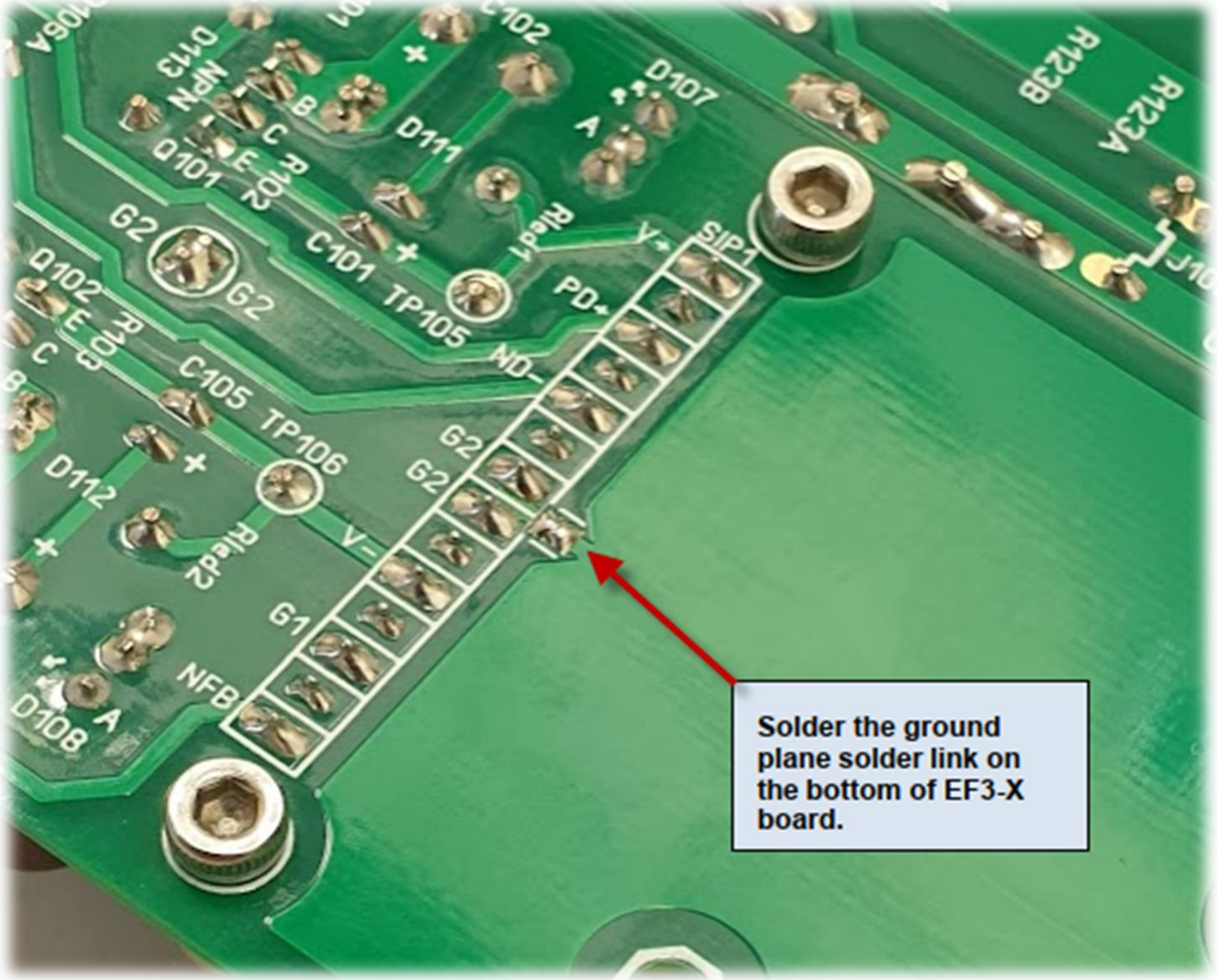


SIP1 Header Assembly



IV. Another Important Point: When it comes time to solder SIP1 Header in place.
   i. Connect the IPS board to the EF3-X board using the correct length stand-offs.
   ii. At this point neither the male nor female headers should be soldered in.
   iii. Tightened up the screws that attach the stand-offs to the IPS & EF3-X boards while monitoring the alignment and engagement of the header pins.
   iv. As neither SIP1 headers have been soldered they should self-align.
   v. Once the screws are tight and upper and lower SIP1 headers are fully engaged but are not under any compression, solder both male and female SIP1 headers in place.

This should ensure the best alignment of the IPS and EF3-X boards.

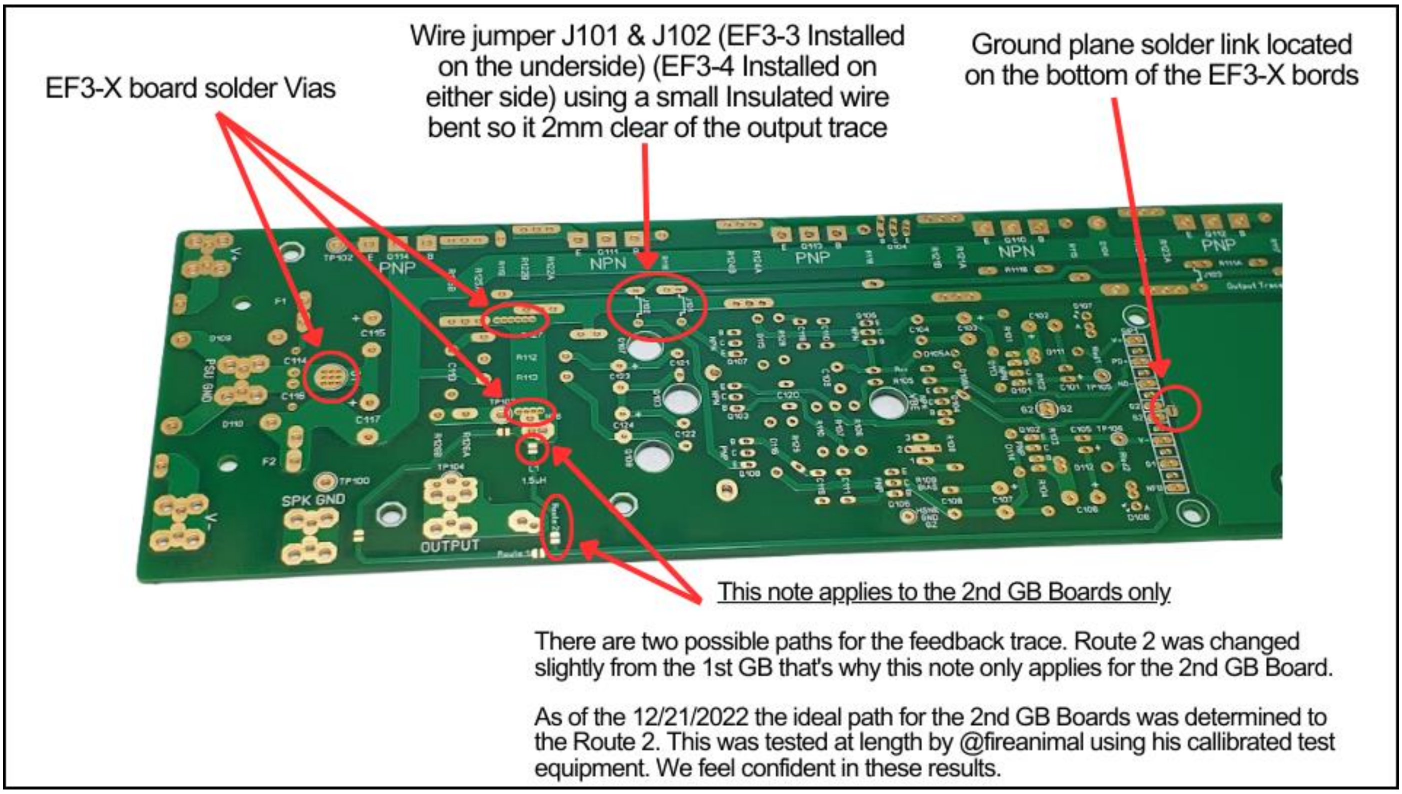

Solder the ground plane solder link on the bottom of EF3-X board.

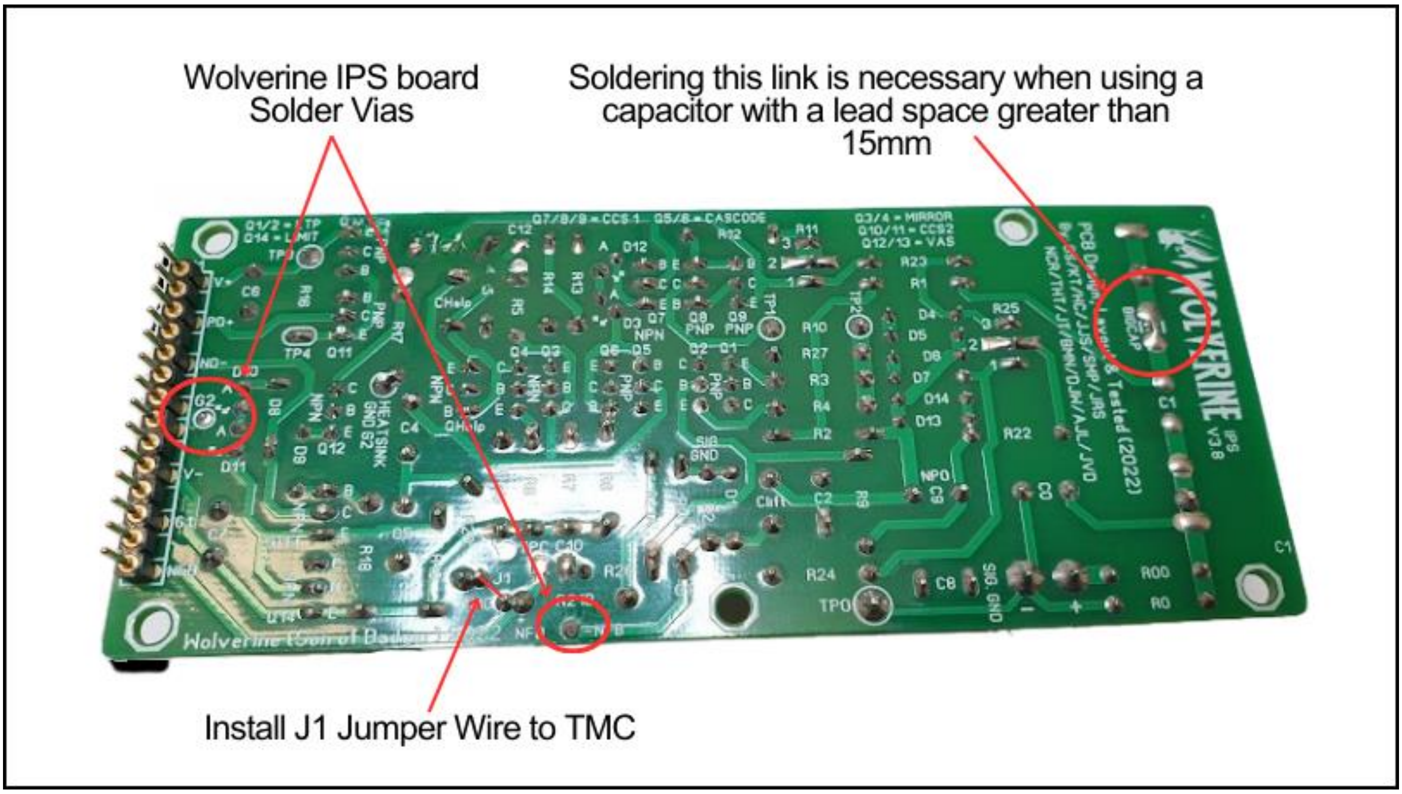
4. Install Links / Solder Vias

It is standard practice to install all the small, low-profile components first. Start by installing all the wire jumper links and soldering Vias. Vias are used to transfer power or signals to the copper layer on the opposite side of the board. It's recommended that each via be soldered through to the opposite side by using a fine wire to help wick the solder through to ensure a solid connection.

- There are 2 jumpers to install and 3 via locations to be soldered as per the above instructions on the EF3-X boards.
- The IPS board has 1 jumper wire link, 1 solder link and 2 solder vias.
- There is also 1 solder link to be bridged. This connects the G2 ground to the IPS Shield.

Further details are shown on the next page.


EF3-X board solder Vias
Wire jumper J101 & J102 (EF3-3 Installed on the underside) (EF3-4 Installed on either side) using a small Insulated wire bent so it 2mm clear of the output trace
Ground plane solder link located on the bottom of the EF3-X bords
This note applies to the 2nd GB Boards only
There are two possible paths for the feedback trace. Route 2 was changed slightly from the 1st GB that's why this note only applies for the 2nd GB Board.
As of the 12/21/2022 the ideal path for the 2nd GB Boards was determined to the Route 2. This was tested at length by @fireanimal using his callibrated test equipment. We feel confident in these results.


Wolverine IPS board Solder Vias
Soldering this link is necessary when using a capacitor with a lead space greater than 15mm
Install J1 Jumper Wire to TMC

5. Install SMD Capacitors

The IPS board has one and potentially two surface-mount capacitors. Now is a good time to install C10, a surface mount capacitor that requires a steady hand. Installing C10 prior to installing any other components provides greater access for such a confined soldering area. The other potential SMD is C12. This capacitor is listed in the BOM as an SMD or through-hole device. If you opted for the SMD device, now is the time to install it. If you opted for a through-hole device, it can be installed after the resistors and diodes. An Alligator clip can be used to hold the SMD capacitor steady while one side is soldered.

6. Heatsinks

I. Wolverine IPS and Precision EF3-X board, heatsink details (Excluding output transistors)
   i. Using the EF3-3 board, the combined IPS and EF3-3 board have 4 transistor heatsinks. Two are used on the IPS board and 2 are used on the EF3-3 output board. The three heatsinks need to be hand fabricated.
   ii. Using the EF3-4 board, the combined IPS and EF3-4 board have a few options for transistor heatsinks arrangements.
   iii. Tightened up the screws that attach the stand-offs to the IPS & EF3-X boards while monitoring the alignment and engagement of the header pins.

- *If you plan on perpendicular mounting the output transistors to the amplifier PCB's please refer to point i. above.*
- *If you plan on mounting the output transistors parallel to the amplifier PCB's there are two options.*
- *You can use the same arrangement as in point i. above or use the preferred option below at point iii.*

II. Where applicable, fabricate the heatsinks as described in the illustrations and mount the transistors, in order to trial fit the assembled heatsinks on the board. This is done to ensure that the components are in the correct position and leads are oriented correctly. Where applicable, for those transistors that have exposed non-insulated tabs, insulating pads and shoulder washers must be used to prevent shorting to the heatsink. These details are further described in the illustrations.

III. Perform a continuity check to make sure that there is no electrical connection between the heatsink and the leads of the transistor. When done, set the assemblies aside. They will be soldered in after the surrounding components are in place, allowing you to judge the preferred height of the heatsink assembly relative to the surrounding components. It's recommended to mount the heatsink elevated approximately 3-4mm above the board to allow air to circulate. Trial fit the driver heatsink (when used) first to provide a good reference for mounting height.

IV. A TO-220 finned heatsink is listed in the BOM for Q7 and Q8. This assembly can be completed for trial fitting as well. As mentioned above, the use of insulating pads depends on the selection of the transistors used. Note that the heatsink can be assembled in either a high or low-elevation orientation. Opting for the elevated mount position will allow better airflow/cooling, particularly when residing next to Q5 and Q6, which tend to operate at elevated temperatures.

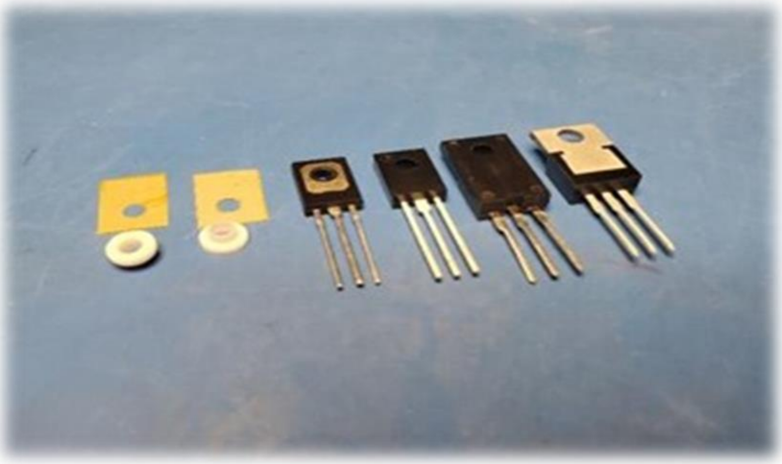
V. The heatsink assembly can be mounted facing in either direction. The silkscreen on the board indicates the correct orientation and the transistors must be mounted on the heatsink accordingly.

6.1 Heatsink Assembly Notes

In general, 3mm button head cap screw fasteners are recommended to mount transistors to heatsinks. A good assortment of metric fasteners is typically available at local hardware stores.

Nylon shoulder washers and insulating pads are available at just about all reputable electronics parts' stores. The insulating pads are identified by the transistor "package", TO-126, TO-220, etc. Note that some TO-126 types are fully plastic encapsulated on their mounting surface and do not require additional electrical isolation. In fact, the addition of insulating pads reduces heat transfer and causes higher running temperatures.

Depending on the transistor used, shoulder washers, insulating pads or Kapton tape is required to electrically isolate the transistor from the heatsink. It's critically important that the builder follow up assembly by using a DMM to verify that the leads of the transistor are not in contact with the heatsink.



A. Before Installing the transistors on their heatsink and in particular the drivers and output transistors it is vital to confirm the following:
   I. Make sure the surface of the heatsink is clean and free of any debris or contaminants. The use of compressed air is useful for this task, if its available. Cleaning the heatsink with isopropyl alcohol is also a good idea.

   II. Inspect all transistor mounting holes. Use a deburring tool or a countersink bit to remove any burrs, raised surfaces or sharp edges around the tapped hole. This step is crucial to ensure a flat and even mounting surface for the transistor.

   III. Once installed, ensure that the transistors are secured correctly. Each transistor should sit flat and evenly on the heatsink with no gaps. A thin layer of thermal grease is recommended.

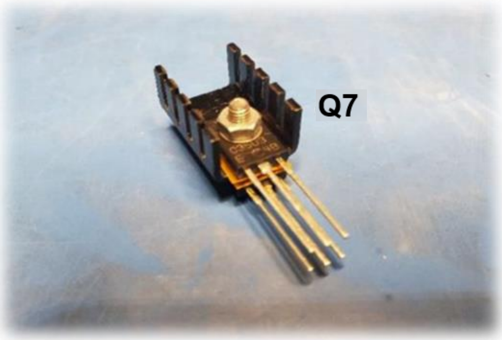
   IV. It is essential for the transistor to sit flat on the heatsink to achieve optimal thermal heat transfer, maximize contact area, eliminate air gaps, enhance heat dissipation, and avoid thermal stress. These factors will contribute to the efficient operation and long-term reliability of your audio amplifier.

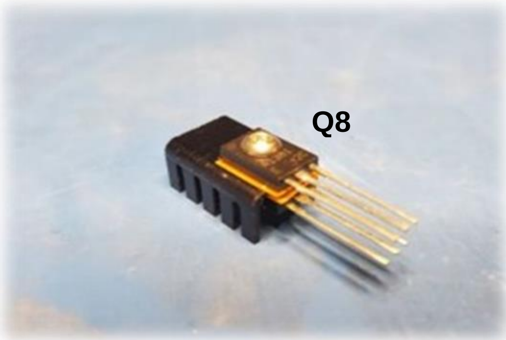
Note: There are many resources available regarding the proper mounting of transistors on heatsinks. If you're new to this subject, here is an excellent source for further information. https://www.giangrandi.org/electronics/thcalc/thcalc.shtml

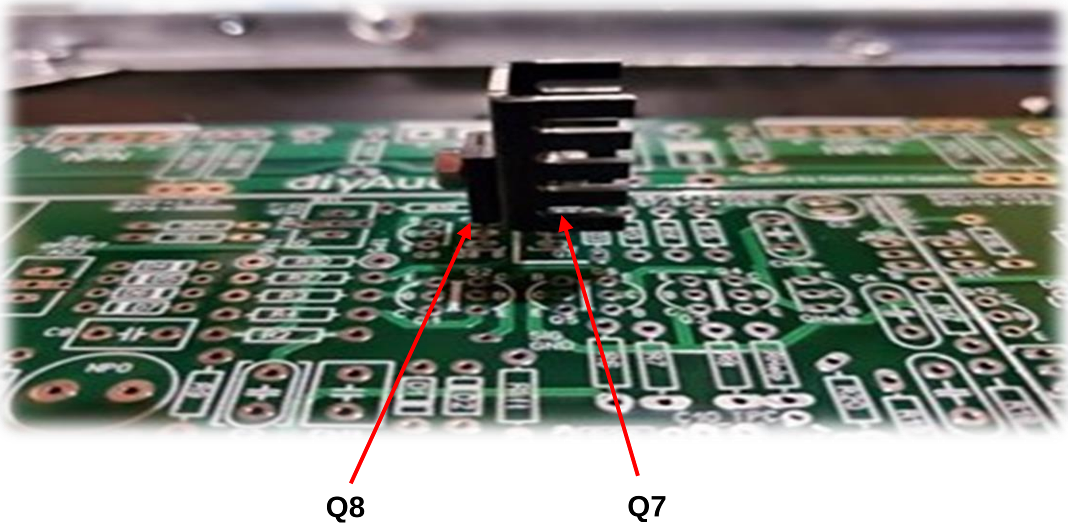
6.2 CCS1 Heatsink

The heatsink used for CCS1 is a TO-220 finned heatsink.
It is listed in the BOM by Mouser part# 532-577102B00. Insulator Pads were used in this assembly due to the exposed die area on the back of the Sanyo transistor. On semi 3181/3503 devices are entirely encapsulated in plastic, so insulator pads are not required. Examples of these devices are seen above.

Thermal grease is commonly used to improve the transfer of heat from the device to the heatsink. We advise that the builder use this very sparingly, as only a thin film is required. There are many articles written on this subject, so if you're new to this topic it would be a good idea to check it out.

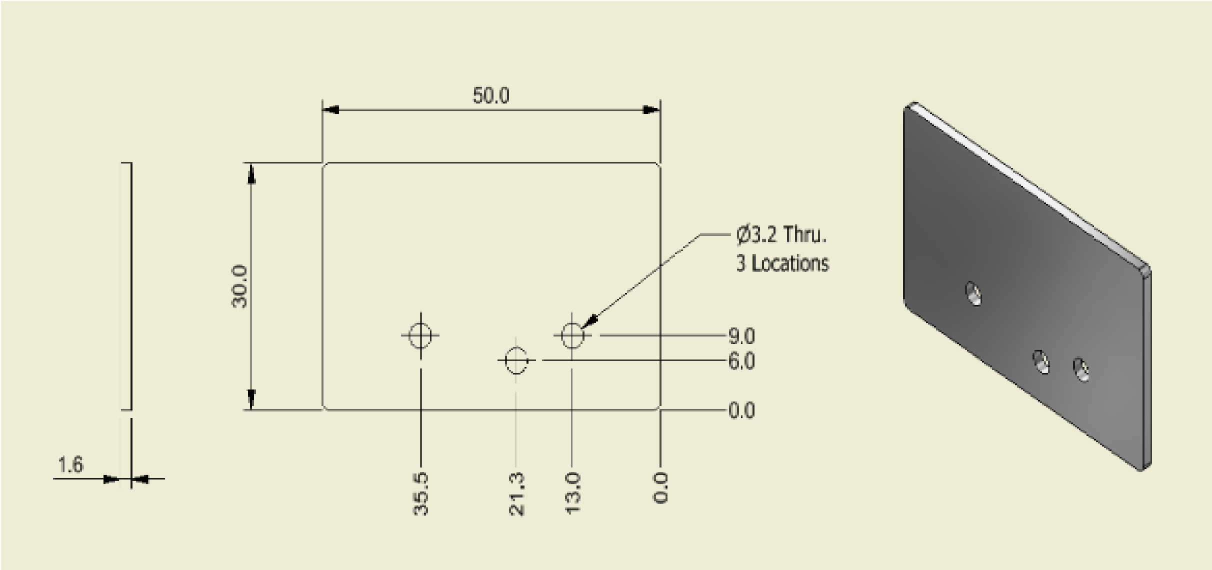




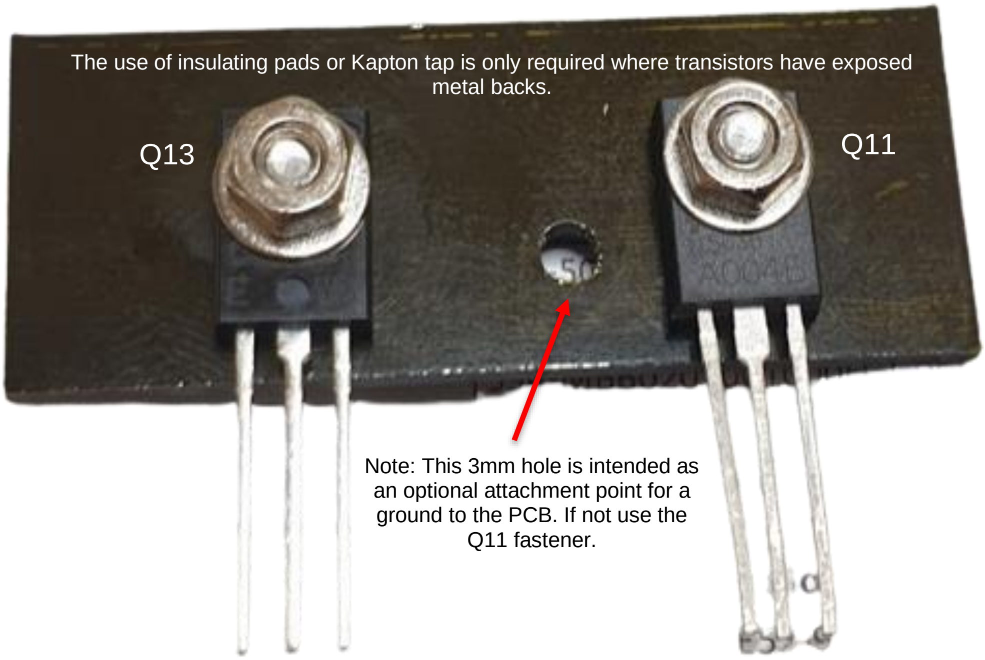


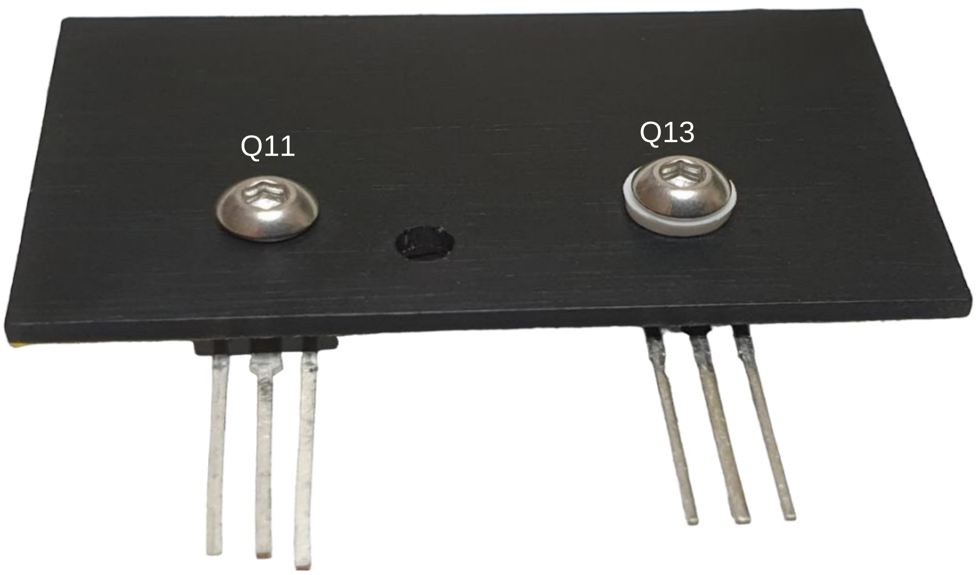
6.3 CCS2 and VAS Heatsink

50mm x 30mm x 1.2mm (Minimum)



