

Aleph 2 Clone Build Notes

Thank you for participating the Classic Aleph Group Buy!

The idea for this PCB is to bring back the classic Aleph circuit for DIY builders in a flexible and expandable format. This document focuses on building the Aleph 2 Clone circuit as a pair of monoblock amplifiers.

Project Difficulty: **NOVICE** **INTERMEDIATE** **EXPERT**



Questions?

You're probably not alone!

Post your question(s) on the DIYAudio forums.