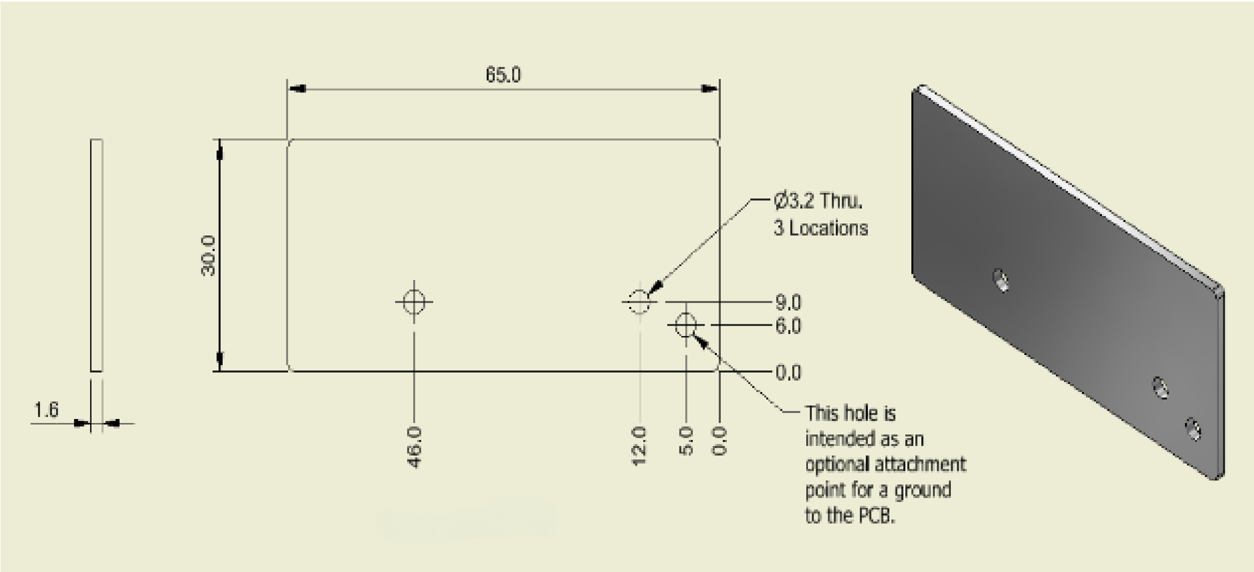
CCS2 & VAS Heatsink





6.4 Pre Driver Heatsink

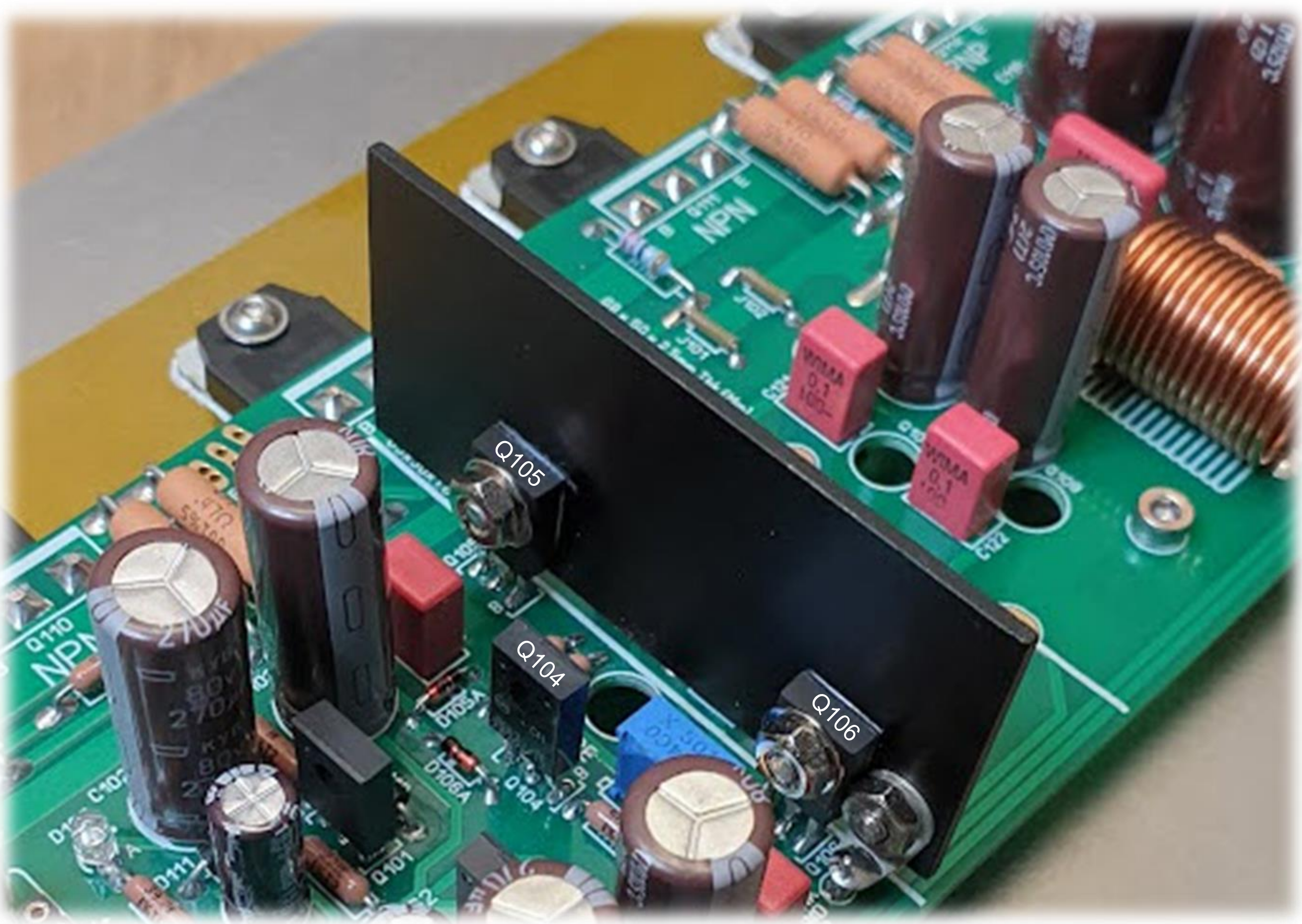
65mm × 30mm × 1.6mm

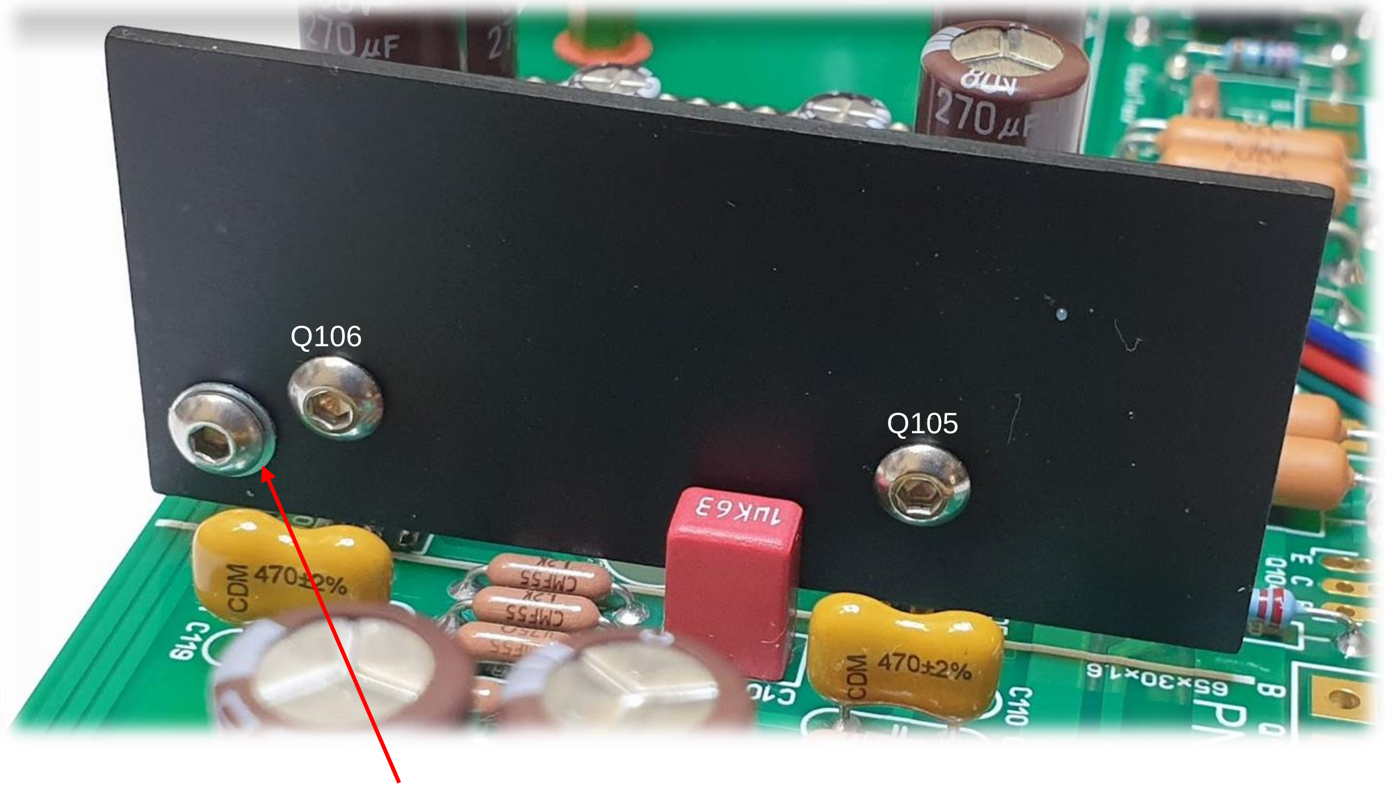


Pre-Driver Heatsink

Note: Drill the holes to accommodate 3mm screws only when installing transistors that are fully insulated. Transistors such as early Sanyo are exposed on the back, requiring the use of insulating shoulder washers and gaskets to prevent shorting to the heat sink. In this case, holes would be sized to accommodate the fit of the shoulder washers.

The use of insulating pads or Kapton tap is only required where transistors have exposed metal blocks.





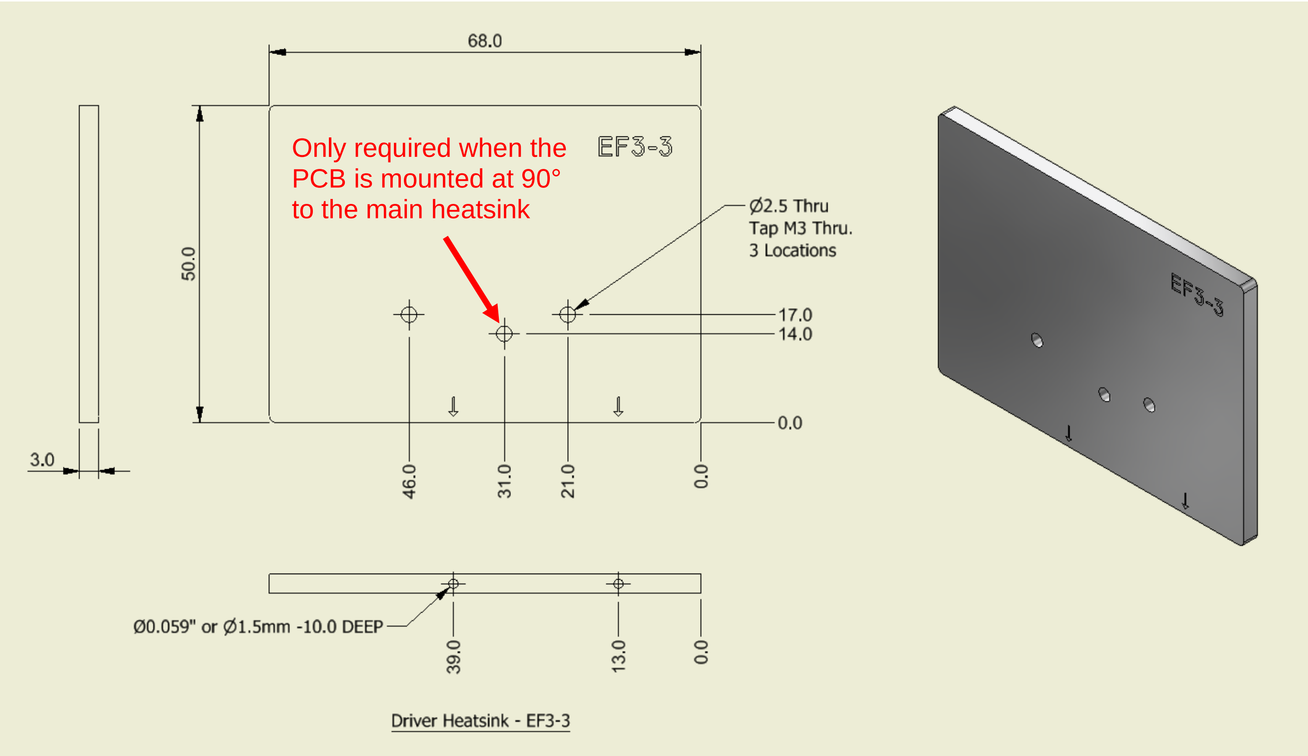
If the heatsinks are anodized, scrape off the anodizing to ensure a good connection to ground. The use of a shakeproof washer is also recommended.

6.5 EF3-3 Driver Heatsink

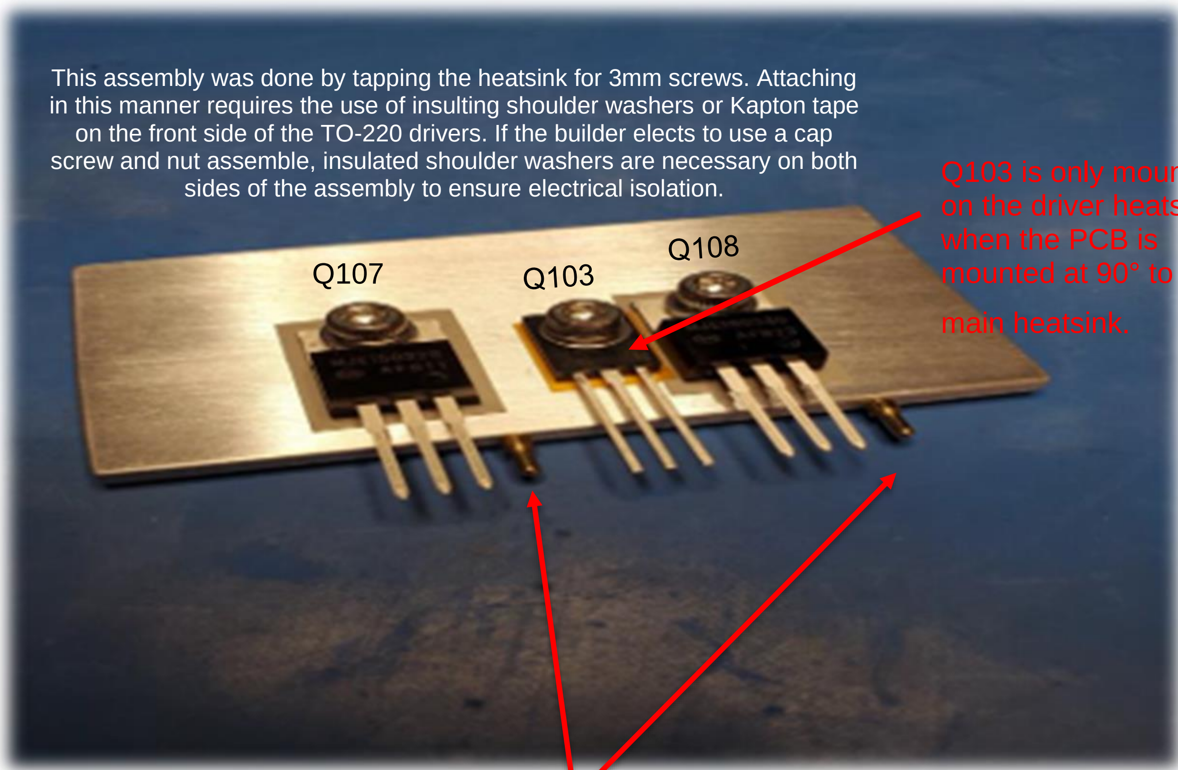
50mm × 68mm × 3.0mm

Note: The length of the driver heatsinkcan be extended to 74mm if the outputs are mounted parallel to

68mm (74mm optional see note)



Driver Heatsink - EF3-3



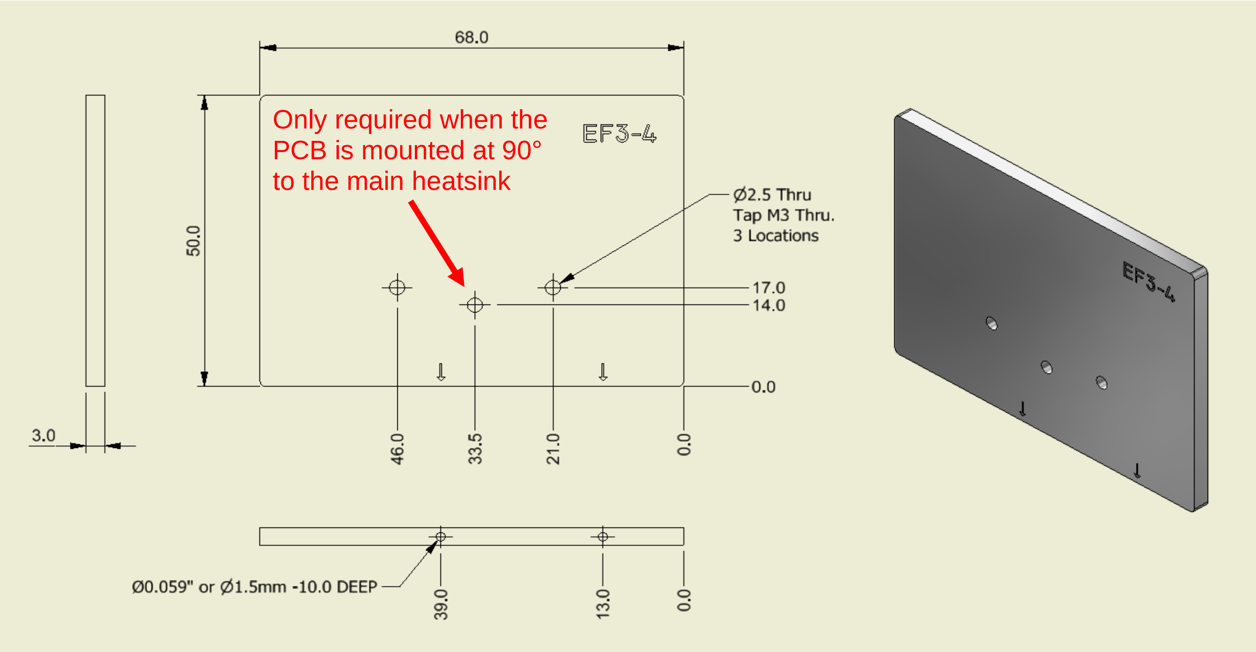
Note: These pins are grounded to the PCB.

The EF3-4 pair board has slightly different driver transistor spacing dimensions as shown below.

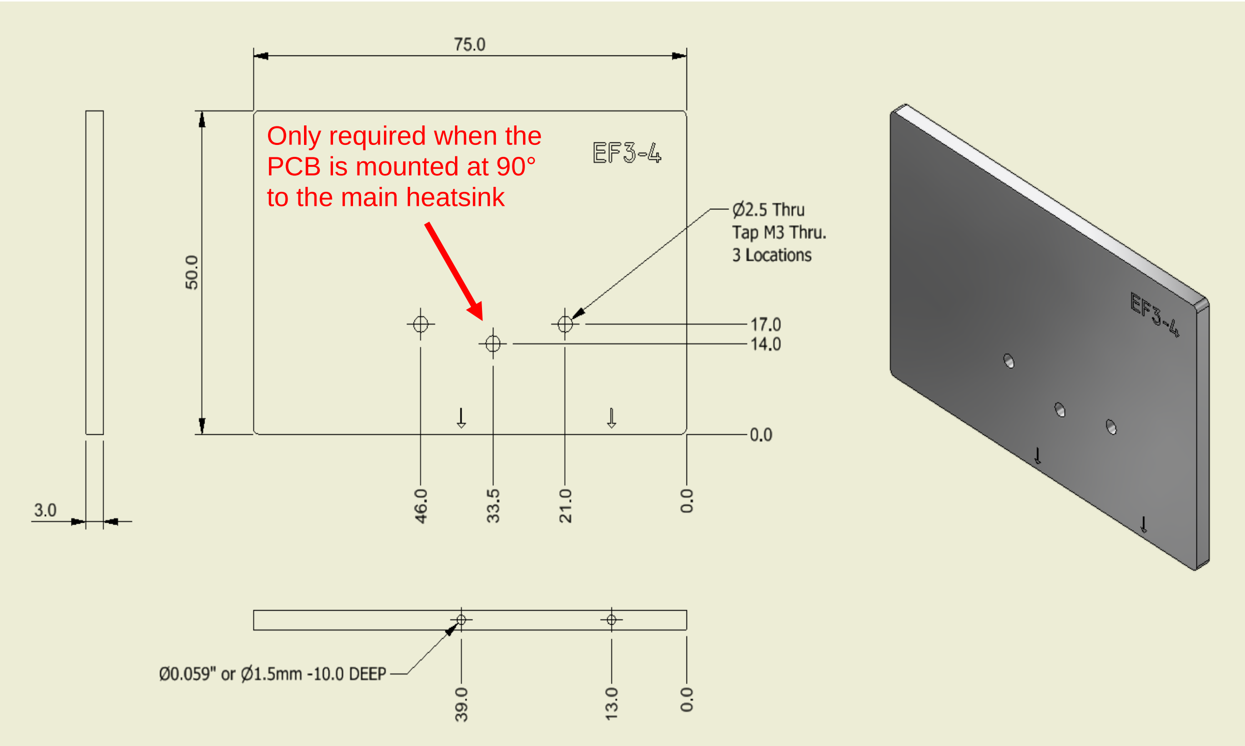
6.6 EF3-4 Driver Heatsink

50mm x 68mm x 3.0mm

Note: The length of the driver heatsink can be extended to 75mm if MT-200 outputs are not used.

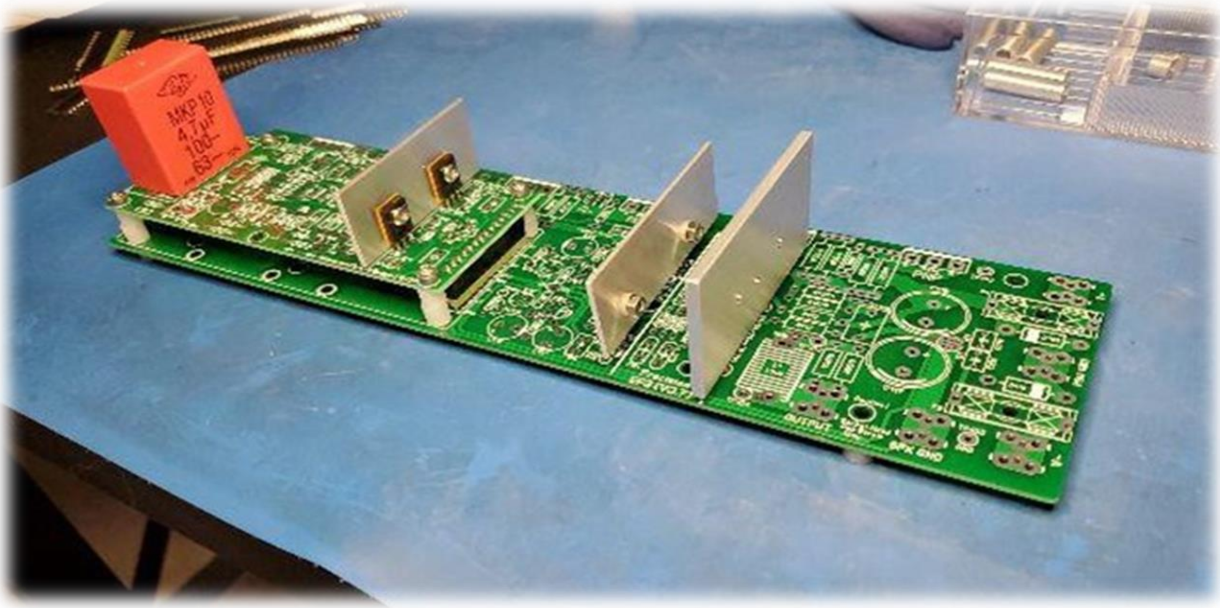


Driver Heatsink - EF3-4

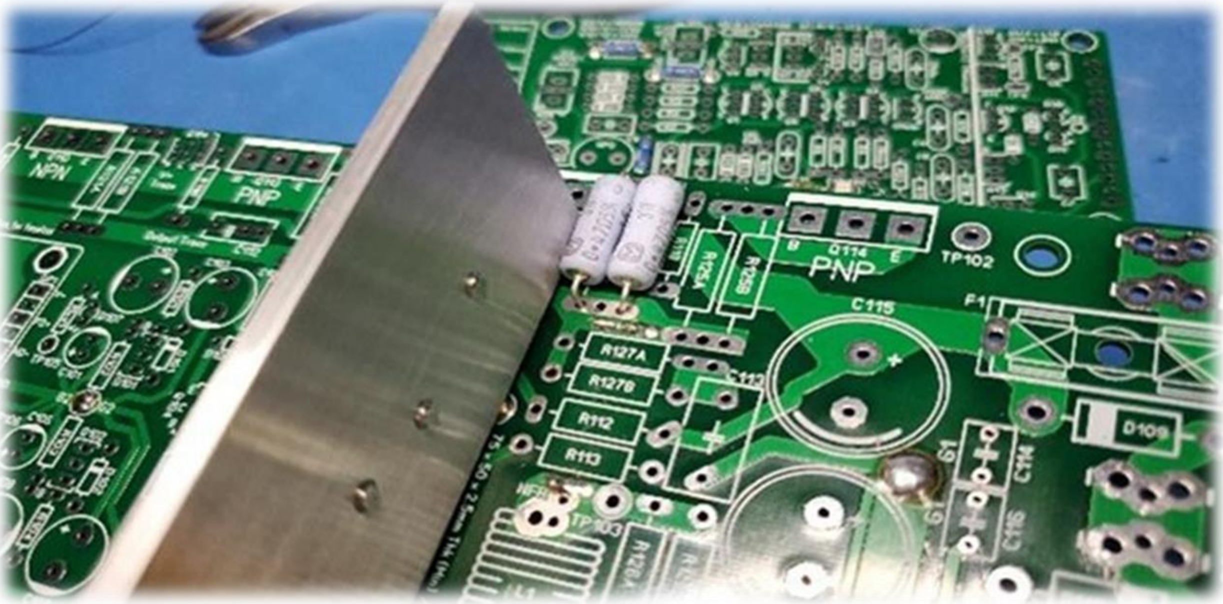


Driver Heatsink - EF3-4 Extended

Mockup of assembled EF3-3 heatsinks and mounting hardware



Note: Depending on what you sourced for the emitter resistors, some larger-bodied resistors can be a tight fit. Now is a good time to check your clearance to the driver heatsink. If necessary, simply file a slight relief on the edge of the heatsink.



7. Install Resistors & Diodes

I. It is recommended for optimum performance that all resistors rated for 1/4W – 1W are 1% tolerance metal film. Resistors rated greater than 1W are often only offered as metal oxide with a tolerance of 2% or higher, but will work fine. The BOM provides guidance for the selection of those resistors.

II. Resistors having a power rating greater than 1/2W should be installed slightly elevated (~1-2mm) above the board surface to promote heat dissipation.

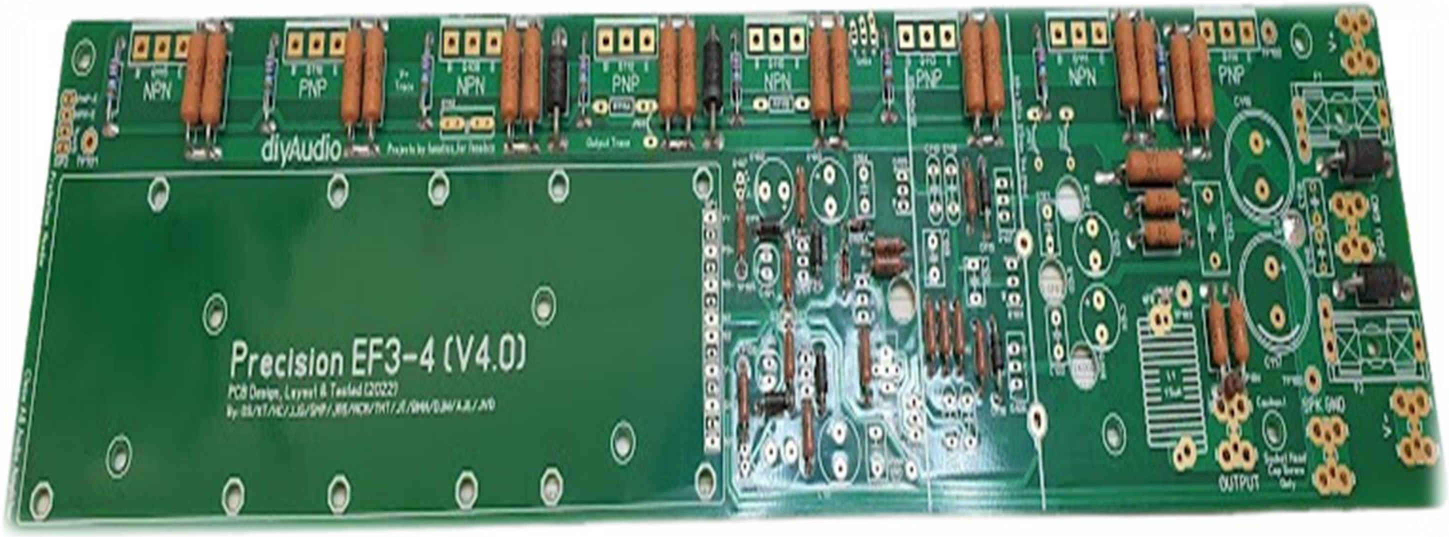
III. Resistors that also serve as test points may also benefit from being installed slightly elevated to allow the Test Hook Clips to be easily attached. Namely TP1, TP2, TP3, TP4, TP105, TP106, TP107 & TP108

IV. The various colors used in the manufacture of resistors often make it difficult to accurately interpret the color bands. To prevent the chance of mistakes, take the time to measure each resistor to ensure that it meets the specified value before installing it. Failing to do this can be a costly mistake.

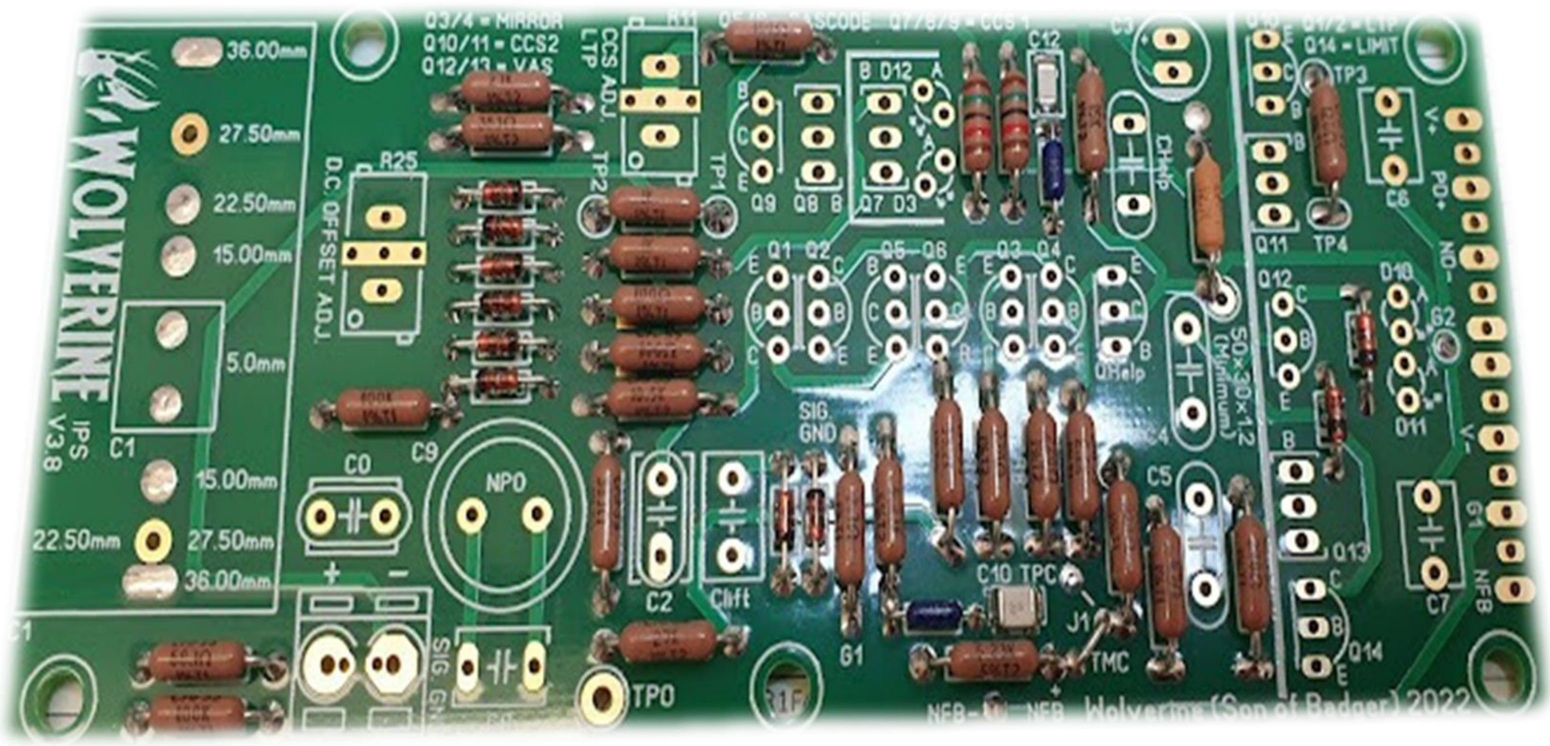
V. Install resistors paying close attention to the power ratings and values listed in the BOM for your rail voltage. Specifically, R6, R15, R17, R20, Rled, Rcc.

VI. Carefully match the values of resistors R3/R4 and R7/R8 to within 1% of each other. The closer the better, as this is critical to the paired performance of LTP Q1/2 and current mirror Q3/4. For those inclined to want to know a little more about this: https://www.brown.edu/Departments/Engineering/Courses/En162/IC_figures3.pdf

VII. Observing the polarity markings on the boards, install all diodes and LED’s D1-D14 excluding D3 and D10-D12 on the IPS board followed by D101-D108 on the precision output stage board. Reminder: The long lead on the LED’s is the anode.



EF3-X - Resistors & Diodes



IPS - Resistors & Diodes

8. Wolverine IPS Board – Suggested Component Installation Order

- Installing components left to right, top to bottom, and smallest to largest, can help ensure that have sufficient access to allow your soldering iron room to reach the top of each component pad.
- Try and install components with their component values facing up or in a position where the component value can be read once all components are installed.
- You may need to refer to other sections in the build guide when it's time to install one of the components listed below.
- You may want to use the suggested component installation order as a checklist, crossing off each component as you go.
- When it comes time to solder in the TO-92 small signal transistor. Care must be taken to install them at a height where good access to the top of the pad is available if you install them too low you may burn them with your soldering iron.

C10, C12, R00, R0, R22, R23, R1, D4, D5, D6, D7, D14, D13, R10, R27, R3, R4, R2, R9, R24, R12, D1, D2, Rlift, R21A, R6, R7, R8, RHelp, R26, R21B, R20, R19, R18, R13, R14, R5, R17, R16, D9, D8.

Take a break here and clean the board. Refer to point 9.

Q1, Q2, Q5, Q6, Q3, Q4, C4, C5, Chelp.

CCS Heatsink and Q11, Q13.

Q12, Q10, Q14, C3, D10, D11, C6, C7, Q7, Q8, D3, D12, Q9, C3, R11, R25, Clift, C2, TP0, C8, C0, Input Terminal block, C9, C1

Take a break here and clean the board once again. Refer to point 9.

8.1 Precision Output Stage Board

Suggested Component Installation Order for the EF3-3 & EF3-4

- Installing components left to right, top to bottom, and smallest to largest, can help ensure that you have sufficient access to allow your soldering iron room to reach the top of each component pad.
- Try and install components with their component values facing up or in a position where the component value can be read once all components are installed.
- You may want to use the suggested component installation order as a checklist, crossing off each component as you go.
- You may need to refer to other sections in the build guide when it's time to install one of the components listed below.
- The installation order shown below is for the EF3-4 Precision output stage board. It can still be used as a guide for the EF3-3.
    - EF3-3 specific components will be noted with an *.
    - EF3-4 specific components will be noted with an #.

Follow the following installation order.

Rled1, Rled2, D111, D112, R102, R103, R101, R104, D106A, D105A, R108, RCC, R105, R106, R107, R110, R129, D116, R128, D115, R127, *R127A, *R127B, R112, R113, R126A, TP104, R126B, TP101, #R120, #R126A/B, #R121, #R127A/B, R114, R120A/B, D103, R117, R111A, R123A/B, D104, R115, R111B, R121A, R121B, R118, R124A/B, R116, R122A/B, R119, R125A/B, TP102.

Take a break here and clean the board once again. Refer to point 9.

C112, Q101, Q102, D113, D114, D107, D108, C101, C105, C102, C106
C118, C119, *Q103 (refer to section 14.2 for further details).

Driver Heatsink instructions, for EF3-3 boards which are mounted parallel or perpendicular to the main heatsink and for EF3-4 boards which are mounted perpendicular to the main heatsink.

- Install Q107, Q108 as described in sections 6.5 & 6.iii. Once installed, trial fit the heatsink along with Q107, Q108 and installing the Mill-Max pins into their mounting locations. With the shoulder of the Mill-Max pins fully seated on the top of the PCB, solder the Mill-Max pins in place from the underside of the PCB.
- The driver heatsink should now slide on and off the Mill-Max pins and the legs of the transistors should be well aligned with the holes in the PCB. There should be no stress on the legs of the driver transistors.
- Remove the driver heatsink and solder the top of the Mill-Max pins to the PCB.
- Finally install the driver heatsink sliding it onto the Mill-Max pins while at the same time aligning and installing Q107, Q108.
- Check Q107, Q108 are positioned correctly, then solder them in place.

Now install the Pre-Driver Heatsink mounted a minimum of 3mm off the PCB to ensure ease of assembly and disassembly if required along with Q105, Q106 and the G2 ground connection.

Continue installing the remaining components following the suggested installation order.
C104, C109, C108, C103, C107, #C121, #C122, #C123, #C124, C113.

Take a break here and clean the board once again. Refer to point 9.

Next install the terminal mounts for PSU GND, V+, V-, SPK GND, and Output.
D109, D110, F1, F2, L1, C114, C116, C115, C117, TP103, TP100

Driver Heatsink instructions, for EF3-4 boards which are mounted parallel to the main heatsink.

- Install Q103, Q107, Q108
- Refer to section 14.2 for further details

Now position and install the SIP Header.

Don't forget to test your board here. Please follow the instructions shown under the heading "Setup for initial testing"

Once tested, install.

#Q115, #Q116, Q109, Q112, Q110, Q113, Q111, Q114.

9. Wash & Inspect the Boards

Now is a good time to clean the boards to remove the flux and inspect your work. Depending on the solder you're using, there can be a considerable amount of residual material that requires 2-3 scrubbings with alcohol and a brush to get it clean. It's best to give it a first cleaning before installing the taller, more delicate components, making the job much easier when it comes time to give it a final cleaning.

10. Install Trimpots

Note: When the adjustment screw is turned clockwise the resistance between pins 1 & 2 increases and the resistance between pins 2 & 3 decreases. Pin numbers are listed on the trimpot itself, on the datasheet and on the silkscreen on the underside of the PCB. The use of genuine high-quality trimpots is a must. Only use Bourns or Vishay for this application.

I. To enable the trimpots to be accessible and adjustable it's important to consider how you plan to mount each amplifier PCB in your chassis. Trimpots are available with their adjusting screws located on the side or on the top. Part numbers for each can be found in the BOM. PCB's mounted parallel to the heatsink generally requires trimpots with side adjustment screws, whereas perpendicular-mounted PCB's generally require trimpots with top adjustment screws.

II. Care must also be taken in the case where parallel mounting and rotating of the PCB is required. This option is generally used so the PSU input and the amplifier output terminals are located at the same end of the chassis. Generally, once again the trimpots will have their adjustment screws located on the side. Furthermore, to ensure they are still accessible and adjustable the trimpots themselves will also need to be rotated 180 degrees. This will then change the position of the trimpot pins 1 & 2 on the PCB and you will need to turn the adjustment screw anti-clockwise to increase the resistance between pins 2 & 3.

III. Using your ohmmeter to measure across pins 1 and 2 (*), adjust CCS1 trimmer R11 to its approximate midpoint of 50 Ohms before installing it. This is a good starting point for the initial power-up.

IV. Measuring across pins 1 and 2 (*), adjust the trimpot (R25) for the DC offset to 2.2k Ohms its approximate midpoint before installing it.

V. Adjust bias trimmer R109 to its maximum value of 500 Ohms prior to installation. This measurement is taken between pins 1 and 2 (*). This is an important adjustment, as getting it backward will apply maximum bias upon power-up. The pins are clearly marked on the board and the trimmer.

(*) Assuming that pins 1 & 2 on the trimpot match pins 1 & 2 on the PCB.

11. Install Capacitors

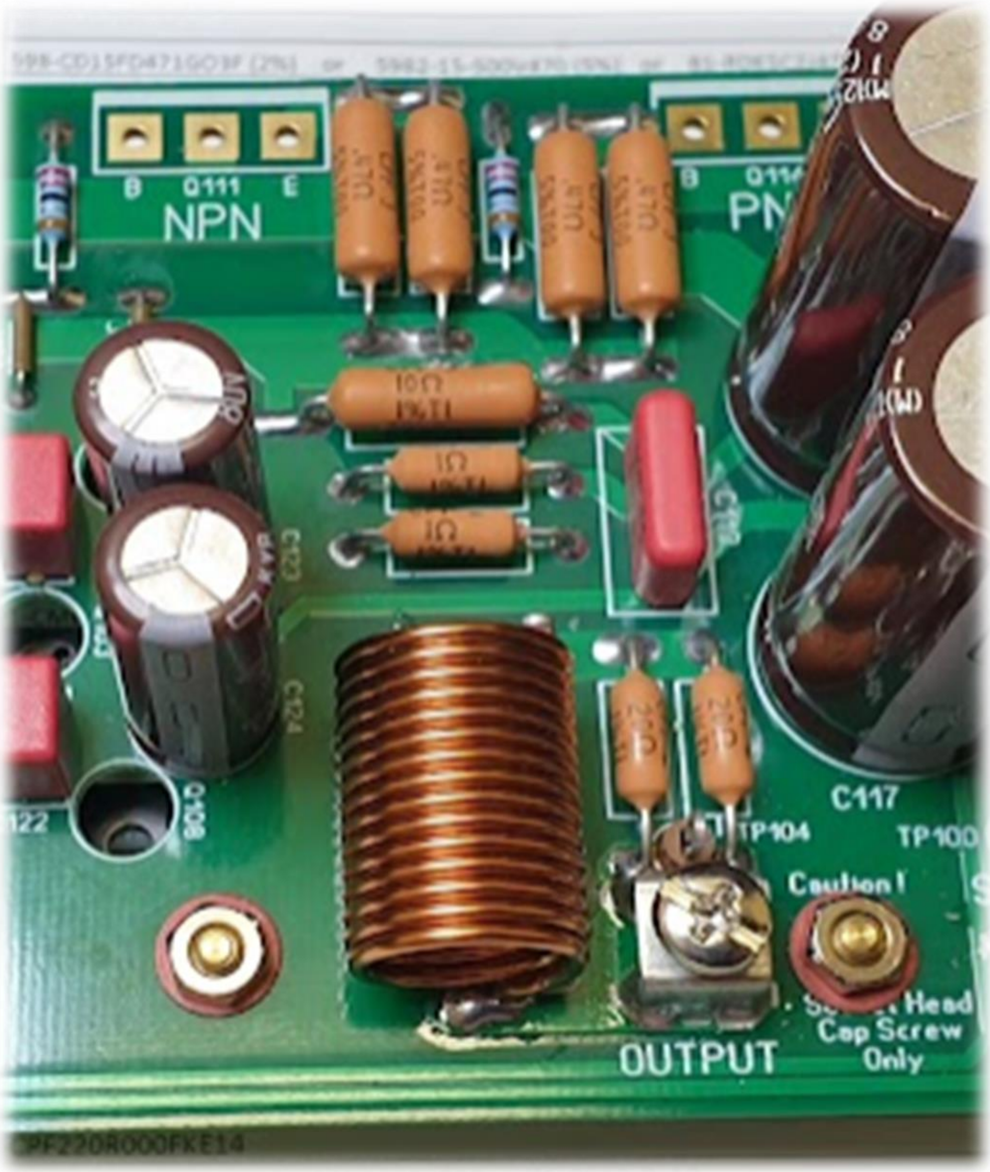
I. Following the BOM, install the remaining capacitors starting with the smaller-bodied devices like the Wima polys and dipped micas.

II. The electrolytic caps are polarized (with the exception of C9). The negative terminal is clearly marked on the body of the device and the board is marked for proper orientation

III. The recommended compensation capacitors are C4 = 100pF and C5 = 470pF as they should work with all transistor combinations. Feel free to try to lower compensation capacitors as some Output and Driver Transistors combinations will allow for this, but be prepared to check for oscillations using an oscilloscope to verify stability.

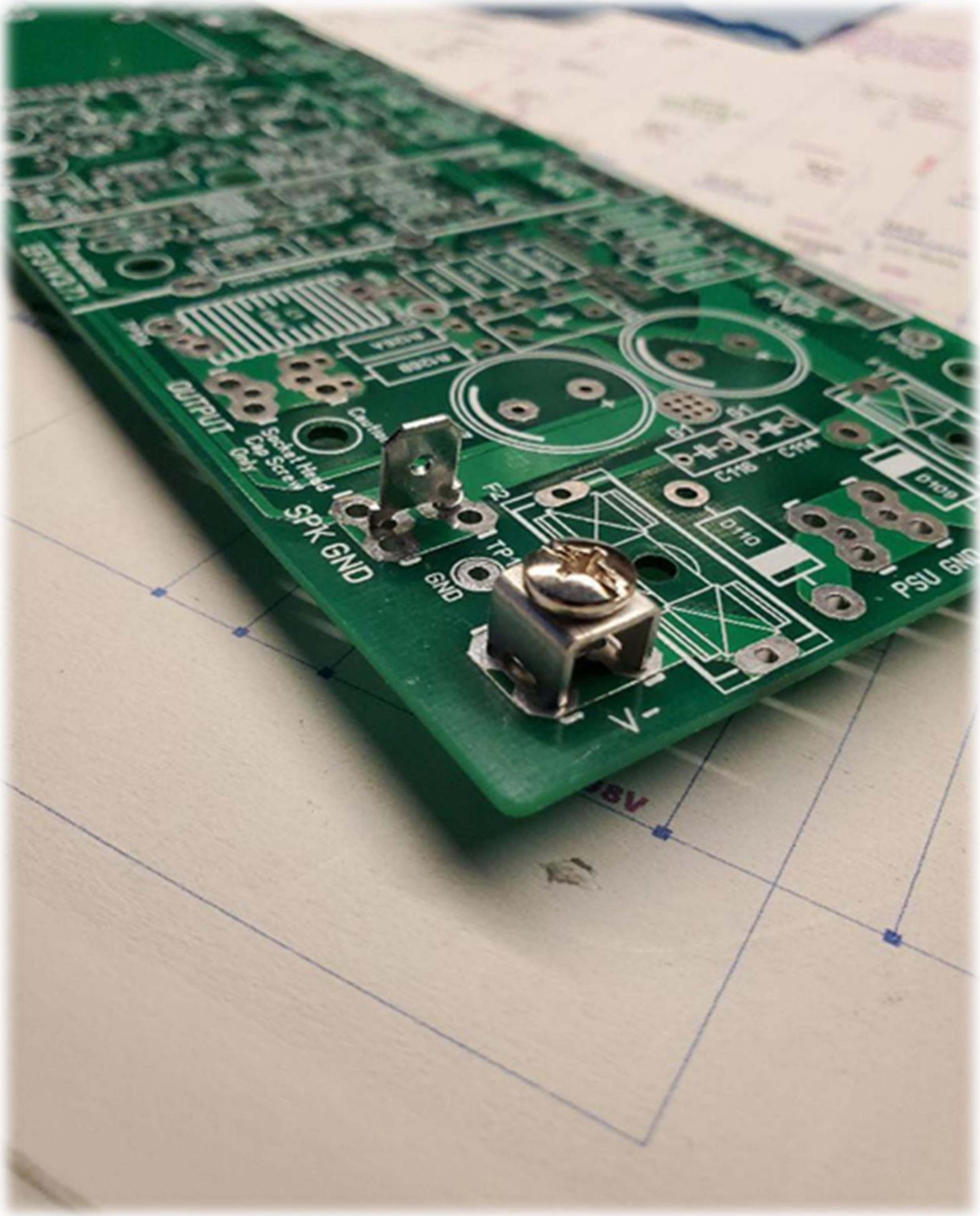
12. Create & Install Output Coil L1

Output coil L1 is made by winding 19 (clockwise) turns of 18ga. enameled copper wire on a 10mm diameter former. This will result in a coil 18-19mm long fitting the board perfectly. The finished value should be ~1.5 μH. An alternative to 18ga. wire for higher currents would be to wind 14 turns of 16ga. wire on a 12mm former. This will yield 1.5 μH, as well.



13. Installation of Input, Output, and Power Rail Attachment Hardware

The options for the high-power attachments are screw connectors or blade-style Faston connectors. The screw connections with wire ring terminals are the recommended choice.

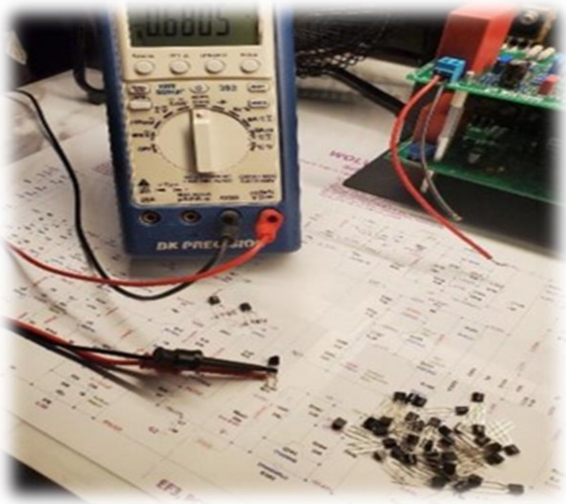


14. Installing Transistors

I. Long-tailed pair Q1/Q2 and current mirrors Q3/Q4 should be matched in performance as well as one can within practical means. Matching the Vbe and hfe is desired, but not often achievable by many. At the very least, use a digital voltmeter (DMM) in the diode test mode to measure the forward voltage across the base and emitter (Vbe). This requires a multimeter capable of measuring millivolts to at least 3 digits to be effective.

II. While measuring, don’t touch the transistor with your fingers, as it’s very sensitive to temperature. Let it sit until the voltage (.7XXv) stabilizes, which could take a couple of minutes, then tag the device with the value. Measuring several devices will eventually yield which ones are best matched for Vbe.

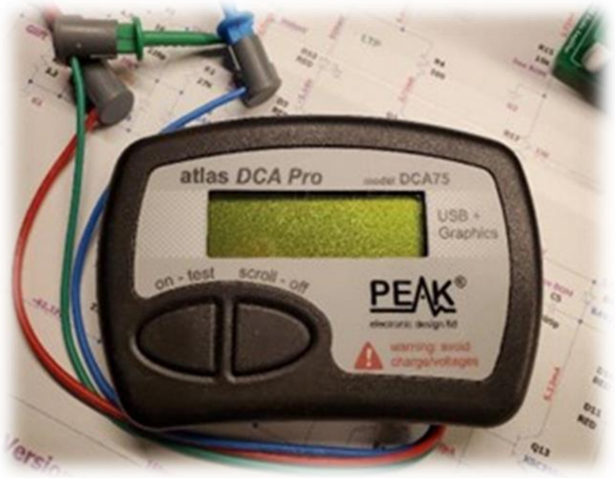
III. An alternative to the above, using a test device such as a Peak DCA75 will provide a reasonable estimate of performance for the purpose of matching Vbe and gain (hfe). If you have several to select from, measure each one and match them by Vbe first, then select the devices having the closest value for hfe.



14.1 Basic Transistor Matching

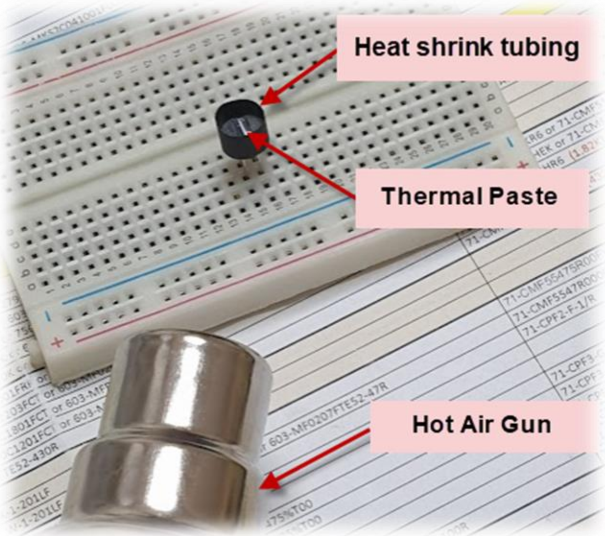
Use a voltmeter capable of at least 3 decimal points to get meaningful readings. Clip the leads to the base and emitter and watch the voltage. If you don’t get a reading, reverse the leads. Don’t touch the transistor while allowing it to stabilize. The reading will increase as it stabilizes.

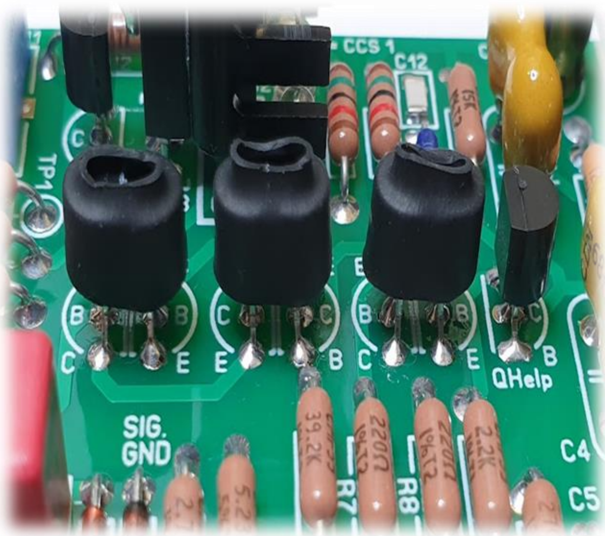
Measuring Vbe by using the diode test mode on a voltmeter is the very least that you should do to match the small signal transistors.



There are several transistor testing devices on the market capable of providing some degree of measurement. When it comes to assessing the performance of a transistor, none of these basic test devices are perfect. But, they're better than doing nothing when it comes to trying to match devices.

IV. Thermally coupling the paired transistors is necessary for them to operate on an equal basis during changes in temperature. After pairing transistors Q1 and Q2; Q3 and Q4; Q5and Q6, bind them together using heat shrink tubing or other means. A small smear of thermal paste between the transistors will help with coupling.





V. Install Q9, Q10, Q12 and Qhelp.

VI. Install Q101 and Q102.

Important: Note that the pinouts are reversed relative to each other on the board! Observe the "base" marking on the board.

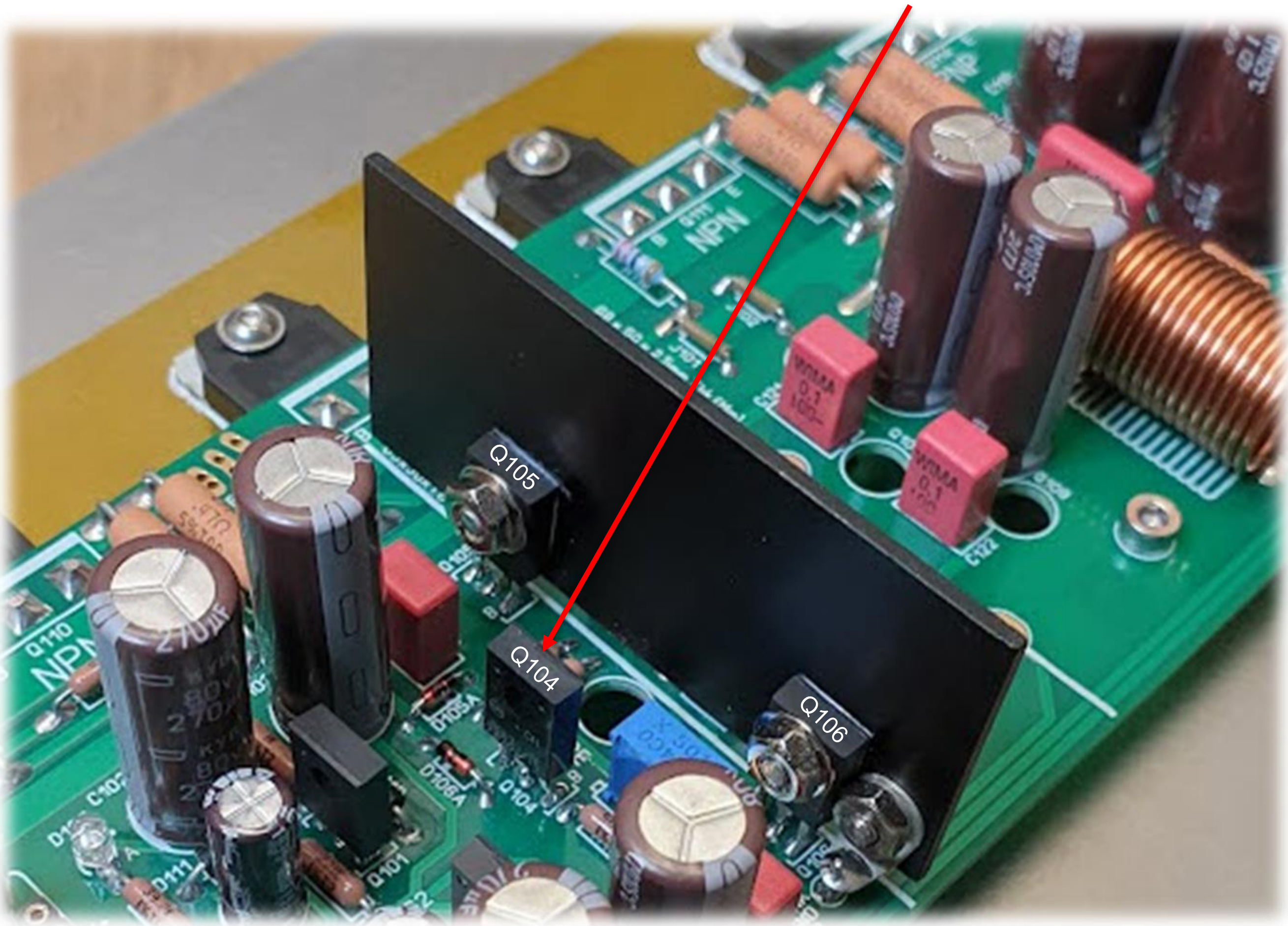
14.2 Installing Q103 and Q104.

Q103 and Q104 have now changed roles for improved thermal performance.

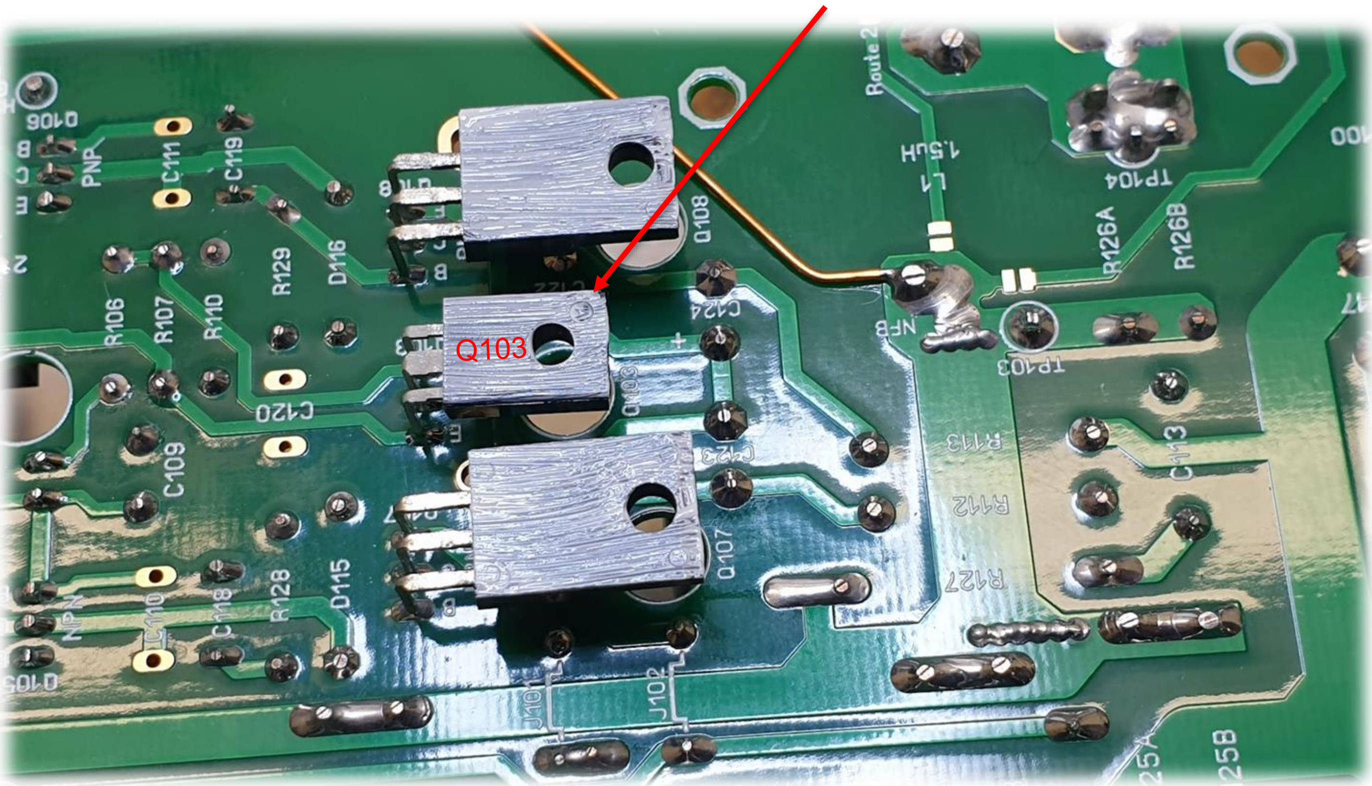
Q104 is now the ambient temperature sensor mounted in free air in its location on the PCB.

Q103 is now the heatsink temperature sensor and is mounted on the main heatsink between the driver transistors, or on an output transistor itself.

Please see below picture for Q104 ambient temperature sensor transistor location.



Please see below for the Q103 optimal mounting location. For the EF3-4



I. Q103 can be installed in a couple of ways depending on how you intend to mount the output transistors to the output heatsink… parallel or perpendicular. However, Q103 must be mounted in a manner that thermally couples it to the main heatsink.

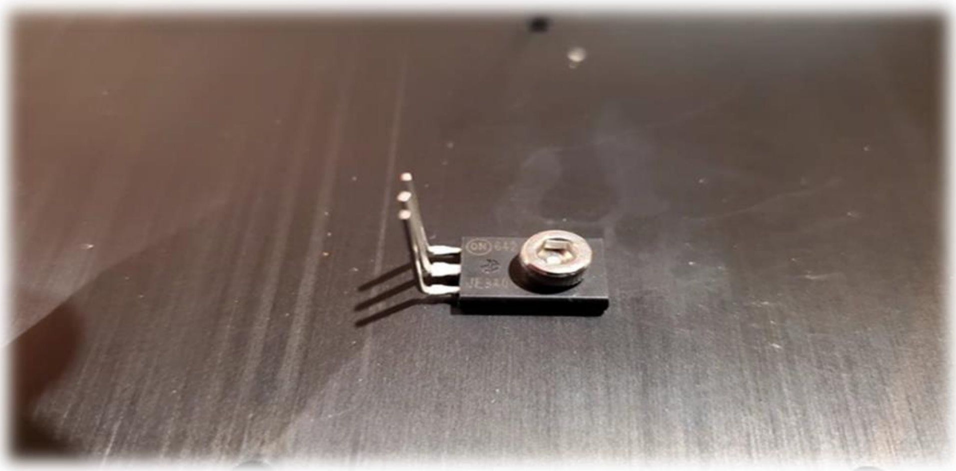
II. When opting for parallel mounting, the Precision EF3-4 board allows Q103 to be mounted directly to the main heatsink underneath the EF3-4 board, alternatively for the EF3-3 or EF3-4 you can offset it to the side on a short lead, on or between the output transistors. However, for the EF3-4 mounting Q103 between the driver transistors on the main heatsink is the recommended option.

III. For the EF3-4 boards, if you plan on mounting the output transistors parallel to the circuit board/heatsink, trial fit the board by installing standoffs where the mounting screw locations are for the circuit board on the UMS heatsink. If you are not using a heatsink that is drilled and tapped to UMS standards, you must locate, drill and tap the holes yourself. The EF3-4 board will work fine as a template for the hole locations.

IV. Locate the screw hole on the UMS heatsink where the large hole near the middle of the circuit board lines up. Trial fit Q103 by lightly holding it with a screw. Do not screw it down tight yet, it should be able to move left and right a little bit. You may need to drill and tap this hole if it’s not present in your heatsink. (No Insulator is required transistor as they have insulated backs)

V. Bend Q103 leads straight up so they are 90° to the heatsink. The bend point is where the lead goes from narrow to wide. Install the circuit board on the standoffs, carefully guiding Q103's leads through the through holes for that transistor.

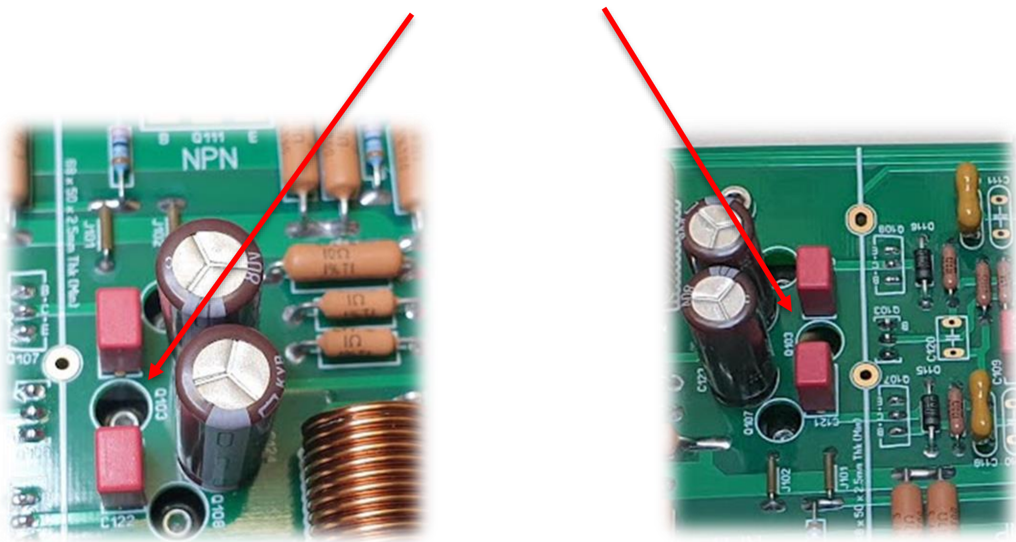
Important: If mounting the boards/outputs parallel to the heatsink, the recommended minimum height of the standoffs is 8mm. A height of 10mm is considered the maximum.



VI. For mounting perpendicular to the heatsink, attaching Q103 directly to one of the output transistors is the most efficient method to achieve the best thermal coupling. This is done by attaching the transistor using “flying wires”, as shown in the following photo. Solder three wires to the circuit board and route them to Q103, being mindful of proper pinouts. Do not make the wires excessively long as they just need to be long enough to mount the transistor to its position on the heatsink. Use heat shrink around the wire/transistor connections to reinforce the solder connections.

VII. Note that Q103 can be mounted in a variety of different ways, but must be mounted directly to an output transistor or to the heatsink between the driver transistors. This view shows the EF3-3 board with Q103 mounted using fly wires. Fly wires enable the transistor to be located where most effective.

As an alternative to the above, the board was designed with a provision to access the screw to mount Q103 directly to the heatsink underneath the board.



15. Install the Pre-Assembled Transistor/Heatsink Combinations

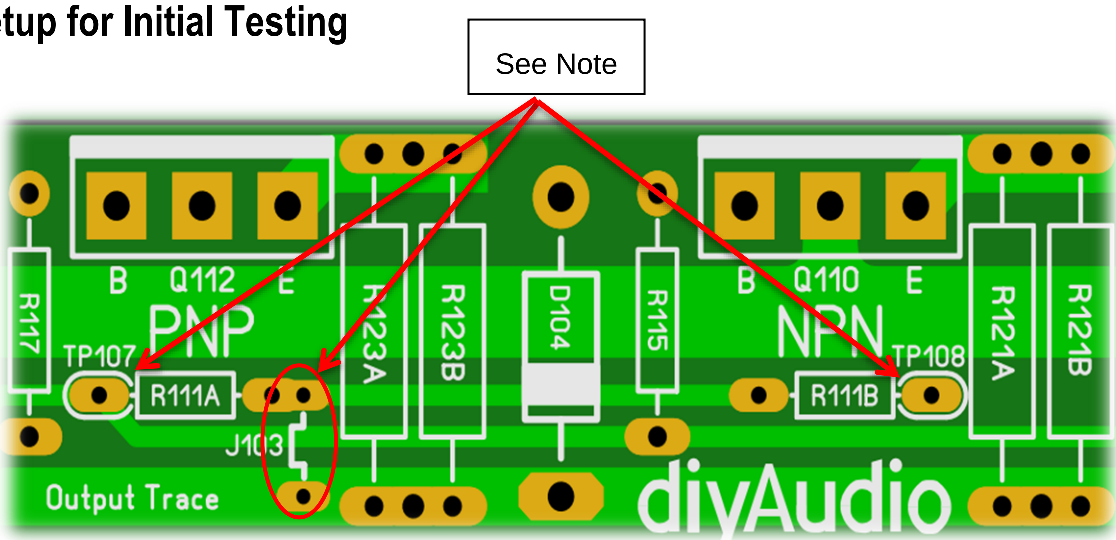
I. For the EF3-3 output board.
   i. We recommend installing the driver heatsink assembly first, in order to provide a reference for elevation. As an example, the pre-driver heatsink as seen in the mockup photos is 3mm above the surface of the board.

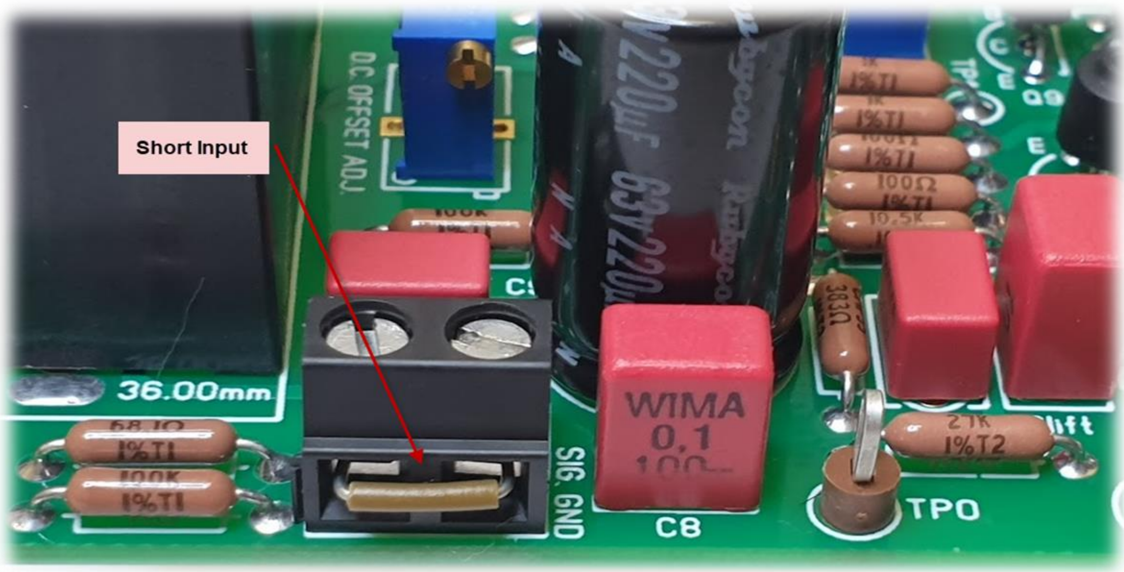
II. Fit each heatsink assembly in place while double-checking for the correct device and proper pin orientation. Adjust for the preferred height and tack solder one pin on each transistor to hold it in place, in order to confirm that the heatsink is at the preferred elevation and level relative to the board.

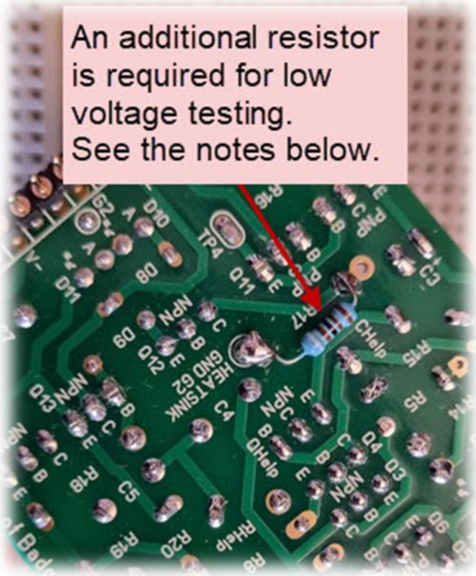
III. Install SIP1 pin headers. Assemble the IPS board with the EF3-X board using the spacers/hardware to ensure that everything is properly aligned before soldering both sides of the pin header.

IV. With the exception of the output transistors, the board is now complete. Once again, wash the board and inspect your work.

15.1 Setup for Initial Testing







*Note **

- Temporary jumper wire J-103 is installed only for the purpose of performing initial testing without the output transistors installed.
- Use a small piece of wire to short out the input until it’s time to put a signal in.
- Use Test Hook Clips to make your connections between R111A (TP107) (DMM +) and R111B (TP108) (DMM -), and set up your voltmeter to monitor the bias voltage (millivolts) between R111A (TP107) and R111B (TP108) at the location shown in the image above.
- If you are preforming your initial test with a low voltage 30v dual rail power supply it is necessary to place temporary 9.1K resistor in parallel with R17. This must be removed before returning before returning to a full rail voltage test.

16. Initial Testing Without the Output Transistors Installed

The goal of this procedure is to enable the builder to power up under restricted conditions, in order to take essential measurements and check functionality. Safety precautions must be exercised in all aspects of this work. If you are not comfortable with this step, don't be shy about submitting your questions/concerns to the Wolverine forum. There are many who will provide answers and guidance.

Our testing revealed that a dual rail power supply capable of supplying at least 30Vdc at 0.3 Amps will suffice for this procedure. An excellent choice for this is a bench power supply that can limit current to 0.3 Amps. If you don't have an adjustable bench supply or Variac to slowly bring the voltage up, a simple current-limiting incandescent lamp will work fine for the initial power-up. This and the use of conservative fuses will provide protection for the initial test.

16.1 Measurements Notes

I. A floating DMM is required when taking differential nonground referenced voltage measurements.
II. The use of insulated Hooks or Parrot clips for test point connections is recommended.
III. It is recommended to use a handheld DMM for taking voltage measurements as they are floating and not ground-referenced.
IV. If a Bench DMM is to be used ensure it's able to take floating not ground-referenced voltage measurements.
V. Keep your measurement leads short and twist them if you can, to minimize any pickup and noise injection into the measurement points.
VI. Were practical, ensure your measurement leads are not running over the top of other components and try to keep them neat and off the surface of the PCB.
VII. If you are preforming your initial test with a low voltage 30v dual rail power supply it is necessary to place temporary 9.1K resistor in parallel with R17. This must be removed before returning before returning to a full rail voltage test.

16.2 Setup

I. Verify that the J-103 jumper wire is installed. This jumper will essentially have the drivers functioning as the outputs for the purpose of testing.
II. Verify that DC offset pot R25 is set to its midpoint of ~2.5k Ohm as measured between pins 1 and 2.
III. Verify that the R11 CCS1 pot is set to its midpoint of ~50 Ohm as measured between pins 1 and 2.
IV. Verify that R109 is set to its maximum resistance of ~500 Ohm as measured between pins 1 and 2.
V. Install 0.5 Amp fuses in the board. These will be adequate for the initial functional tests. If you blow these, something is wrong, and you must stop and review your work.
VI. Install a short piece of wire to short out the input.
VII. Do not apply a load to the output.
VIII. Use Test Hook Clips to make your connections, and set up your voltmeter to monitor the bias voltage (millivolts) across R111A (TP107) and R111B (TP108) as shown in

the image above.

16.3 Taking actual measurments

For the following DMM measurements (noting their polarity), the board is not connected to the power supply and the output is not connected to any load:

I. Verify that measuring from V+ (DMM +) to Ground (DMM -) (TP-100) is greater than 10k Ohms.
II. Verify that measuring from V- (DMM -) to Ground (DMM +) (TP-100) is greater than 10k Ohms.
III. Verify that measuring from Output (DMM +) (TP104) to Ground (DMM -) (TP-100) is greater than 10k Ohms.
IV. Verify that measuring from V+ (DMM +) to Output (DMM -) (TP-104) is greater than 10k Ohms.
V. Verify that measuring from V- (DMM -) to Output (DMM +) (TP-104) is greater than 10k Ohms.

Note: Q103 does not have to be mounted on the heatsink at this point of testing

17. Initial Power Up

Initial Power up three things to observe for “normal” behavior.

A. The voltmeter monitoring the output voltage (millivolts) across R111A (TP107) and R111B (TP108) should immediately read between 750 mV to 800 mV and not spike any higher. If the voltage immediately spikes, shut it down. It’s either due to R109 not being adjusted to its maximum resistance of 500 Ohm or something else is seriously wrong. This must be investigated before you do anything else.

B. Immediate lighting of all 6 LED’s. All the LED’s should light in a uniform manner. However, if you’re using a power supply of less than 35 V, be aware that D10 and D11 may not light, as they likely do not have sufficient voltage to turn on. This is not a concern. If the rest of the LED’s do not light, immediately shut down and investigate. Check to make sure that the LED’s are installed with the correct polarity. This can be done using your DMM in the diode test mode. If the LED’s are installed correctly, go back over your work to check for errors.

C. If the 0.5 Amp fuses blow under the initial power-up conditions, something is seriously wrong, and you must stop and check your work for errors.

D. If using a current limiting light, you should expect it to initially light and quickly start to dim as the capacitor bank in your PSU charge. If it remains bright, shut down and investigate, as something is wrong causing the system to draw too much current.

17.1 Measurements & Settings (If “normal” are determined above)

E. Using a second voltmeter (continue to use a voltmeter to monitor the bias), carefully measure the voltage between TP-1 (DMM +) and TP-2 (DMM +) while adjusting R11 to achieve 5.0 V. This establishes the proper 5 mA constant current (CCS1) for the long tail pair (LTP).

F. Measure the voltage between TP-3 (DMM +) and TP-4 (DMM -). This is to verify that proper current is established for CCS2 and should measure between 600 mV and 630 mV.

G. Measure the voltage between TP-105 (DMM -) and the positive power rail connection. Place the negative probe on TP-105 and the positive probe on the positive rail. It should read about 1.5 V if you are using the TTC004B transistor for Q101. It should read about 1.7 V if you are using the KSC3503 transistor for Q101. Similarly, place the positive probe on TP-106 (DMM +) and the negative probe on the negative rail. It should read the same 1.5 V or 1.7 V depending on whether you are using the TTA004B or the KSA1381 for Q102. This means that Q101 and Q102 are regulating properly.

H.

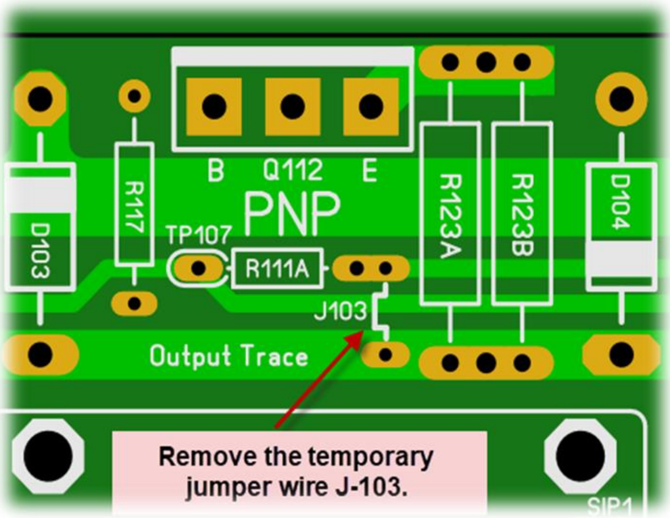
i. Measure the voltage across the output, TP-100 (DMM +) and SPK GND (DMM -), TP-104. While measuring, adjust R25 to achieve a measurement of about 0 V, which is the DC offset. This measurement is highly sensitive and will have to be done many times as adjustments are made and testing progresses.

ii. Measure the AC voltage in millivolts across TP104 (output) and TP100 (Speaker ground) ideally this should be less than 0.080 mV. If this value is greater than 1 mV and depending on the sensitivity of your speakers you may hear a small amount of hiss coming from your speaker's tweaker.

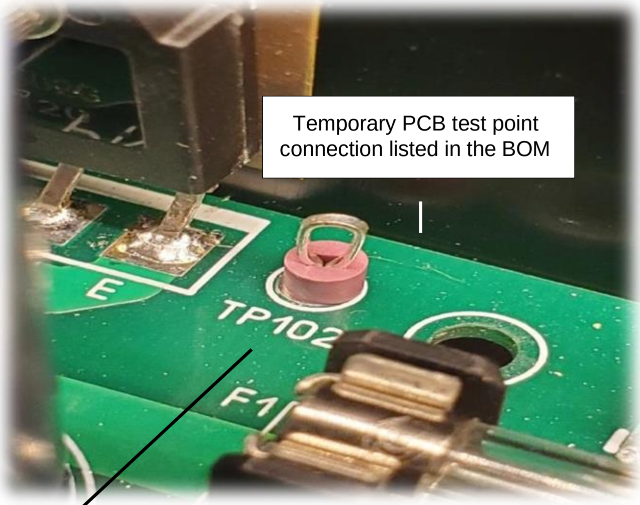
I. Verify that the output bias circuit functions properly by adjusting R109. Take note of the direction you're turning the adjustment screw and go slow. After a few turns you should see the bias increasing. If not, try the other direction. Make small adjustments until the bias voltage across R111A and R111B reads between 1.4 V and 1.45 V and let it sit for a while to stabilize. The bias is very sensitive to temperature and will tend to drift… This is normal. (Remember your output transistors are **not** installed in this step).

J. Upon the completion of the above steps, **adjust R109 to reduce the bias back to its minimum voltage setting (maximum resistance) before shutting down.**

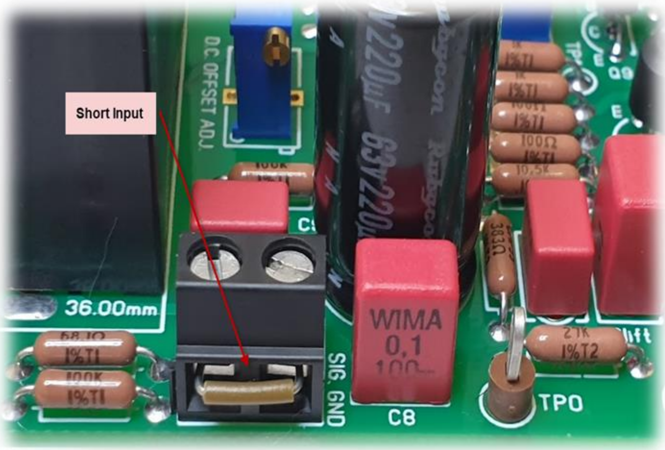
18. Install the Output Transistors



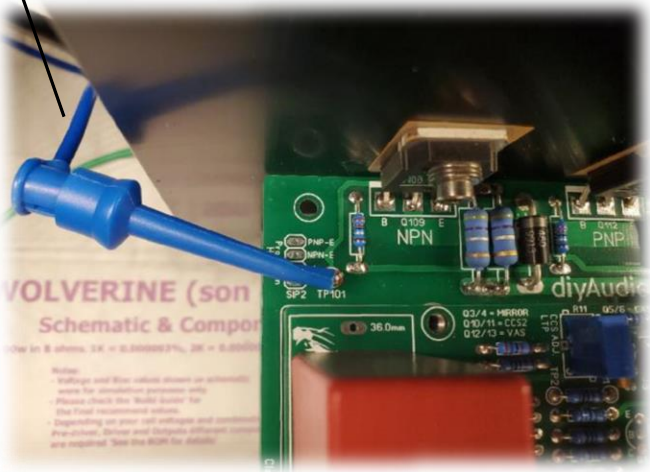
Remove the temporary jumper wire J-103.



The use of Test Clips to attach your meter leads is highly recommended to keep your test connections secure and to reduce the possibility of shorts.



Use a short piece of wire to short out the input until it’stime to put a signal in.



B. Remove temporary jumper wire J-103

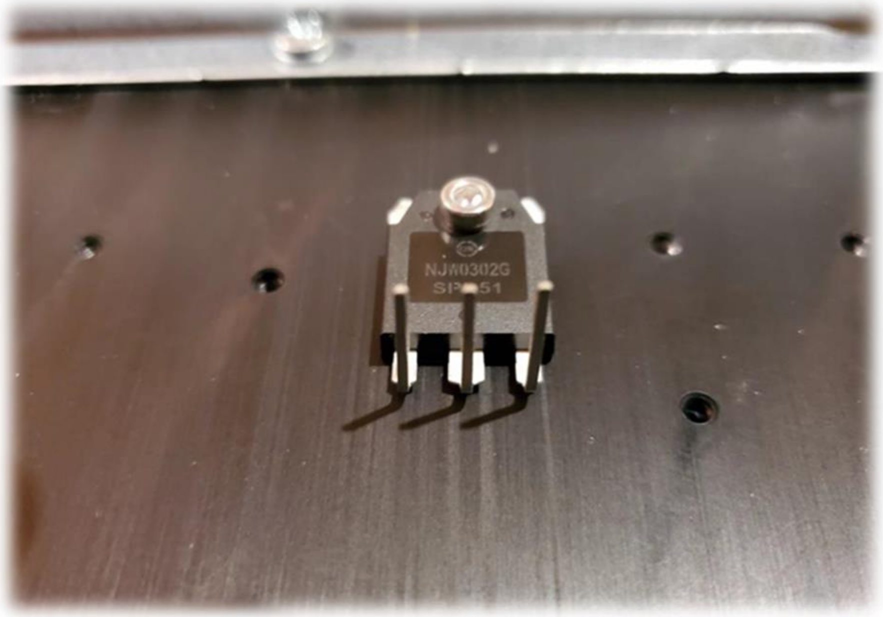
C. Install the output transistors. Make sure that proper insulation pads or Kapton tape is used and verify electrical isolation by doing a continuity check when complete. To mount the output transistors parallel to the circuit board, trial fitting is necessary to properly line up the components before soldering is done.

D. Bend all the output transistor leads straight up so they are 90° to the heatsink. The bend point is where the lead goes from narrow to wide. Support the transistor leads with long-nose pliers between the bend and the transistor body to avoid stress on the transistor body. This should be done for all transistor lead bends.

E. Temporarily screw down the output transistors with no Insulator pads at this time. Do not screw them down tight, as they should be able to move left and right a little bit.

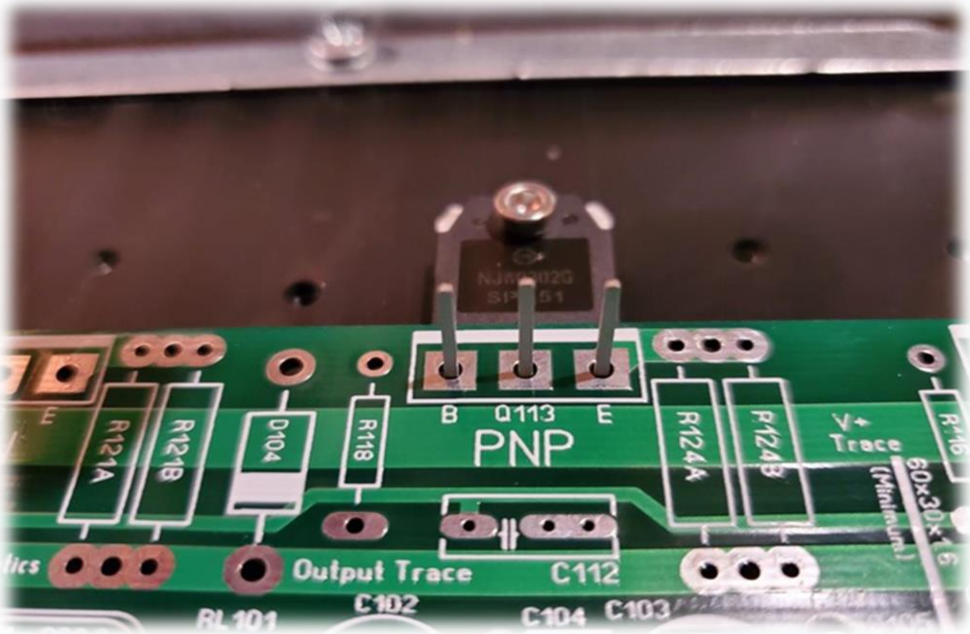
F. For vertical board mounting the output transistor leads are left straight, however, it is vital to run a strip of Kapton tape below the output transistors where the PCB edge may contact the heatsink to avoid accidental electrical contact between the heatsink and the board or components when fully assembled.

G. Bend all the output transistor leads straight up so they are 90° to the heatsink. The bend point is where the lead goes from narrow to wide.



H. Install the circuit board standoffs on the heatsink, if you haven't done so already, and install the circuit board on the standoffs. Carefully guide the output transistor leads through the solder pad holes.

I. Lightly tighten the screws for the circuit board followed by tightening the screws on the output transistors (Just snug enough to hold their positions). Solder the outputs to the circuit board. This ensures there is no mechanical stress on the transistors after the final assembly.



J. Unscrew the circuit board and output transistors and inspect your work. Add solder as necessary on the bottom side of the board to ensure a solid connection to the output transistors.

K. Clean the surface of the heatsink with some alcohol. This is particularly when using Kapton/Polyimide tape.

L. Install Insulator Pads for the Output transistors and Q103. Use either 1mil thick Kapton/Polyimide tape with 1.5mil silicone adhesive or Mica insulators. With Kapton/Polyimide tape, apply it directly to the heatsink, work out all air bubbles, and poke a hole where the screw holes are. Use thermal grease between the tape and transistors. With Mica, use thermal grease on both sides of the insulators. Sil pads will work too, but they do not work as well thermally.

https://www.newark.com/mcm/21-1170/transistor-insulator-to-3p-mica/dp/02J5529

19. Install the Circuit Board & Output Transistors

Install the circuit board and output transistors by screwing them down. It helps to use a split, spring, or wave lock washer (Belleville) and an oversized diameter flat washer to distribute the pressure more evenly between the screw head and the Output transistors. The Belleville and flat washers are available from the DIYA store.

https://www.mcmaster.com/standard-washers/

A. Mount Q103 on the heatsink or output device. If mounting on the heatsink use appropriate insulating pads and thermal grease and secure with the Belleville and flat washers.

B. For the following DMM measurements (noting their polarity), the board is not connected to the power supply and the output is not connected to any load:
    i. Verify that measuring from V+ (DMM +) to Ground (DMM -) (TP-100) is greater than 10k Ohms.
    ii. Verify that measuring from V- (DMM -) to Ground (DMM +) (TP-100) is greater than 10k Ohms.
    iii. Verify that measuring from Output (DMM +) (TP104) to Ground (DMM-) (TP-100) is greater than 10k Ohms.
    iv. Verify that measuring from V+ (DMM +) to Output (DMM -) (TP-104) is greater than 10k Ohms.
    v. Verify that measuring from V- (DMM -) to Output (DMM +) (TP-104) is greater than 10k Ohms.

20. Powering Up with Output Transistors Installed

A. Install 2 Amp fast blow fuses at F1 & F2.

B. Verify that the input terminal is still shorted out.

C. Verify that there is no load hooked up to the output terminal.

D. Connect your power supply to the board rail connections V+, V- & PSU GND. If you have a current limiter light or a Variac to adjust the voltage to the power supply, now is the time to use it.

E. As done in the initial power-up procedure, have a voltmeter connected to TP-101 (DMM +) and TP-102 (DMM -) to monitor the output bias.

F. Power up… As before, you should expect the LEDs to uniformly light and the bias voltage to measure only a few millivolts, if at all. If you observe the bias at a few millivolts and very slowly increasing, this would be normal. However, the bias voltage should stop increasing before reaching 20 mV. If it does not, shut down and re-check the R109 trimpot setting as described in step 14D. **If you are using a current limiting bulb, it should not be lit.**

G. With power applied to the amplifier and everything appearing normal, measure across TP-1 (DMM +) and TP-2 (DMM -) and adjust R11 for 5 V. This establishes the proper 5 mA constant current (CCS1) for the long tail pair (LTP).

H.

i. Measure the DC offset **across TP104 (output) (DMM +) and TP100 (Speaker ground) (DMM -)** and adjust R25 to achieve as close to 0 mV as practical.

ii. Measure the AC voltage in millivolts **across TP104 (output) and TP100 (Speaker ground)** ideally this should be less than 0.080 mV. If this value is greater than 1 mV and depending on the sensitivity of your speakers you may hear a small amount of hiss coming from your speaker's tweaker.

J. Adjust R109 while measuring **across TP101 and TP102 to read ~40 mV** of output transistor bias. If you are using a current limiting bulb it may begin to light, **this is normal**. The measurement will have a tendency to drift and it will take several minutes for it to stabilize. Make very small adjustments and give it time to respond. At this point, it's time to let it thoroughly warm up and "soak". When done, reduce the bias back to a **minimum voltage** condition before shutting down. **If you are using a current limiting bulb, it should *not* be lit**. Adjusting the steps in 18.H & 18.I is an iterative process and may take a few adjustments between each to reach the final values shown.

K. Upon the completion of the previous steps, the amplifier is ready for full power and bias adjustment. **Remove current limiting devices** & install fuses as per sheet 2 of the BOM.

L. Once again, with the input shorted and no load connected to the output. Power on and monitor the bias. Adjust R109 to achieve 44 mV bias between TP-101 (DMM +) and TP-102 (DMM -) at the locations shown in the images at point 16, until stable. Recheck the DC offset at the output and adjust as necessary.

21. Burn-In

If the amplifier measurements remain stable for 30 minutes or more, it's time to put a signal in it! If you don't have an oscilloscope and signal generator to look at the signal, connect it to a preamp and put some music through it!

Since this amp doesn't have short circuit protection, always make connections to a speaker or dummy load with the power off.

A. Connect the amp to an old speaker or an 8 Ohm dummy load. Do not run continuously at high power with a function generator without putting a fan on the heatsink. When listening to music, high peak power is not continuous so you should not need a fan if your heatsink is adequately sized.

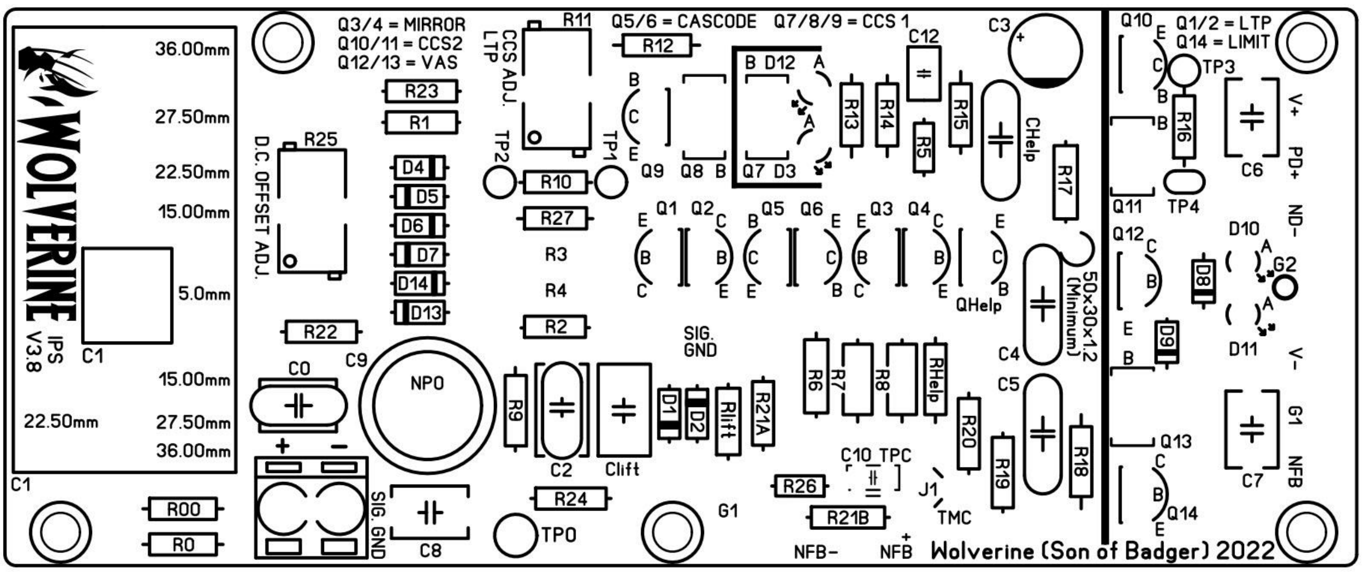
B. After running for a while, turn off the input signal and note the bias between TP-101 (DMM +) and TP-102 (DMM -). It should steadily drop back down to ~40 mV to 44 mV, reflecting the temperature of the heatsink. Without turning the amp off, allow it to cool down and return to its steady state temperature. This may take 30 min or more.

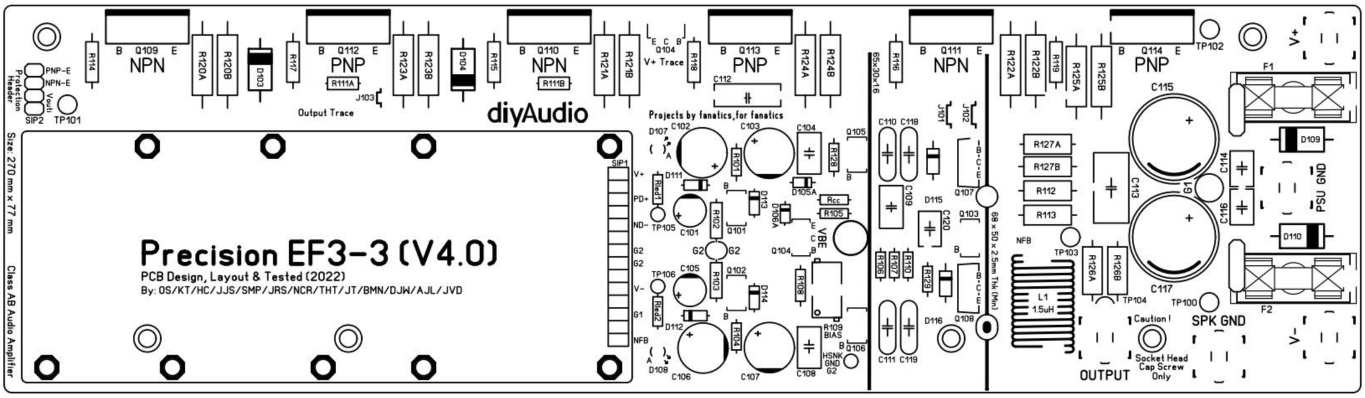
C. Recheck the Output Transistor Bias between TP-101 (DMM +) and TP-102 (DMM -) and adjust R109 as necessary.

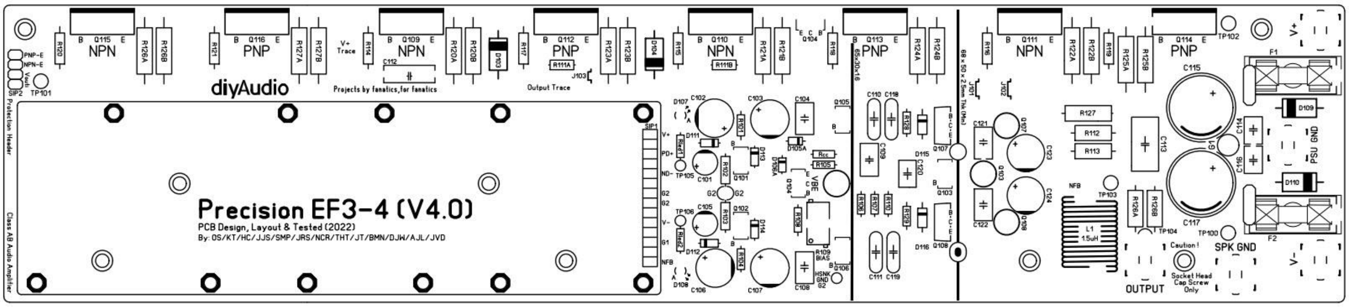
D. Recheck the DC Offset at the Output (TP-104) (DMM +) to Ground (TP-100) (DMM -) and adjust R25 as necessary.

E. If you had to make any adjustments, it does not hurt to burn it in again.

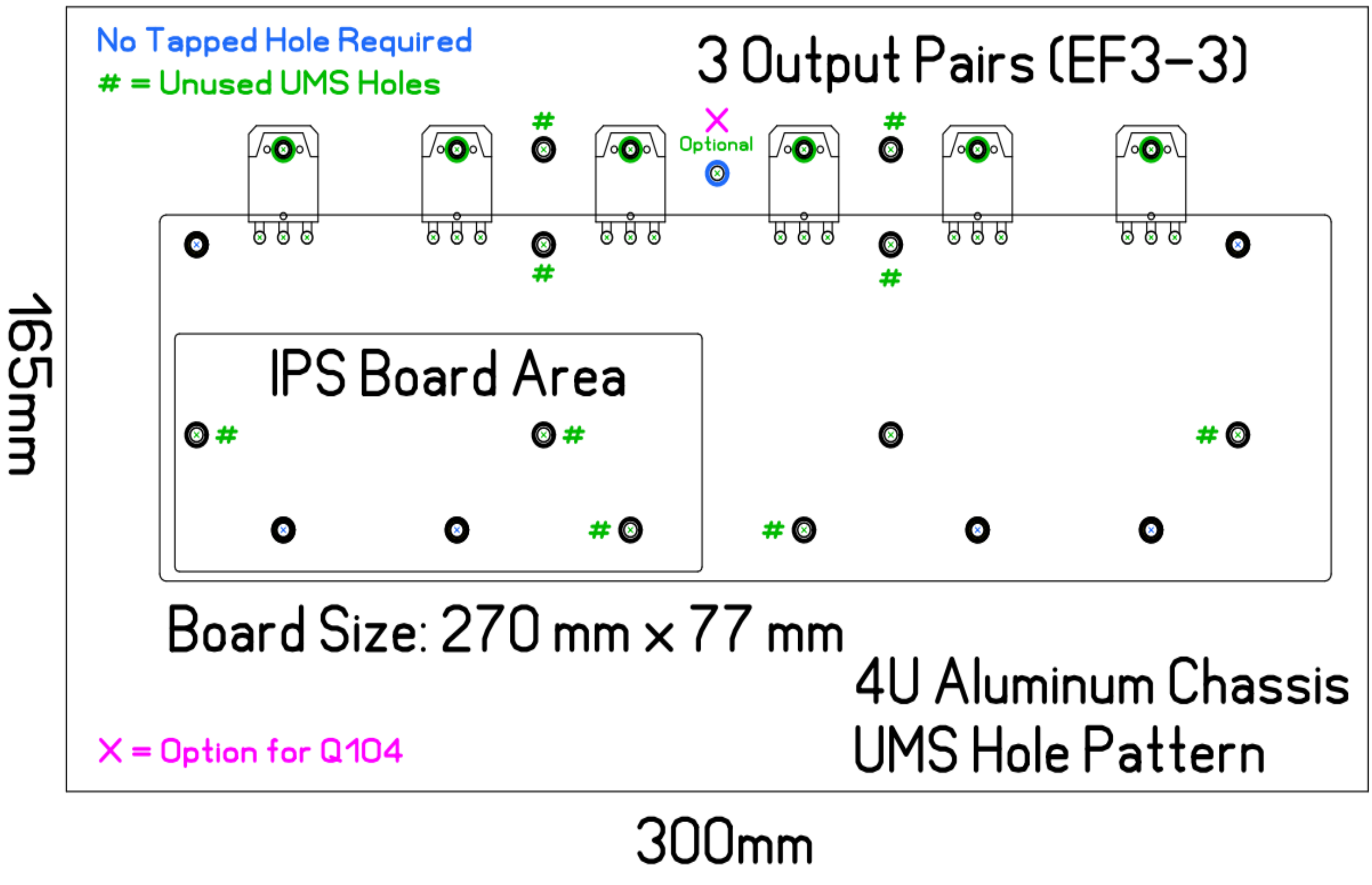
21.1 For reference, the component layout silk screen (showing the 2nd group buy boards)

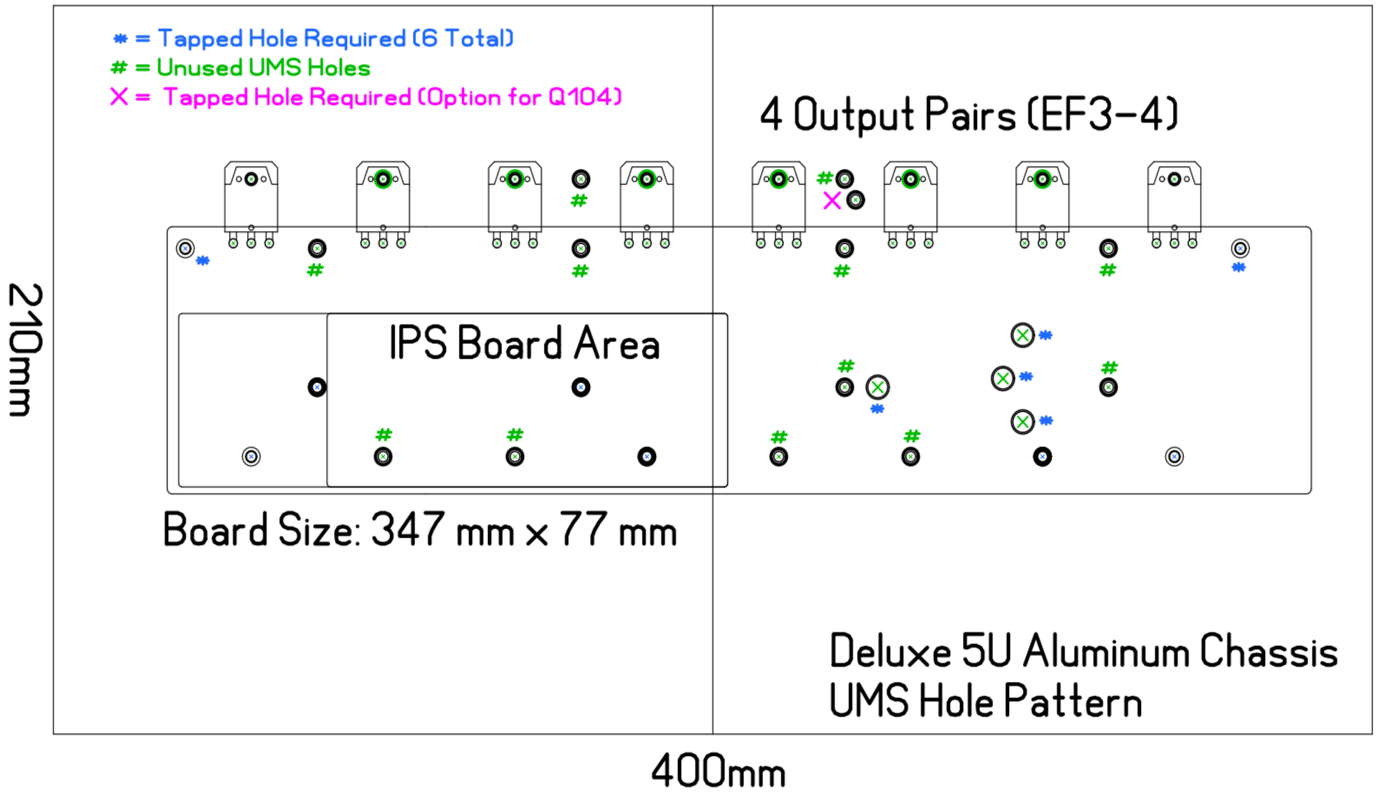




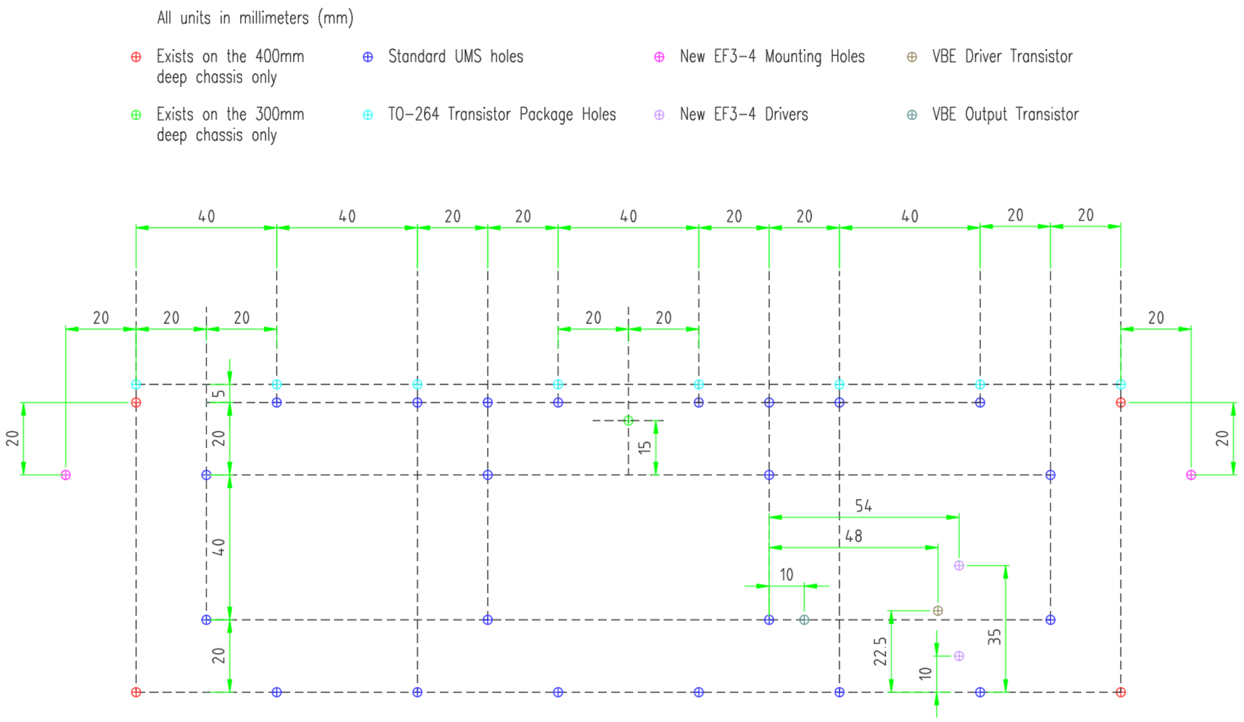


21.2 For reference, EF3-X Boards on Main Heatsink



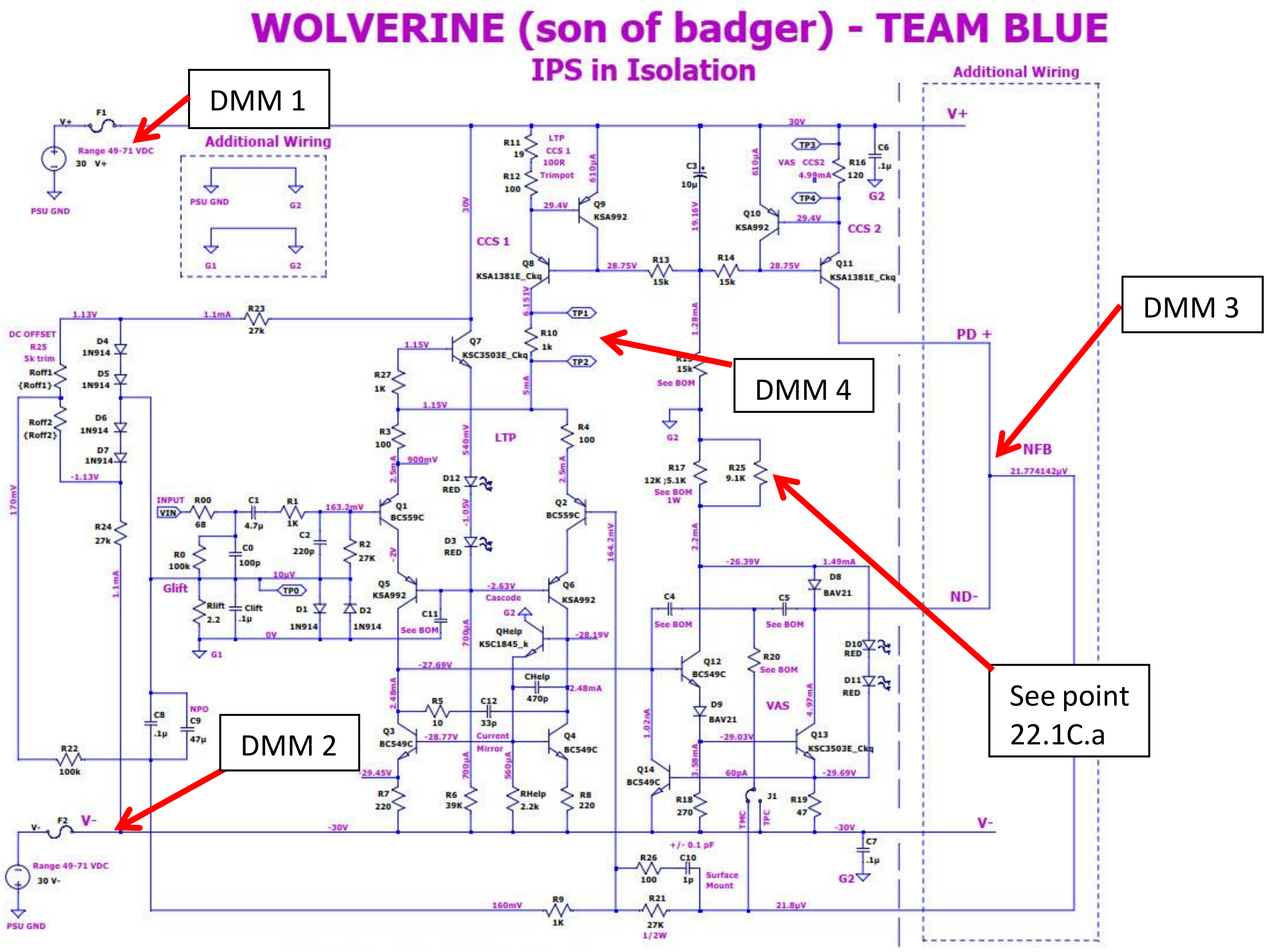


21.3 For Reference, Heatsink Hole Locations



22. Troubleshooting & Testing in Isolation

Wolverine IPS Board schematic:



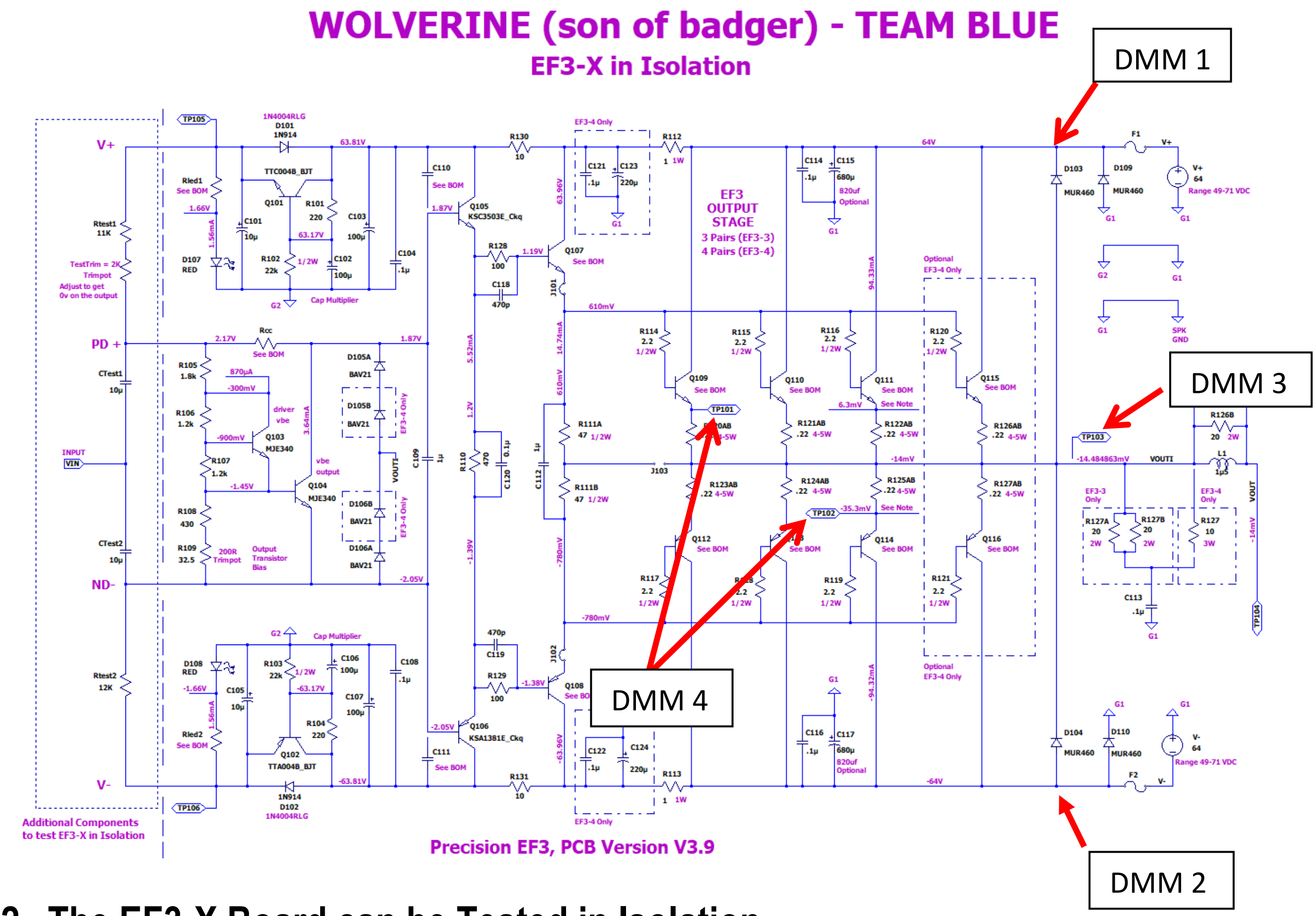
22.1 The IPS Board can be Tested in Isolation

A. Add the additional wiring connections as shown in the schematic above.
   a. Ensure you add all additional wiring including the C1 to G2 connection.
B. Connect a suitable supply voltage and appropriate 50mA fuses.
C. A current limiting DC power supply is ideal for initial tests or troubleshooting.
   a. If you are preforming your troubleshooting tests with a low voltage 30v dual rail power supply it is necessary to place temporary 9.1K resistor in parallel with R17. This must be removed before returning before returning to a full rail voltage test.
D. Next, short the signal Input terminal block shown on the schematic above to ground.
E. Connect DMM's as shown with the negative lead of DMM 1, 2 & 3 going to ground and turn your supply voltage on.
   i. Check DMM 1 & 2 are close to the expected rail voltage.

If not turn the voltage supply off, and check your fuses, you may need to follow some of the advice listed at Point 22 to help trace down the problem.

   ii. Check DMM 3 is close to 0mv if not adjust the DC offset using the trimpot R25.
   iii. Check DMM4 is close to 5V if not adjust CCS1 using the trimpot R11.
F. If everything checks out, apply an input sine wave of 100 mVpp & scope or measure the AC voltage with a DMM at the Collector of Q13 you should see around 2.5 Vpp.

EF3-X Board schematic:



22.2 The EF3-X Board can be Tested in Isolation

A. Add the additional components & wiring connections as shown in the schematic above.
   i. Rtest1 = 11K (0.5 Watt)
   ii. Rtest2 = 12K (0.5 Watt)
   iii. TestTrim = 2K Trimpot
   iv. Ctest1, 2 = 10uF (Non Polarized / Bipolar)
B. A current limiting DC power supply is ideal for initial tests or troubleshooting.
C. Increase the resistance for R109 to the maximum resistance of approximately 500 ohms. Measure the resistance between pin's 1 & 2. Use the silkscreen on the underside of the PCB to ensure you measure the resistance correctly. This is very a very important step do not mix up pin's 1 & 2 for pin's 2 & 3 as this will definitely blow your fuses or even damage your output transistors.
D. Next, short the Input shown on the schematic above to ground.

22.3 If your Output Transistors are Not Installed

A. Connect a suitable supply voltage and appropriate 200 mA fuses.
B. Use Test Hook Clips to make your connections between R111A (TP107) (DMM +) and R111B (TP108) (DMM -), and set up your voltmeter to monitor the bias voltage (millivolts) between R111A (TP107) and R111B (TP108).
C. Turn your supply voltage on.
   i) Check DMM 1 & 2 are close to the expected rail voltage.

If not turn the voltage supply off, and check your fuses, you may need to follow some of the advice listed at Point 22 to help trace down the problem.

ii) Check DMM 3 is close to 0 mV if not adjust the DC offset using the 2K Trimpot.

iii) Check DMM4 is between 800 mV and 900 mV. It should be close if R109 is set to maximum resistance.

D. If everything checks out, turn off your voltage supply. The output stage board checks out.

22.4 If your Output Transistors are Installed

A. Connect a suitable supply voltage and appropriate 200 mA fuses.
B. Connect DMM's as shown with the negative lead of DMM 1, 2 & 3 going to ground and DMM 4 going from TP101 (DMM +) to TP102 (DMM -).
C. Turn your supply voltage on.
   i. Check DMM 1 & 2 are close to the expected rail voltage.

If not turn the voltage supply off, and check your fuses, you may need to follow some of the advice listed at Point 22 to help trace down the problem.

   ii. Check DMM 3 is close to 0mv if not adjust the DC offset using the 2K Trimpot.
   iii. Check DMM4 is close to 0mv. It should be close if R109 is set to maximum resistance.

D. If everything checks out, turn off your voltage supply and replace the 200 mA fuses with 1A fuses.
E. Next, turn your supply voltage on.
   i. Check DMM 1 & 2 are still close to the expected voltage.
   ii. Check DMM 3 is still close to 0mv if not adjust the DC offset using the 2K Trimpot.
   iii. While watching DMM4 slowly reduce resistance of R109 until your DMM reads between 5 mV & 10mV.
   iv. Recheck points i & ii above.
   v. Continue to watch DMM4 and slowly reduce R109 a little further until your DMM reads between 20 mV to 22mV.
   vi. Recheck points i & ii above.
   vii. If everything checks out you have completed the DC voltage test. Feel free to check if any other voltages shown on the schematic are close to your measured value.
   viii. Next, turn your supply voltage off so we can prepare for an AC small signal voltage test.
F. Connect an input sine wave from a signal generator as shown above and set its output voltage to 100 mVpp and scope or measure the AC voltage with a DMM at the output terminal to ground. You should see around 95 mVpp.
G. If everything checks out, you have completed the AC small signal voltage test and the output stage board checks out.

23. General Circuit Building Advice

Below are some general tips and basic advice for checking, verifying, or troubleshooting the build of the Wolverine amplifier.

A. Check all resistor values are correct as per the schematic and BOM with a DMM prior to installation.
B. Check all capacitors if polarized are correctly oriented with respect to their polarity.
C. Check all wire links, solder bridges, and terminals are soldered securely.
D. Check the output inductor tightly wound. Then optionally place a thin layer of epoxy resin around the outside to prevent the coils from vibrating. Then soldered it in place securely.
E. Check the appropriate value fuses are installed and are still intact.
F. Check all semiconductors are soldered in the correct orientation.
G. Check there are no bridged tracks or solder pads. Sometimes small bits of excess solder can float around the PCB and short out traces.
H. Check that no output transistor legs are in contact with the heatsink – take extra care with this step, Use the multimeter continuity beep function. There should be no continuity (no beep) when one probe is on the heatsink and the other probe is on any of the output stage transistor legs and metal tabs if exposed and able to be probed.
a. Take extra care with this step. It's VERY important.
I. Check that all wiring is correct and attached securely to the power supply and input terminals.
J. Check that no parts are visibly overheated or burned.
K. Check that the voltages listed on the schematic are similar to the measured voltages.
L. Thoroughly clean all flux and excess solder from the board prior to testing.

Notes: In order to complete a complex project one technique is to treat it as a group of smaller tasks. For example, build the amplifier front end, build the amplifier power supply, build the amplifier output stage, and if possible, test the individual sections on the bench prior to installing them into the chassis.

You can also break the above steps down even further so that as you build you have the best chance of success at the completion of each individual section.

For example: install the smallest and least sensitive components first, then the larger components, and finally the hardware components (Terminal blocks, fuse holders, etc.).

23.1 Other General Suggestions

To break it down even further you can check each component reasonably quickly and easily as you install them and tick them off your BOM list as you solder them into the board.

It's also a good idea to mark each installed component on your schematic as you go.

It is good practice to check each part with a digital multimeter, LCR meter, or component tester if you have one. Check them against the schematic value and BOM before you solder them into the board. This only takes a few seconds and will give you the confidence if something goes wrong that you checked all the components were correct, within tolerance, and installed in the correct location.

Here are a few extra tips on how to measure some basic parts:

Checking resistors:

https://www.allaboutcircuits.com/technical-articles/measuring-resistance-in-circuit-and-out/
https://www.youtube.com/watch?v=v8psOCkN-OQ&t=70s

Checking capacitors:

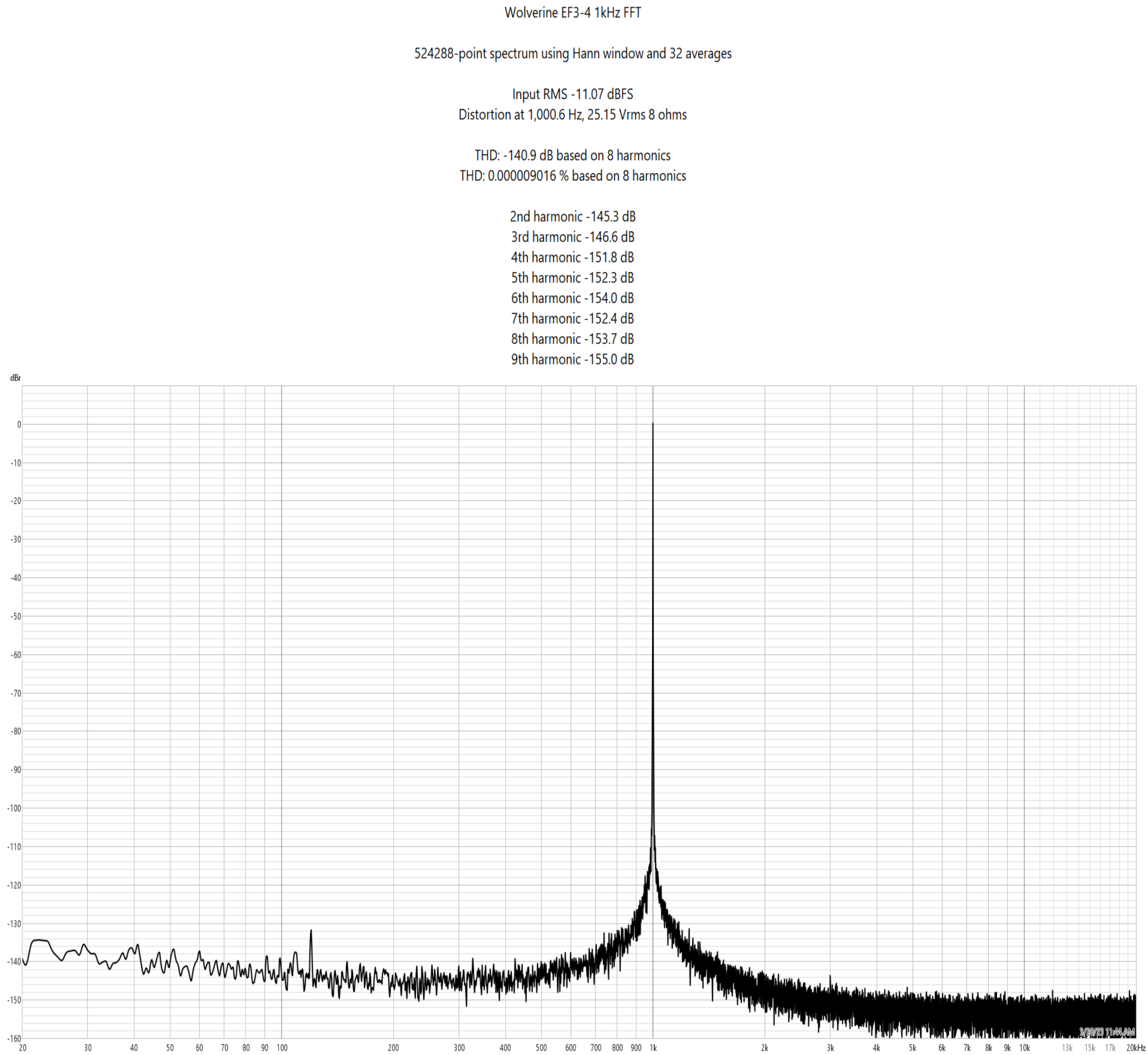
https://www.youtube.com/watch?v=wrJXdJbX5zg

LCR Meter:
Using a basic LCR meter and M328 / Mtester component checker with examples of inductors and transistors from EEVblog: https://www.youtube.com/watch?v=7Br3L1B80ow

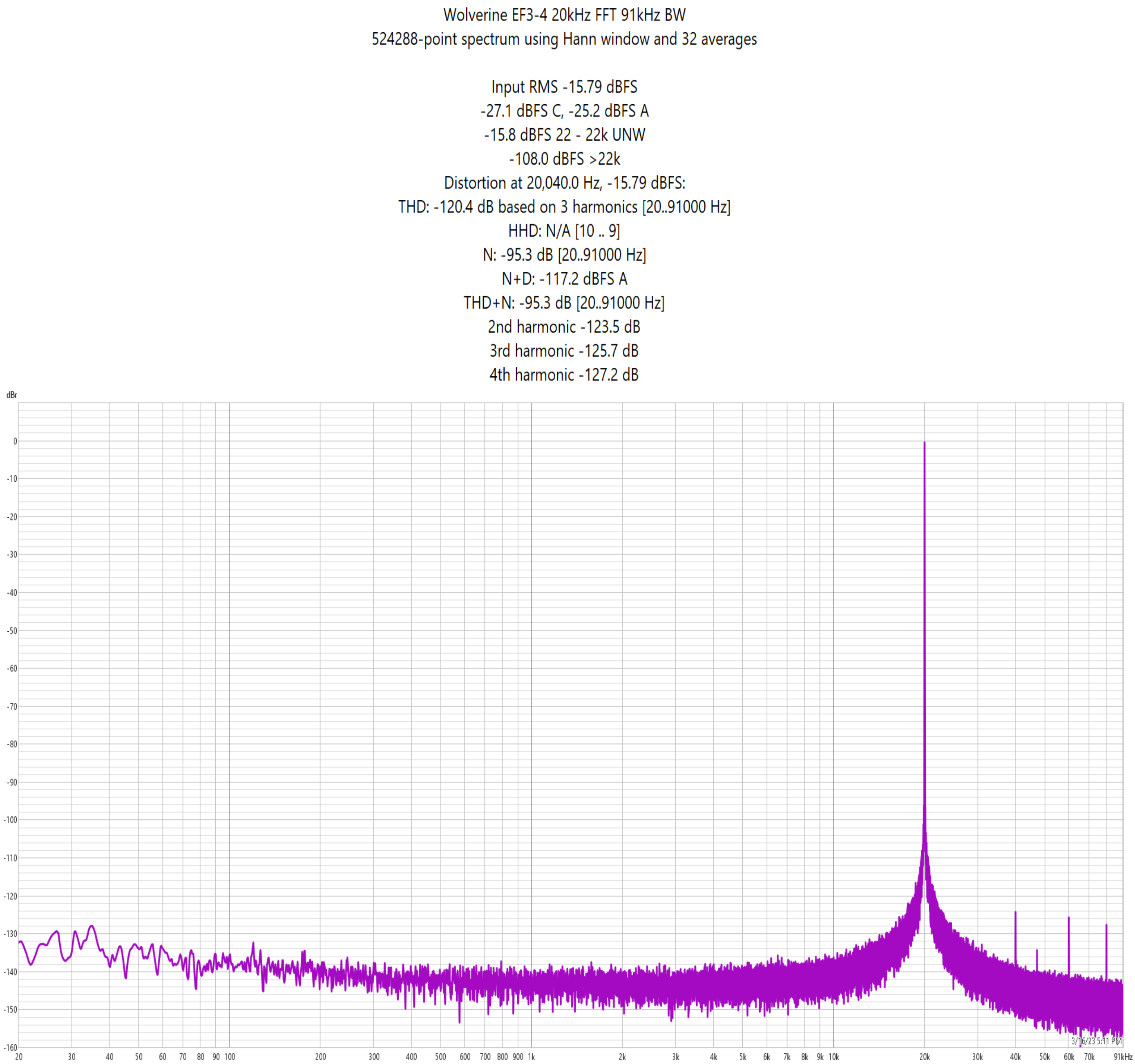
Checking the basic function of a transistor (BJT):
https://www.allaboutcircuits.com/textbook/semiconductors/chpt-4/meter-check-transistor-bjt/

24. Results and Testing

24.1 1 KHz FFT Notch 140.9dBr



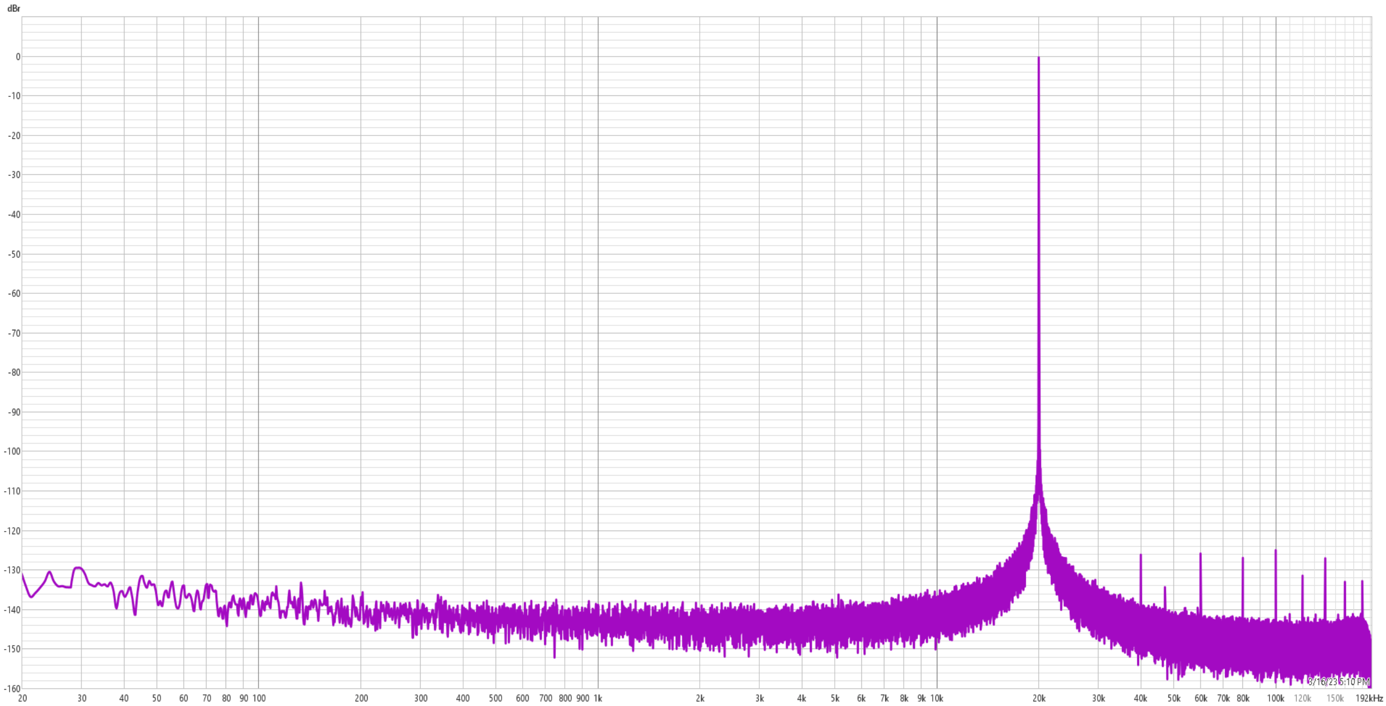
24.2 20 KHz FFT Notch 91 Khz Bandwidth



24.3 20 KHz FFT Notch 192 Khz Bandwidth

Wolverine EF3-4 20kHz FFT 192kHz BW
524288-point spectrum using Hann window and 32 averages

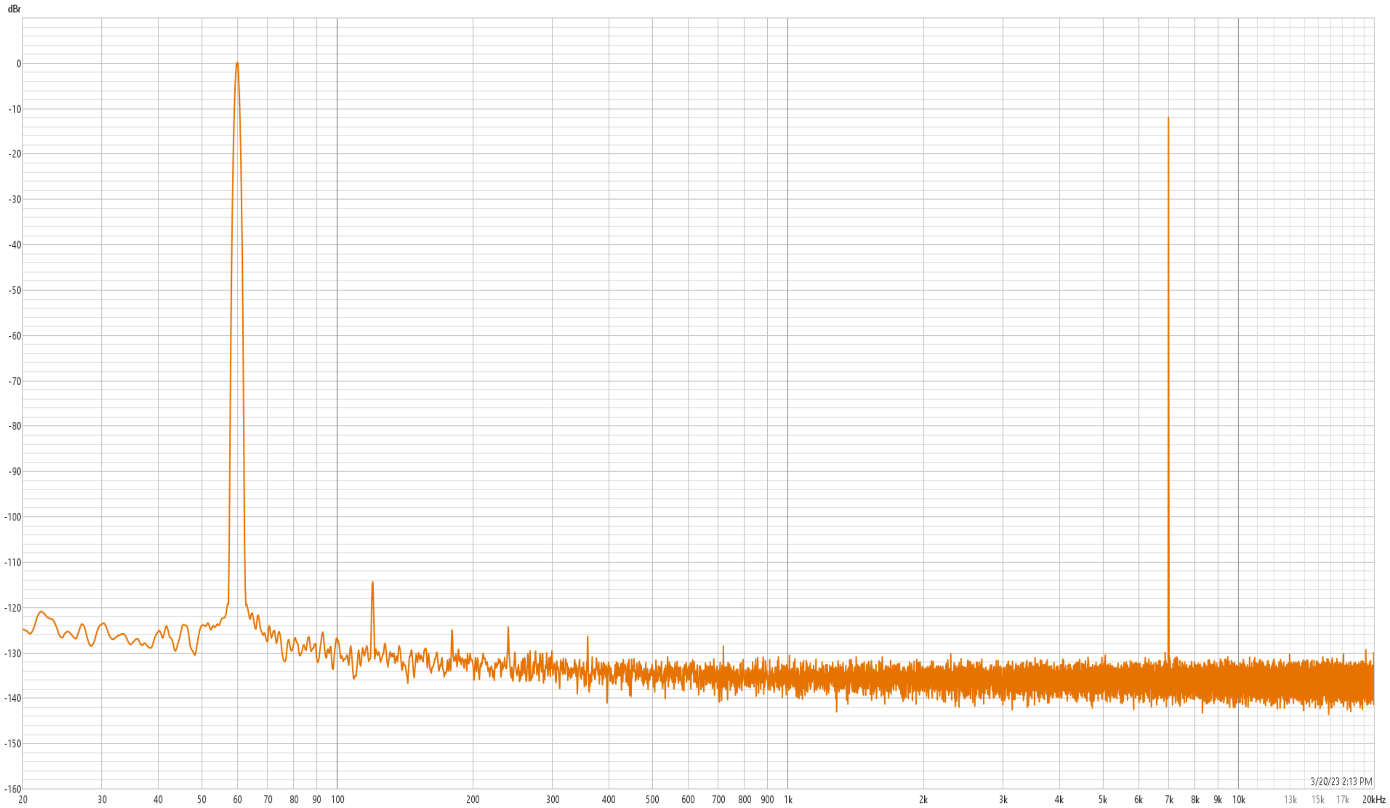
Input RMS -15.63 dBFS
-26.9 dBFS C, -25.0 dBFS A
-15.6 dBFS 22 - 22k UNW
-108.1 dBFS >22k
Distortion at 20,040.0 Hz, -15.63 dBFS:
THD: -117.9 dB based on 8 harmonics [20..182400 Hz]
HHD: N/A [10 .. 9]
N: -93.9 dB [20..182400 Hz]
N+D: -117.1 dBFS A
THD+N: -93.8 dB [20..182400 Hz]
2nd harmonic -124.9 dB
3rd harmonic -125.7 dB
4th harmonic -126.8 dB
5th harmonic -124.2 dB
6th harmonic -130.6 dB
7th harmonic -127.0 dB
8th harmonic -132.4 dB
9th harmonic -131.3 dB



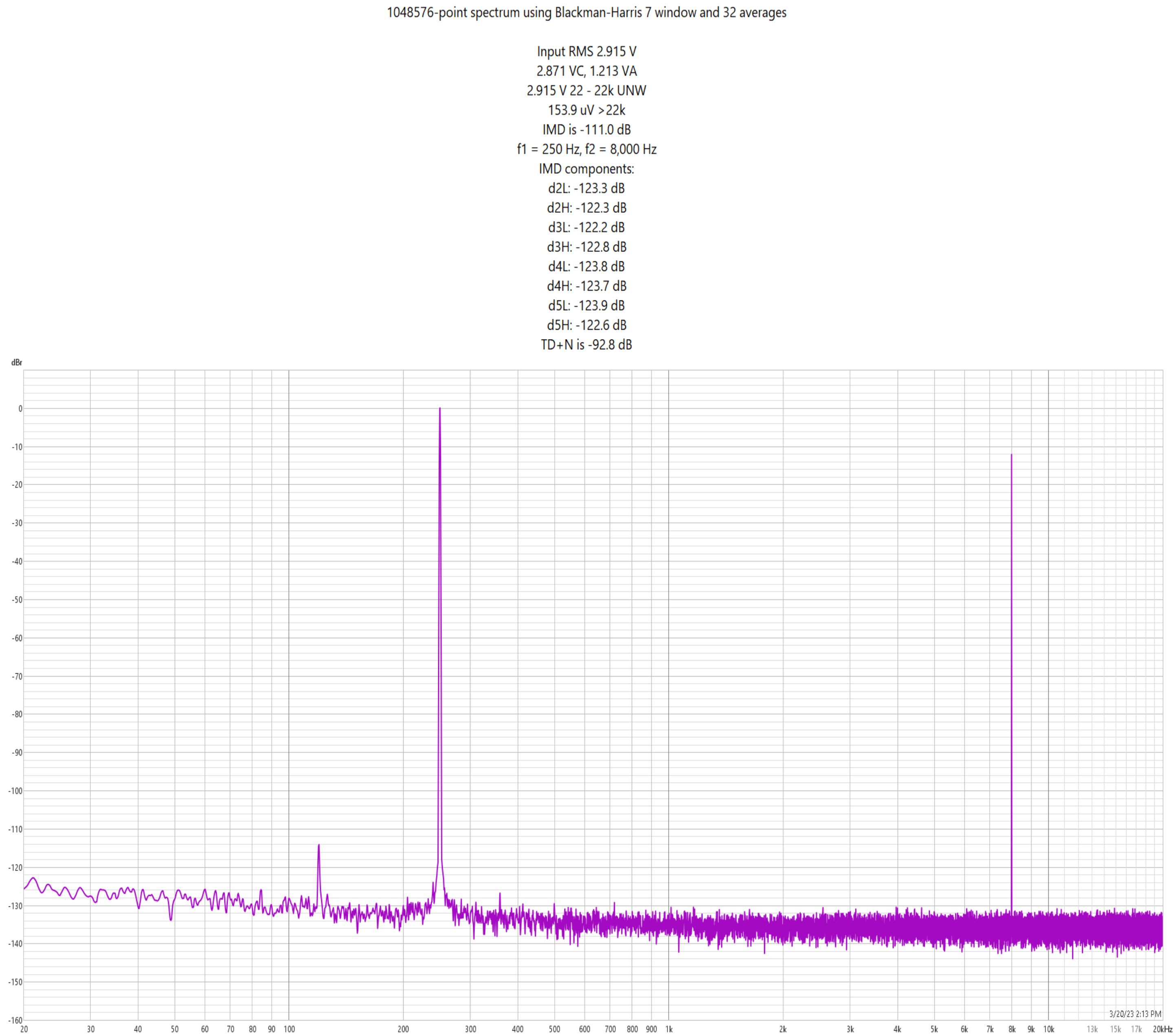
24.4 SMPTE

1048576-point spectrum using Blackman-Harris 7 window and 32 averages

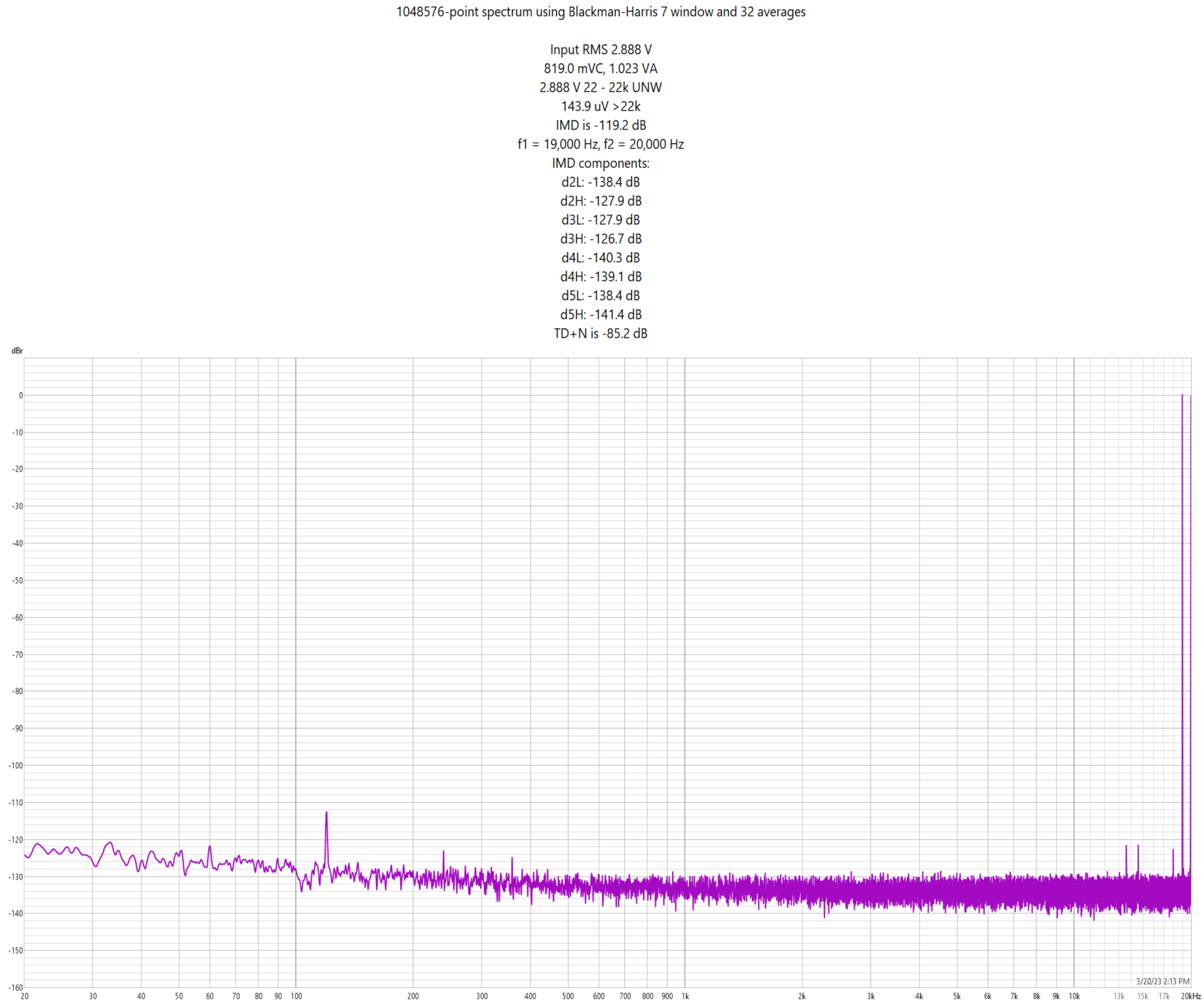
Input RMS 2.909 V
2.598 VC, 679.5 mVA
2.909 V 22 - 22k UNW
153.8 uV >22k
IMD is -111.0 dB
f1 = 60 Hz, f2 = 7,000 Hz
IMD components:
d2L: -123.6 dB
d2H: -121.7 dB
d3L: -119.9 dB
d3H: -124.1 dB
d4L: -123.3 dB
d4H: -124.9 dB
d5L: -124.7 dB
d5H: -124.2 dB
TD+N is -92.8 dB



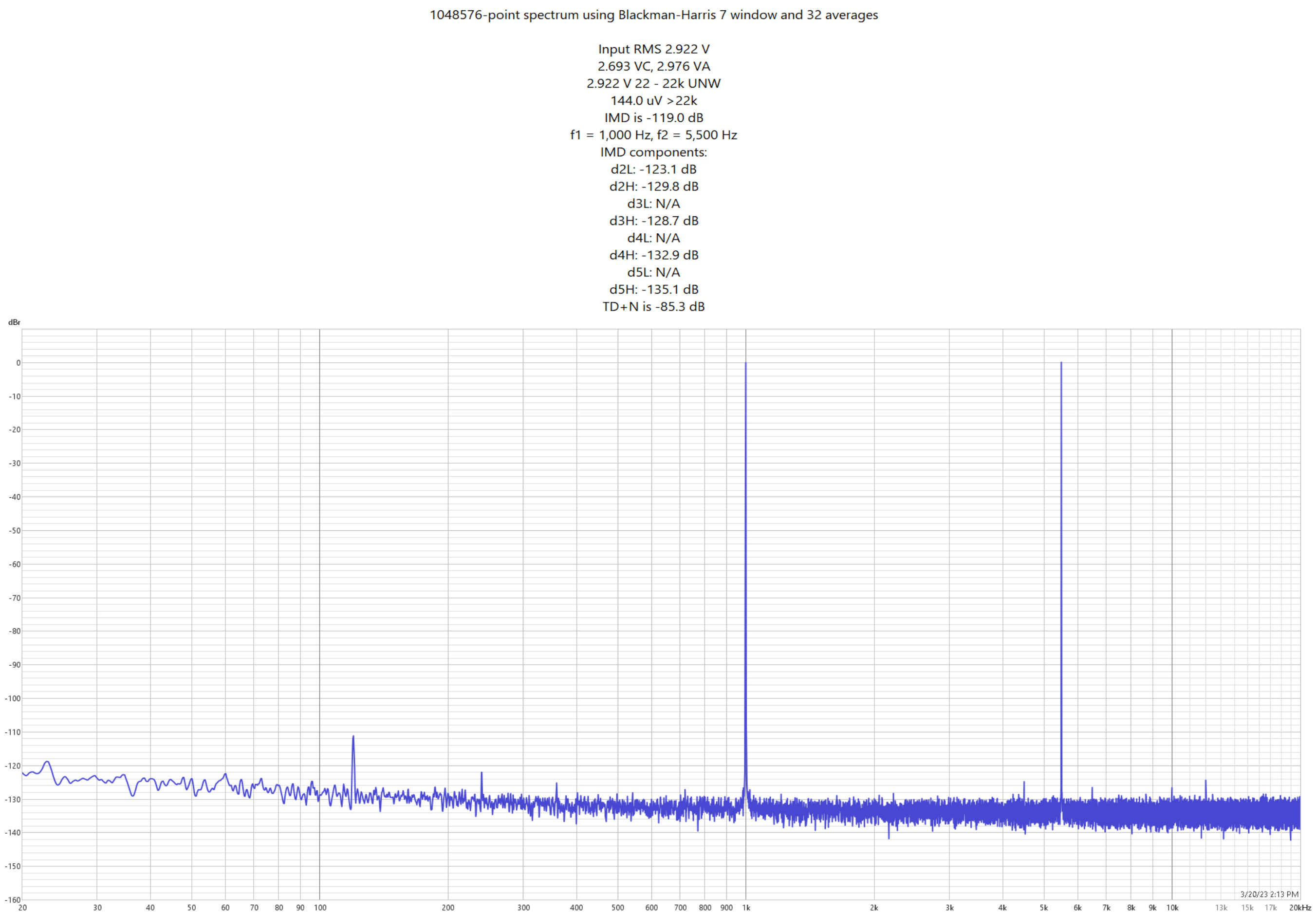
24.5 DIN


1048576-point spectrum using Blackman-Harris 7 window and 32 averages
Input RMS 2.915 V
2.871 VC, 1.213 VA
2.915 V 22 - 22k UNW
153.9 uV >22k
IMD is -111.0 dB
f1 = 250 Hz, f2 = 8,000 Hz
IMD components:
d2L: -123.3 dB
d2H: -122.3 dB
d3L: -122.2 dB
d3H: -122.8 dB
d4L: -123.8 dB
d4H: -123.7 dB
d5L: -123.9 dB
d5H: -122.6 dB
TD+N is -92.8 dB
dBr
3/20/23 2:13 PM
20kHz

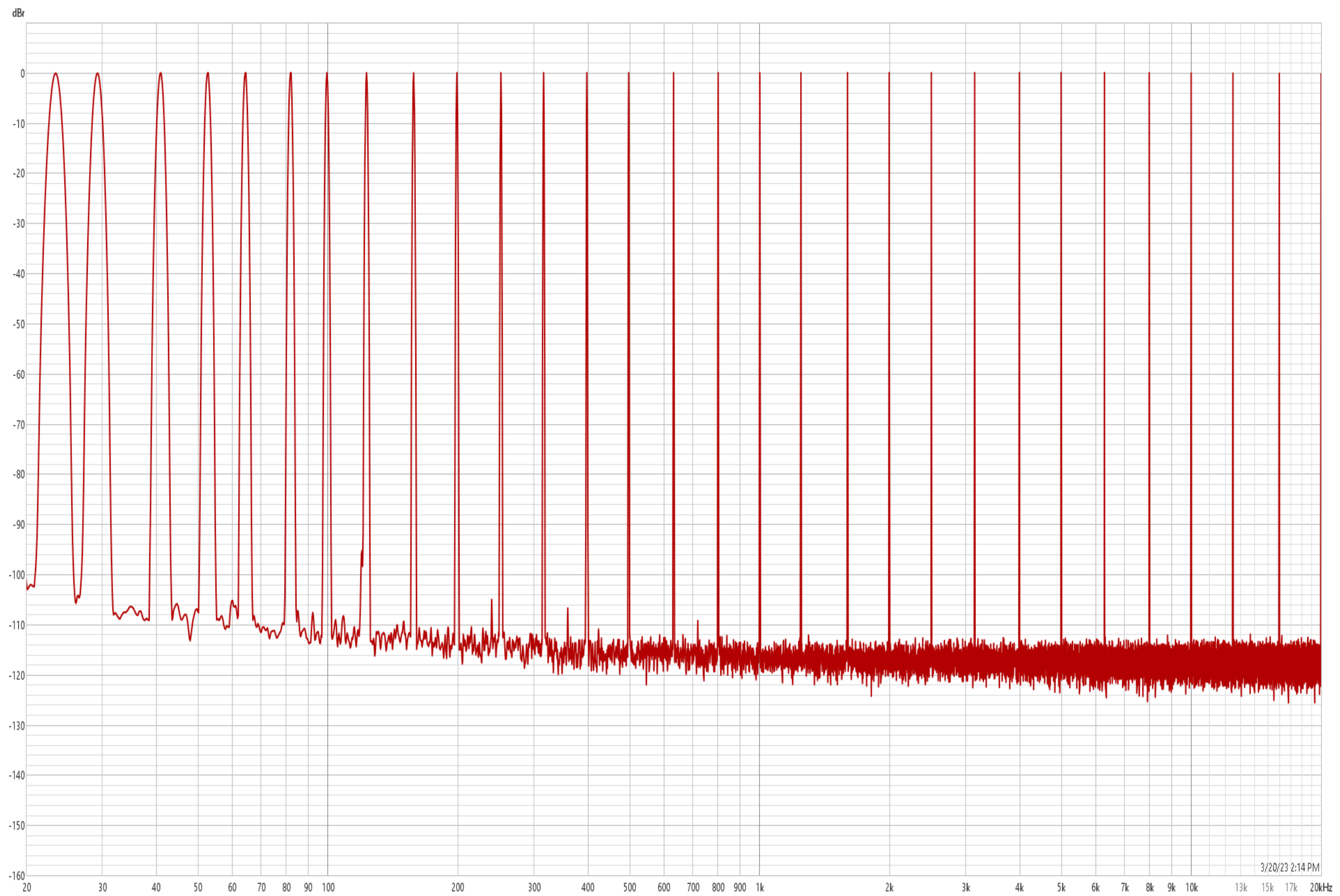
24.6 CCIF


1048576-point spectrum using Blackman-Harris 7 window and 32 averages
Input RMS 2.888 V
819.0 mVC, 1.023 VA
2.888 V 22 - 22k UNW
143.9 uV >22k
IMD is -119.2 dB
f1 = 19,000 Hz, f2 = 20,000 Hz
IMD components:
d2L: -138.4 dB
d2H: -127.9 dB
d3L: -127.9 dB
d3H: -126.7 dB
d4L: -140.3 dB
d4H: -139.1 dB
d5L: -138.4 dB
d5H: -141.4 dB
TD+N is -85.2 dB
dBr
3/20/23 2:13 PM

24.7 1 KHz 5.5 Khz

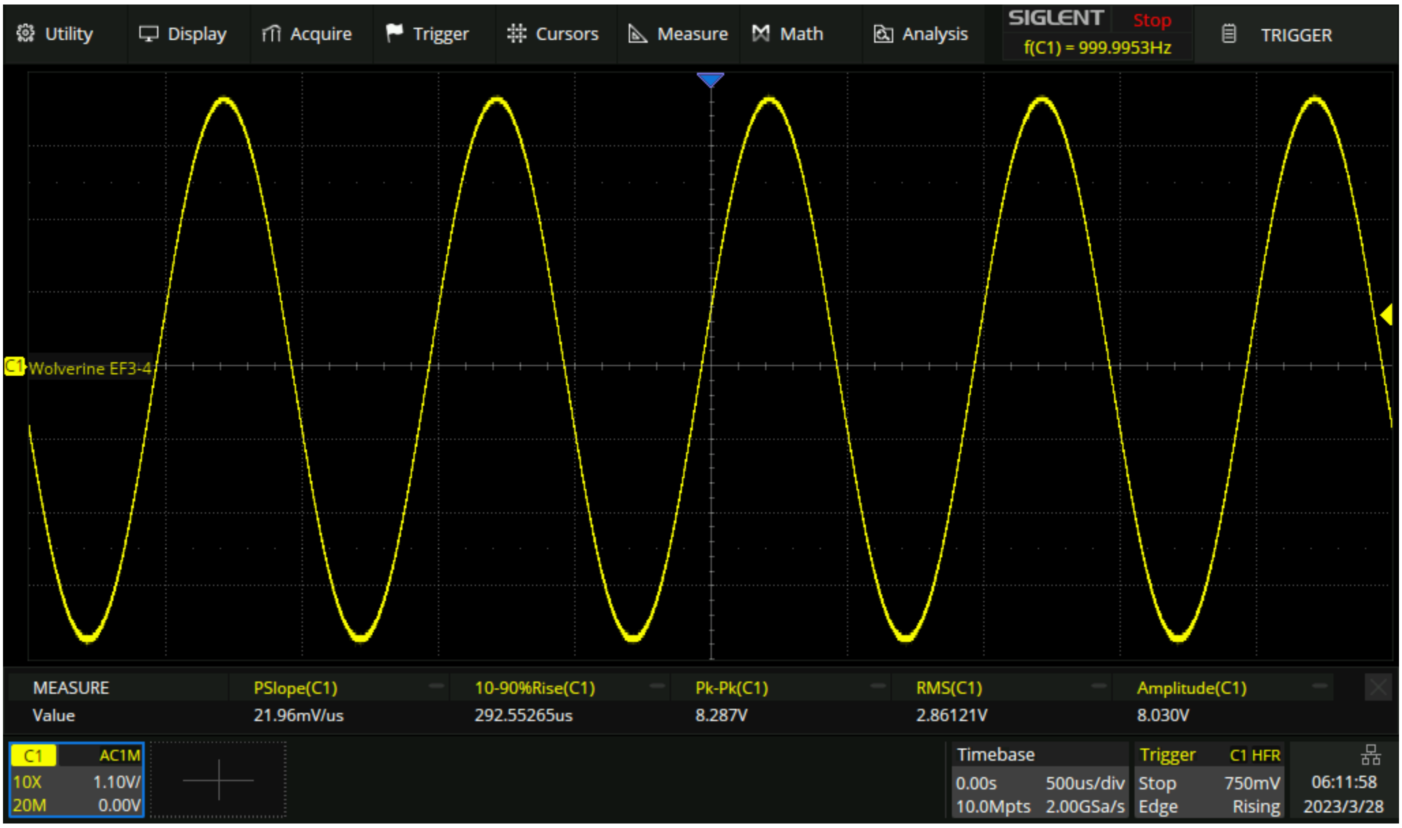


24.8 Multitone



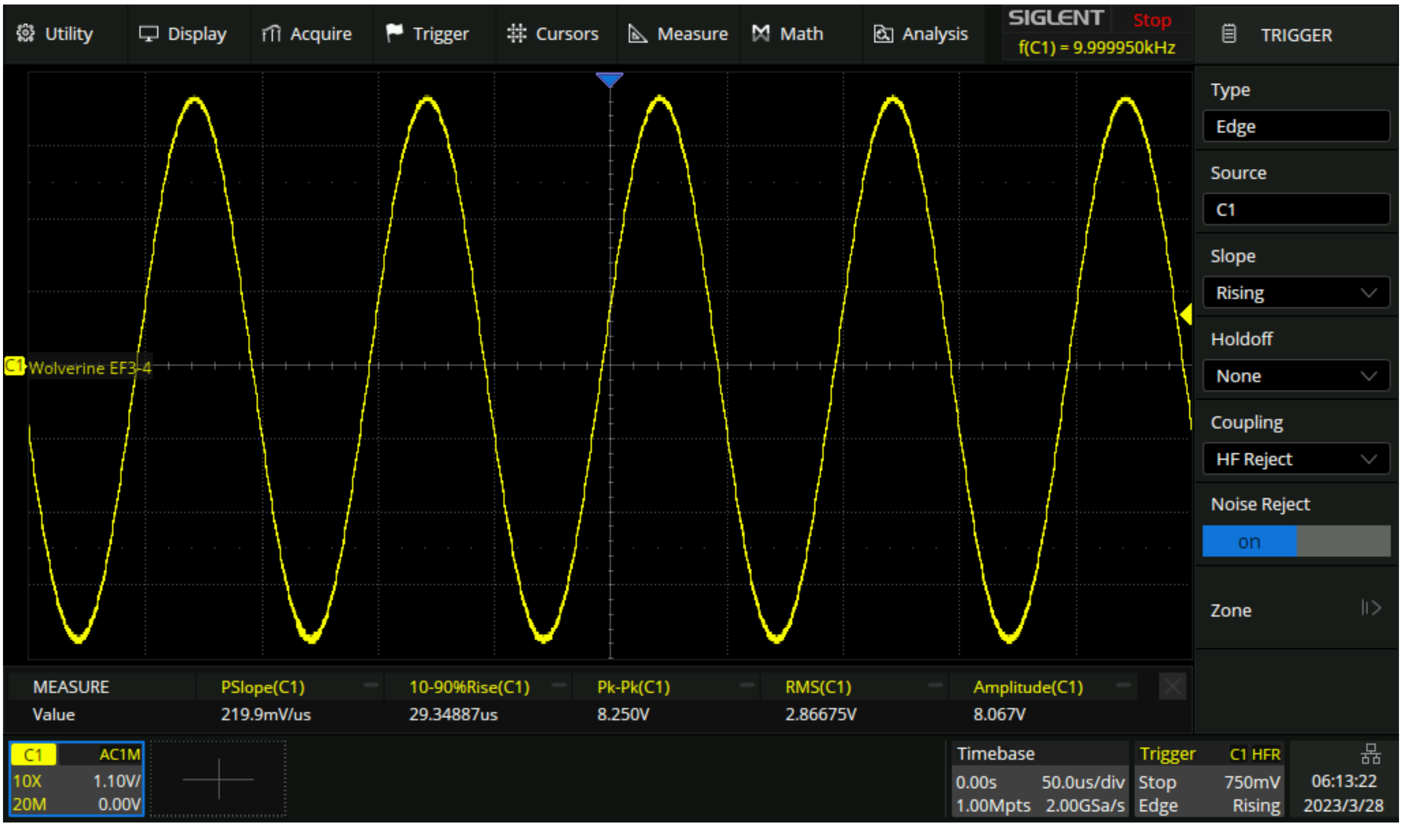
24.9 Sine Wave Testing

24.9.1 1 KHz Sine 2.861Vrms

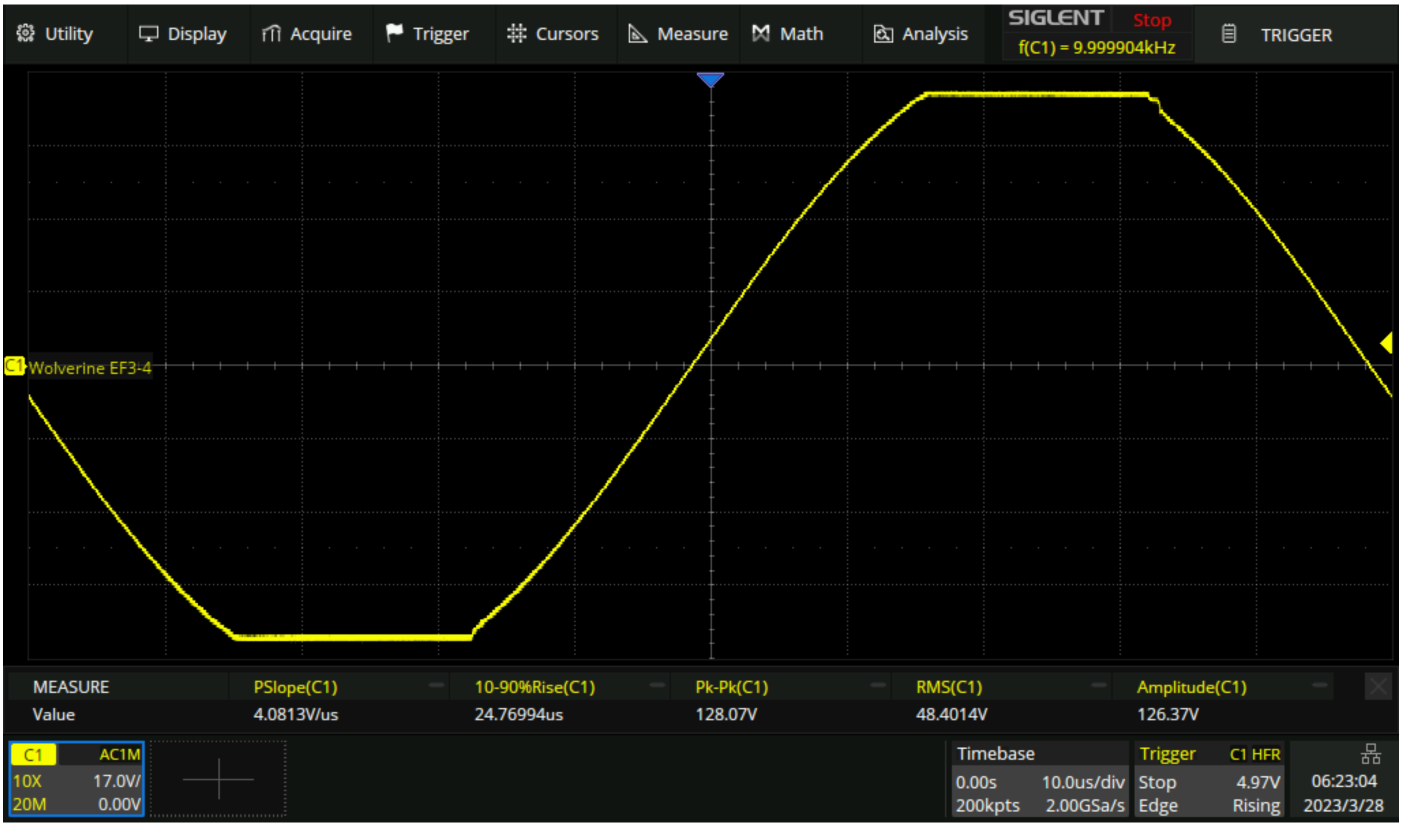


24.9.2 1 KHz Sine Hard Clipping

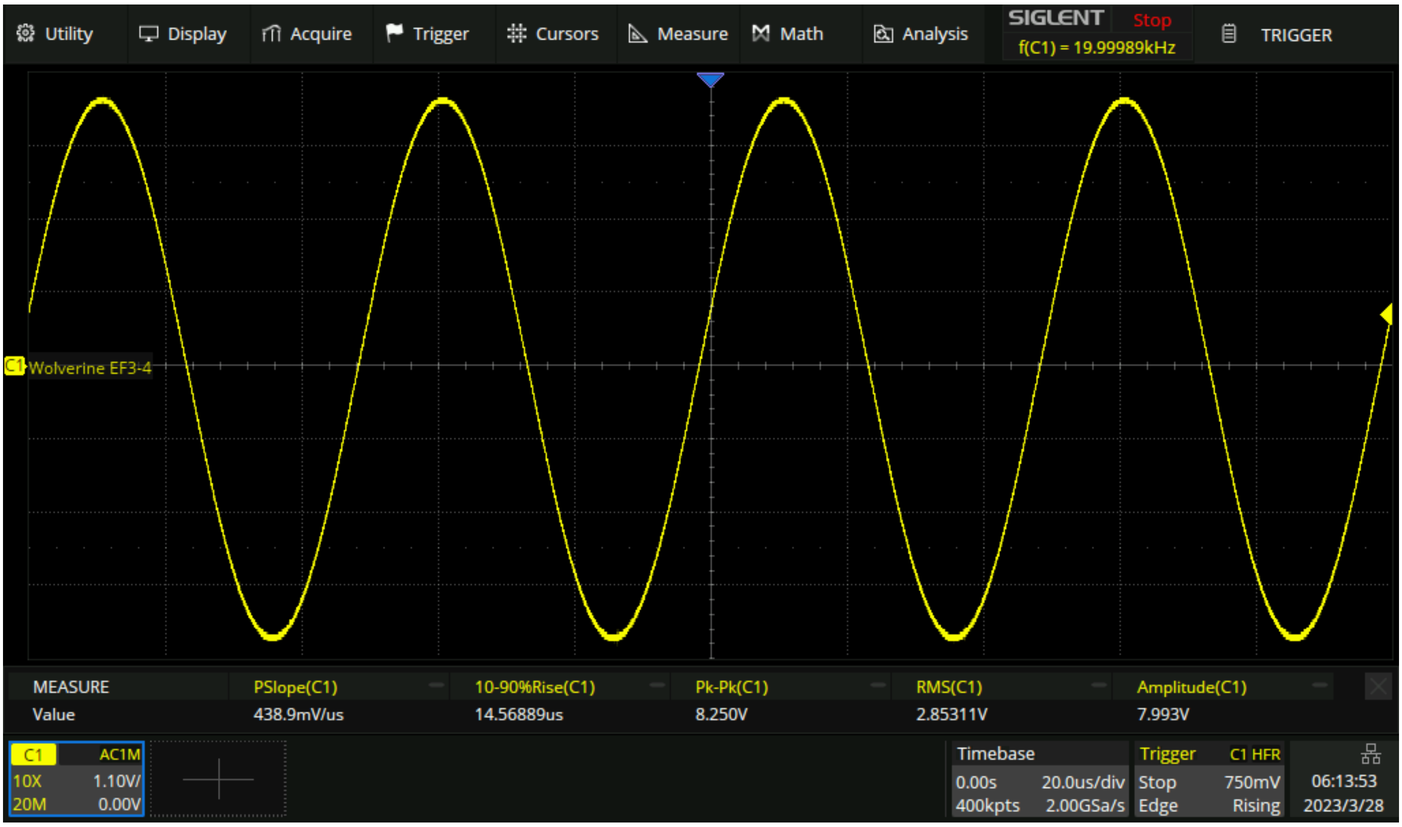
24.9.3 10kHz Sine 2.866Vrms



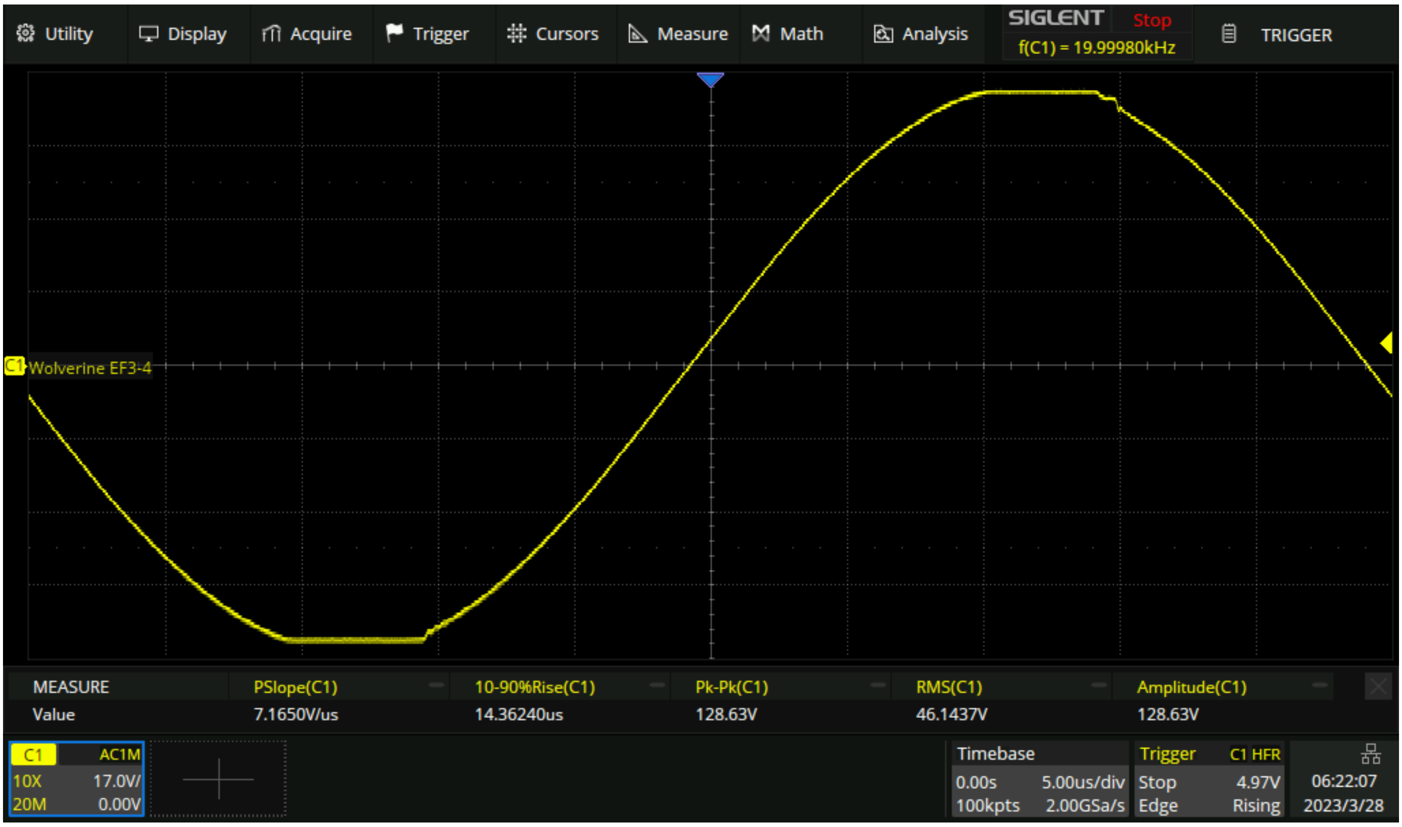
24.9.4 10kHz Sine Hard Clipping



24.9.5 20kHz Sine 2.853Vrms

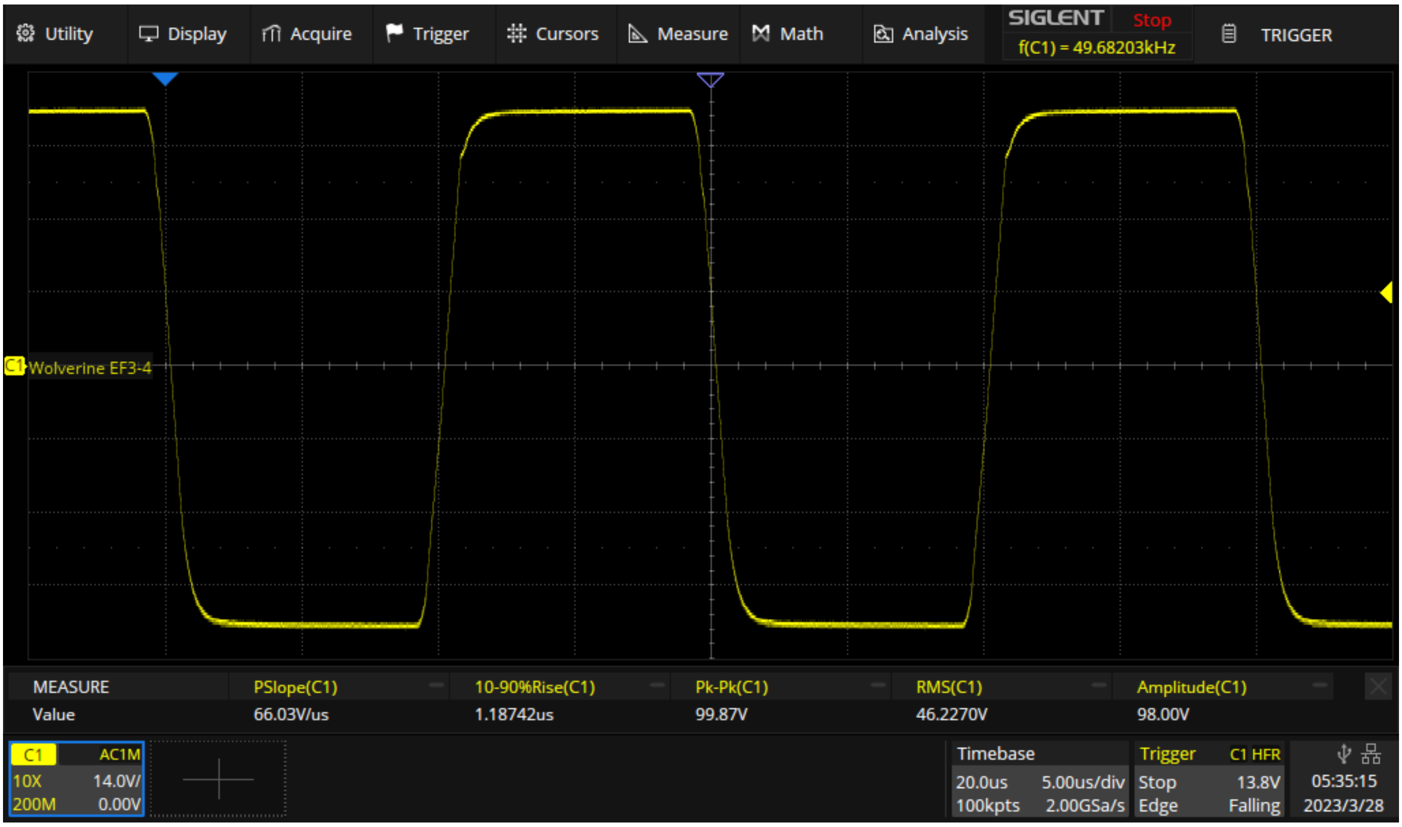


24.9.6 20kHz Sine Hard Clipping

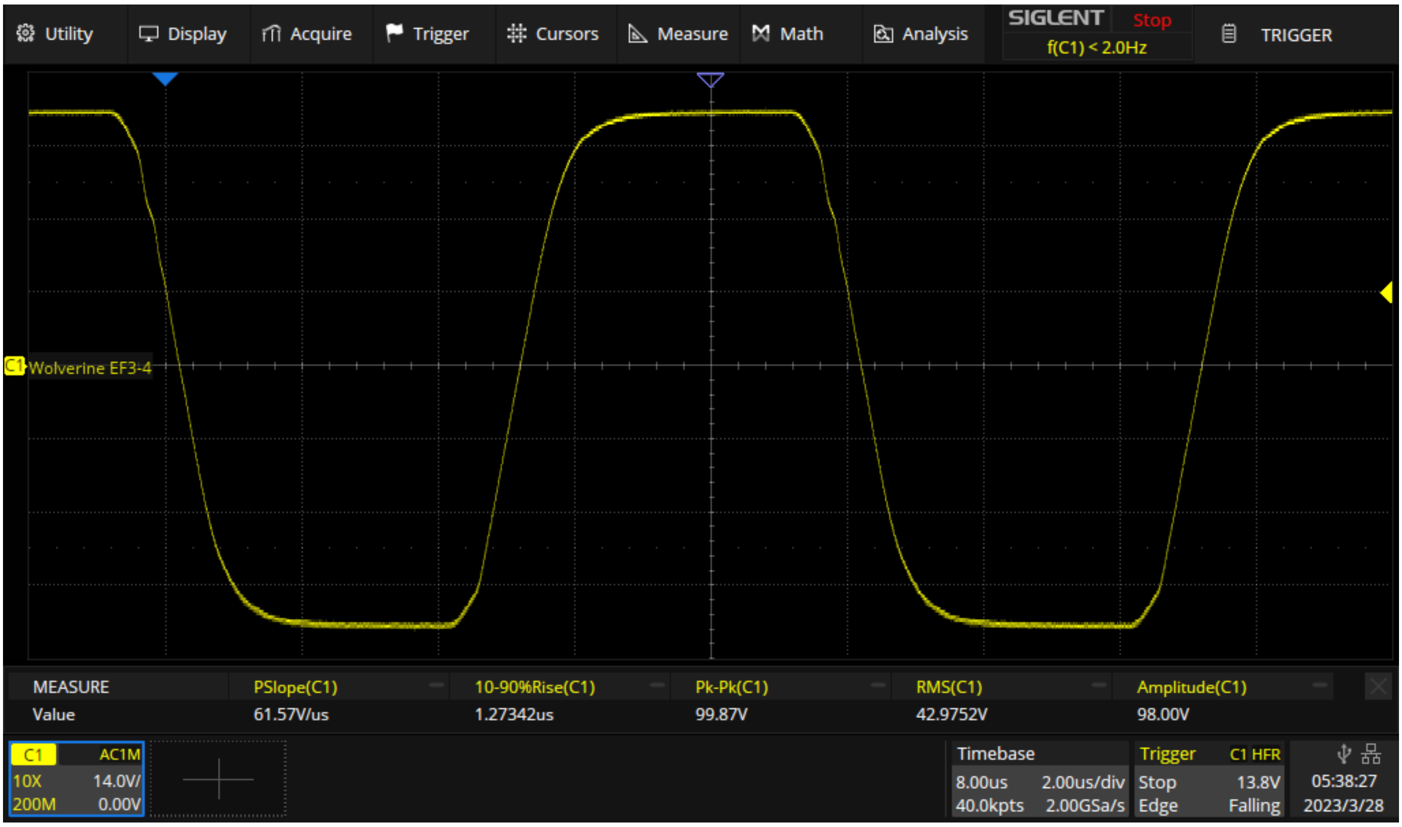


24.10 Square Wave Testing

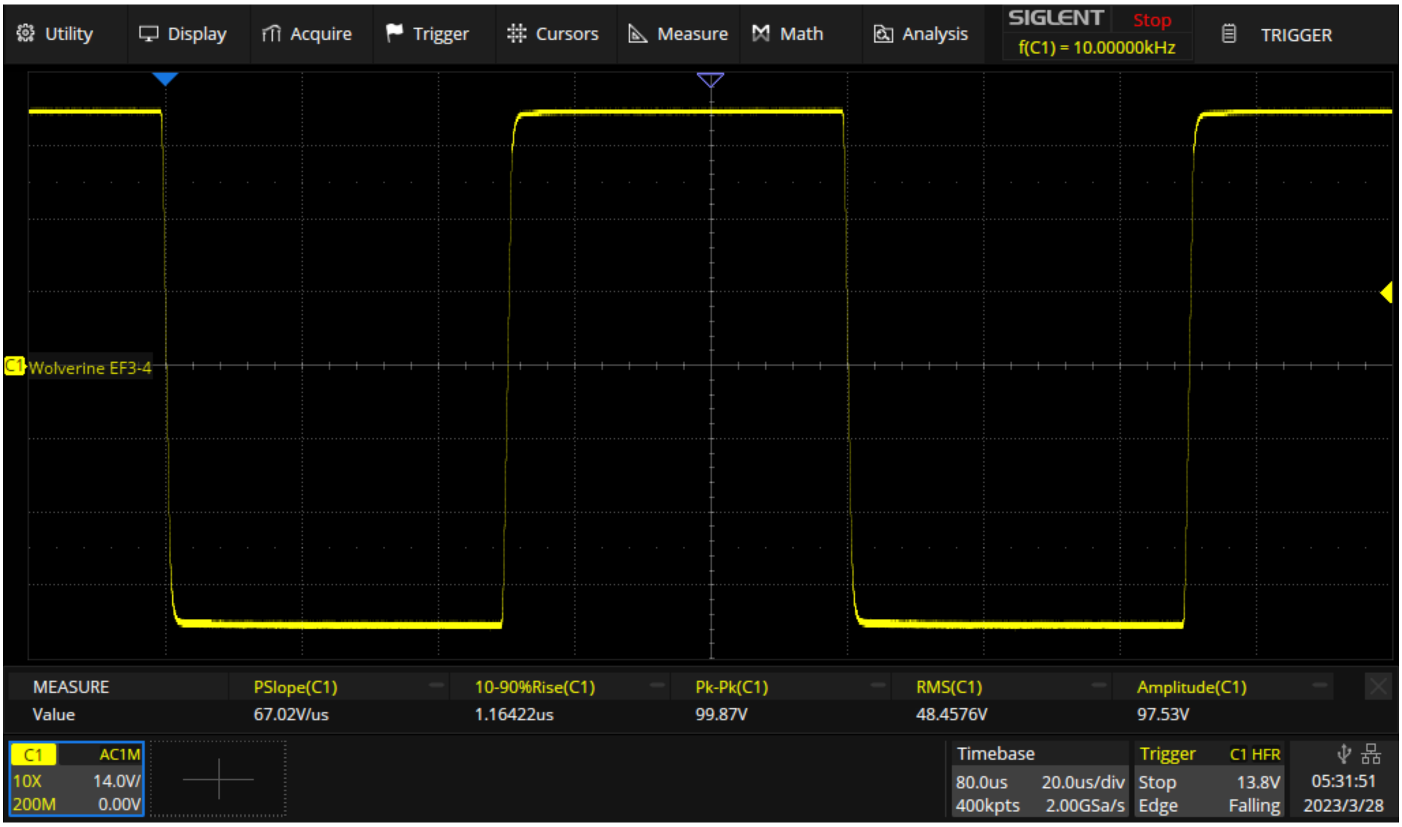
24.10.1 50kHz Square 100Vpp



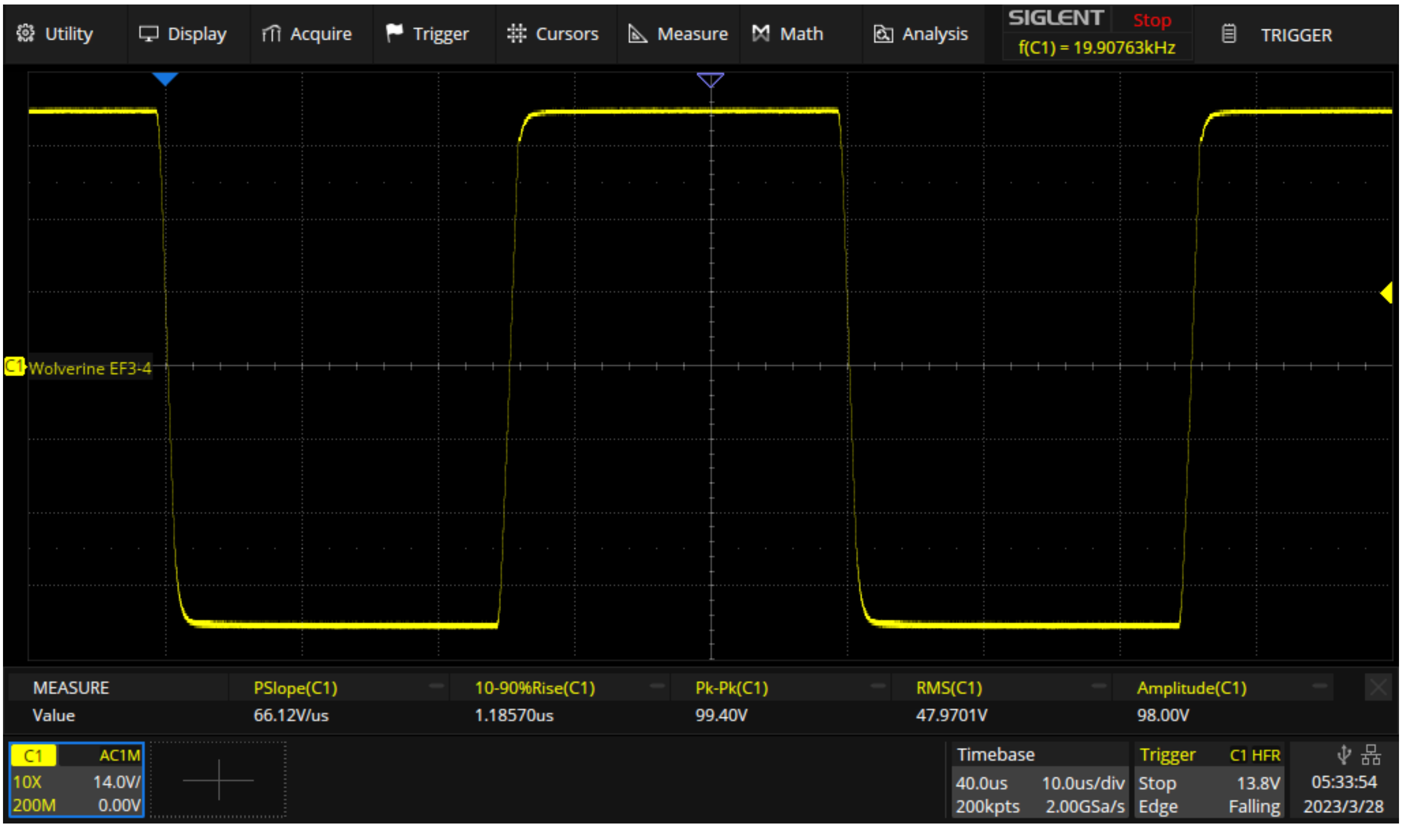
24.10.2 100kHz Square 100Vpp



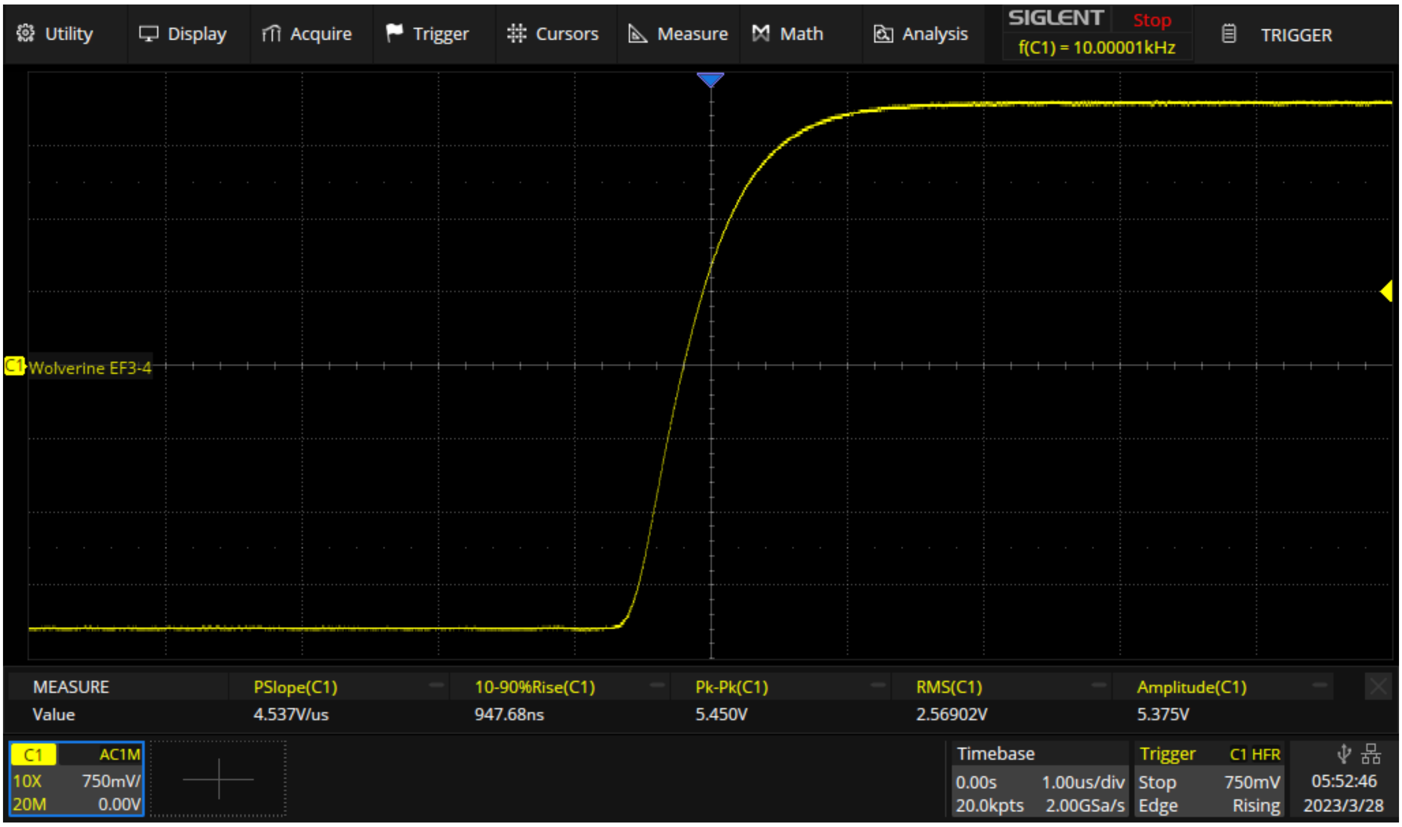
24.10.3 10kHz Square 100Vpp



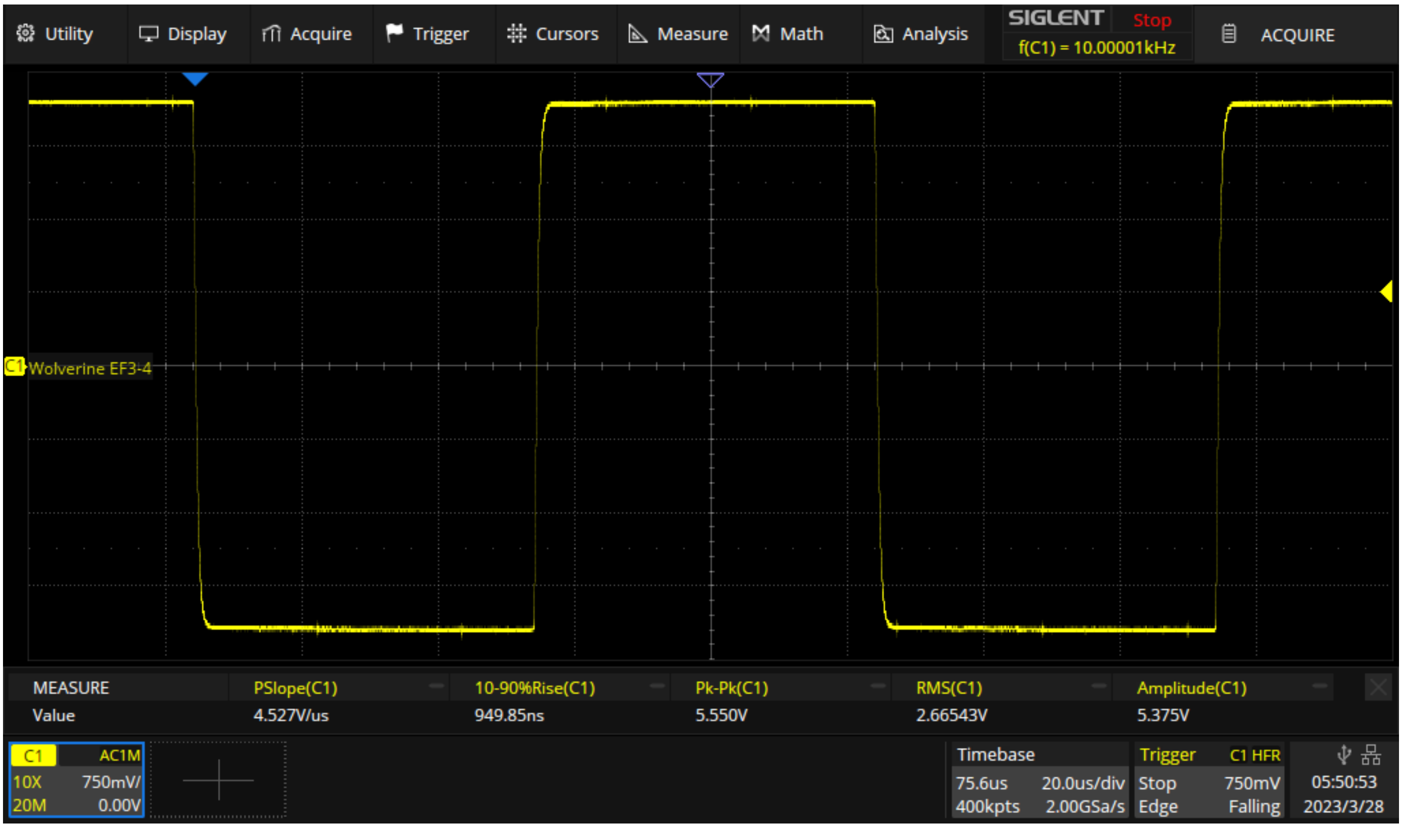
24.10.4 20kHz Square 100Vpp



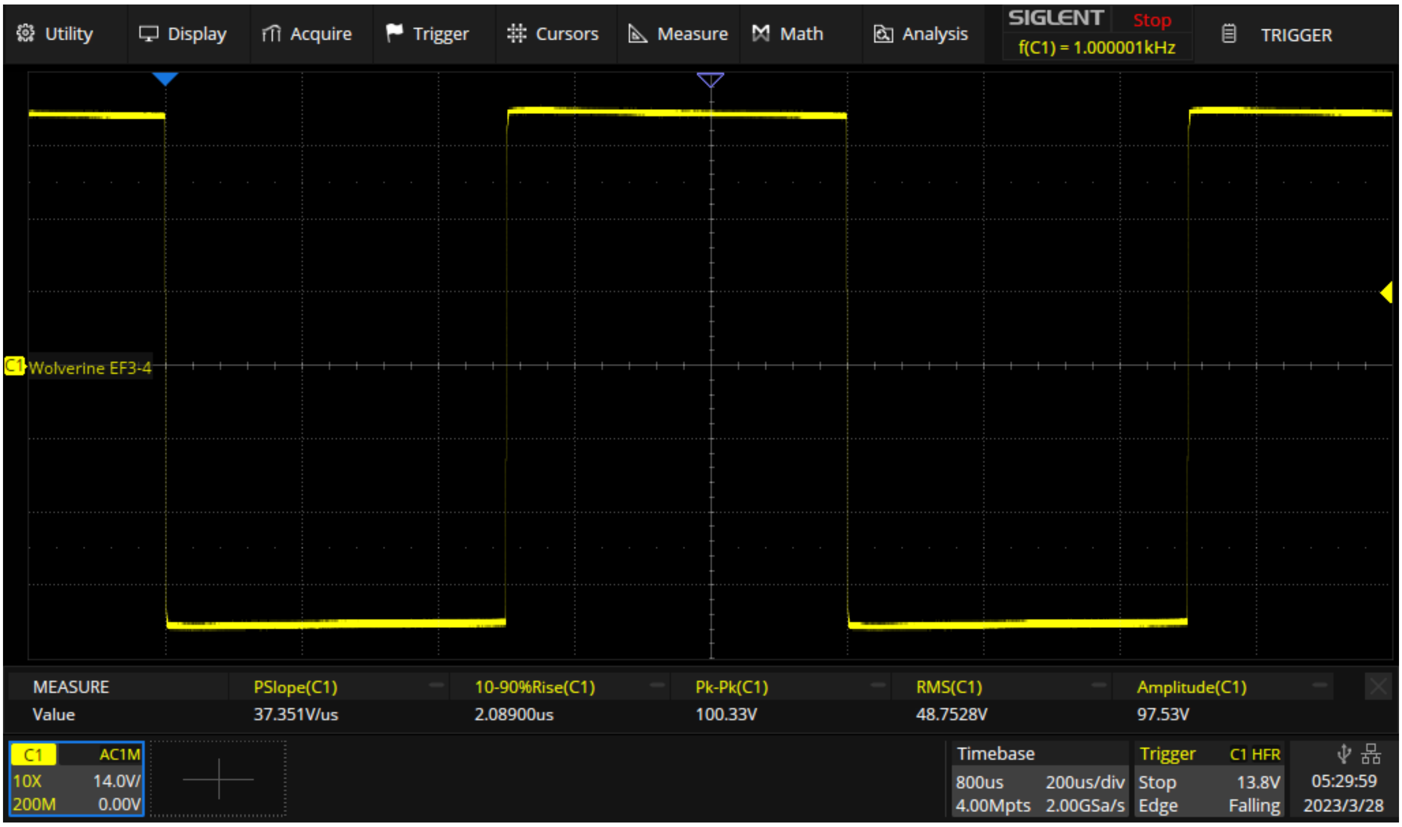
24.10.5 10kHz Square 5.45Vpp



24.10.6 10kHz Square 5.55Vpp

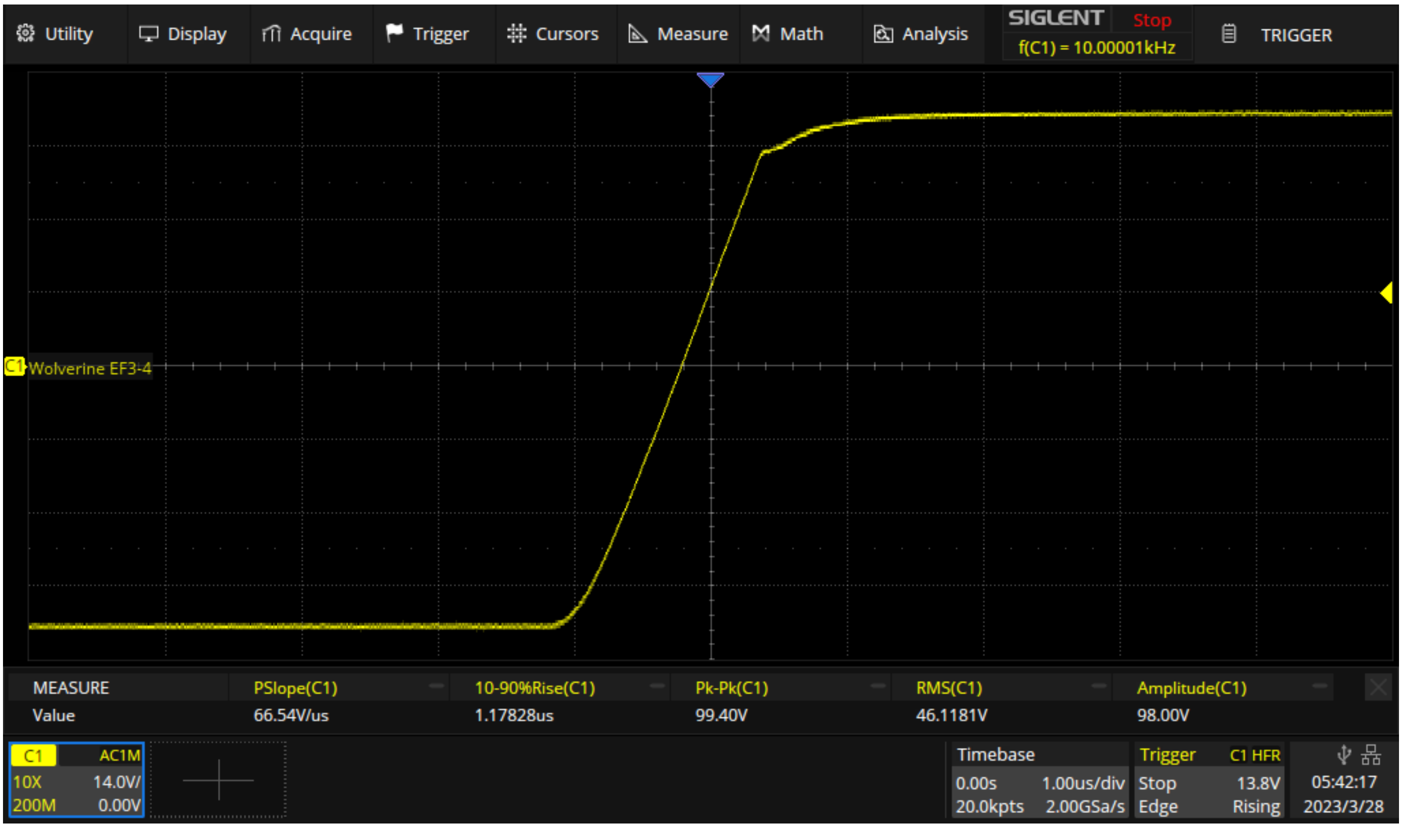


24.10.7 1kHz Square 100Vpp



24.11 Slew Rate Testing

24.11.1 10kHz Slew Rate



The calculated slew rate was 67 V/μs

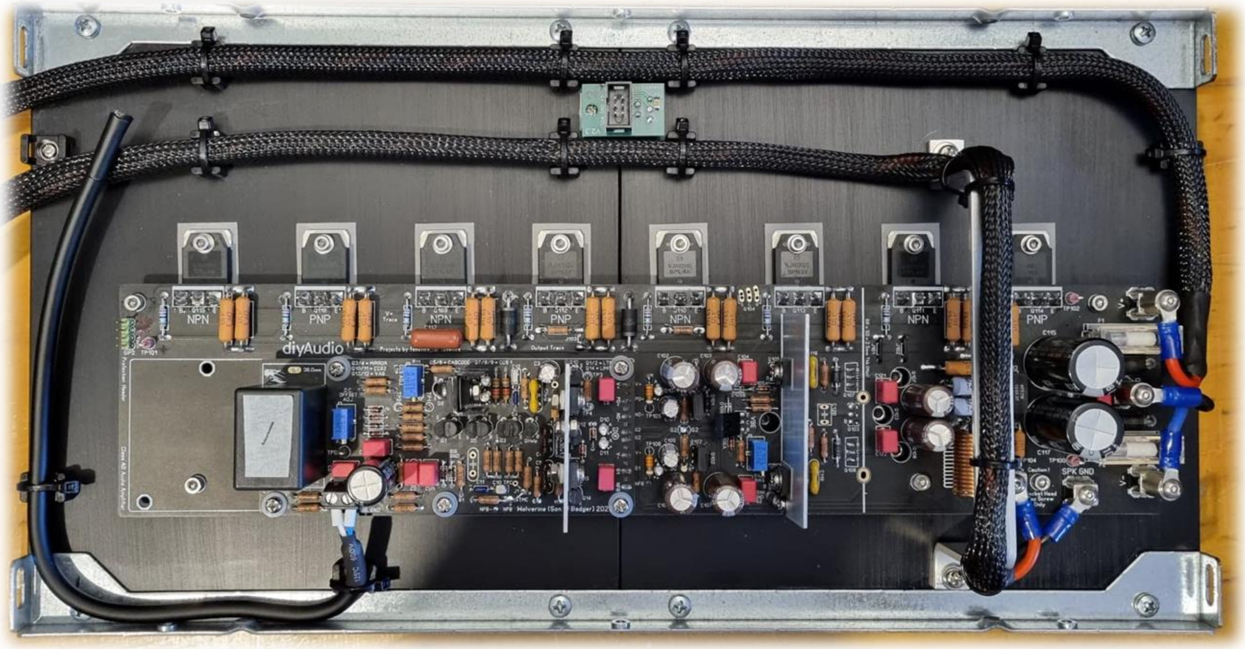
Revisions

- 16/04/2022 – REV 15
  - HEATSINK DIMENSIONS WERE ADDED AND UPDATED ON PAGES 10 & 11.
- 03/05/2022 – REV 16
  - HIGH CURRENT INDUCTOR UPDATE. PAGE 16 POINT 9.
  - YOUTUBE BUILD LINK ADDED.
- 07/05/2022 – REV 17
  - NOTE UPDATED, SMALL HEATSINK FABRICATION. HOLES CAN BE DRILLED ϕ3 OR TAPPED M3.
  - POINT 5. C. ADDED.
- 20/05/2022 – REV 18
  - POINT 14.H UPDATED.
  - POINT 15.A, 15.B & 15.I UPDATED.
- 22/05/2022 – REV 19
  - POINT 17. PART B UPDATED.
- 10/06/2022 – REV 20
  - MEASUREMENTS NOTES ADDED TO POINT 14.
  - POINT 15. PART G & H UPDATED.
- 30/6/2022 – REV 21
  - GROUND CONNECTIONS ADDED TO THE "IPS IN ISOLATION" CIRCUIT DIAGRAM.
  - DIMENSION ADDED TO CCS2 AND VAS HEATSINK.
  - DIMENSION ADDED TO PRE-DRIVERS HEATSINK.
  - DIMENSION ADDED TO EF3-3 DRIVERS HEATSINK.
  - DIMENSION ADDED AND UPDATED ON THE EF3-4 DRIVERS HEATSINK.
- 28/08/2022 – REV 22
  - POINT 18. PART H & I UPDATED.
- 31/08/2022 – REV 23
  - POINT 18. PARTS A TO K UPDATED TO IMPROVE CLARITY.
- 08/09/2022 – REV 24
  - POINT 3. DETAILS ON THE CORRECT FEEDBACK PATH WERE UPDATED.
- 02/12/2022 – REV 25
  - POINT 3. J101 & J102 JUMPER LOCATIONS UPDATED FOR EF3-3 & EF3-4.

  THE J101 & J102 JUMPERS FOR THE EF3-3 PCB MUST BE INSTALLED ON THE UNDERSIDE OF THE PCB TO PREVENT A POSSIBLE SHORT WITH THE DRIVER HEATSINK.
- 27/12/2022 – REV 26
  - POINT 5. PART III. RESISTORS THAT SERVE AS TEST POINTS ARE NOW LISTED.
  - THE IMAGE AND NOTES WERE UPDATED IN THE "SETUP FOR INITIAL TESTING" SECTION.
  - POINT 14. PART H UPDATED TO INCLUDE THE NEW TEST POINTS TP107 & TP108.
  - POINT 15. PART A UPDATED TO INCLUDE THE NEW TEST POINTS TP107 & TP108.
- 08/01/2023 – REV 27
  - POINT 15. PART H. SPLIT INTO POINTS I. AND II.
  - POINT 15. PART H. II. ADDED TO TEST THE AC VOLTAGE ACROSS THE OUTPUT.
  - POINT 18. PART H. SPLIT INTO POINTS I. AND II.
  - POINT 18. PART H. II. ADDED TO TEST THE AC VOLTAGE ACROSS THE OUTPUT.
- 28/03/2023 – REV 28
  - DOCUMENT REFORMATTED
- 31/03/2023 – REV 29
  - DOCUMENT MARKUPS REMOVED FROM POINTS. 22.2, 22.3 & 22.4
- 01/04/2023 – REV 30
  - POINT 6.2 CCS1 HEATSINK TRANSISTOR DESIGNATORS UPDATED (Q7 & Q8 WERE SWAPPED)
- 02/04/2023 – REV 31
  - POINT 15.1 ARROWS CORRECTED ON THE FIRST IMAGE.
- 05/04/2023 – REV 32
  - POINT 6.5 EF3-3 DRIVER HEATSINK WAS THE INCORRECT IMAGE.

- 09/04/2023 – REV 33
  - POINT 6.5 LOWER IMAGE Q108 ADDED TO THE IMAGE OF THE ASSEMBLED DRIVER HEATSINK.

- 17/07/2023 – REV 34
  - POINT 6.4 SPELLING CORRECTION
  - POINT 8.1 UPDATED REFERENCE TO Q103
  - POINT 14.2
    - UPDATED WITH NEW VBE MULTIPLIER CONFIGURATION
    - UPDATED IMAGES AND TEXT
  - POINT 6.3 UPDATED REFERENCE TO Q103
  - PG. 1 UPDATED PHOTO SHOWING LATEST VBE CONFIGURATION
  - LAST PG. UPDATED PHOTO SHOWING LATEST VBE CONFIGURATION
- 27/07/2023 – REV 35
  - UPDATE R109 FROM 200R TO 500R WERE APPLICABLE THROUGHOUT THE DOCUMENT
- 08/08/2023 – REV 36
  - POINT 22. TESTING IN ISOLATION. ARROWS ON THE SCHEMATICS SHOWING THE TEST POINTS WERE UPDATED.
- 01/09/2023 – REV 37
  - POINT 6.4. IMAGES UPDATED Q105 & Q106 TEXT WAS INCORRECTLY PLACED.
  - POINT 14.2. IMAGES UPDATED Q105 & Q106 TEXT WAS INCORRECTLY PLACED.
  - POINT 21.1. COMPONENT LAYOUT SILK SCREEN UPDATED TO SHOW THE 2ND GROUP BUY BOARDS.
- 22/09/2023 – REV 38
  - TITLE PAGE, GROUP BUY BOARD REVISIONS AND MATCHING SCHEMATIC REVISION DETAILS ADDED.
  - POINT 8. INSTALLATION ORDER CORRECTED.
  - POINT 6.1.A ADDED. NOTES ON TRANSISTOR INSTALLATION.
- 24/09/2023 – REV 39
  - SECTION 8.1 - TP104 WAS TR104 & TP101 WAS TO101
  - SECTION 8.1 - R101 & R104 ADDED
  - SECTION 16.2 WAS SPLIT INTO 16.2 & 16.3 TAKING ACTUAL MEASURMENTS.
  - SECTION 15.1 EXTRA NOTE AND AN IMAGE WAS ADDED TO ALLOW FOR 30VDC RAIL LOW VOLTAGE START-UP.
  - SECTION 16.1 - POINT VII EXTRA NOTE ADDED TO ALLOW FOR 30VDC RAIL LOW VOLTAGE START-UP.
  - SECTION 22.1 - POINT C PART A. EXTRA NOTE ADDED TO ALLOW FOR 30VDC RAIL LOW VOLTAGE TROUBLESHOOTING.
- 02/10/2023 – REV 40
  - SECTION 8.1 – INSTALLATION INSTRUCTIONS FOR Q107, Q108 & THE DRIVER HEATSINK (IF REQUIRED) WERE ADDED.
- 07/10/2023 – REV 41
  - SECTION 22.1 – POINT 22.1.C.A – ADDED
  - SECTION 2.0 – TRANSFORMER SELECTION ADDED.



Good luck with your build from the Entire Wolverine Team

I would like to make mention and thank the people on this team for all the unpaid time and effort they have put into this project. The names of the members are Keantoken, JJS (Jeremy), Harry3 (Harry), Neilshop (Neil), Johno (John), thompsontechs (Jim), Thimios (Thimios), Brettn56 (Brett), Stuartmp (Stuart), Danieljw (Daniel), Fireanimal (Andy). Without them, this project would not have been completed, researched, and tested to such a high standard.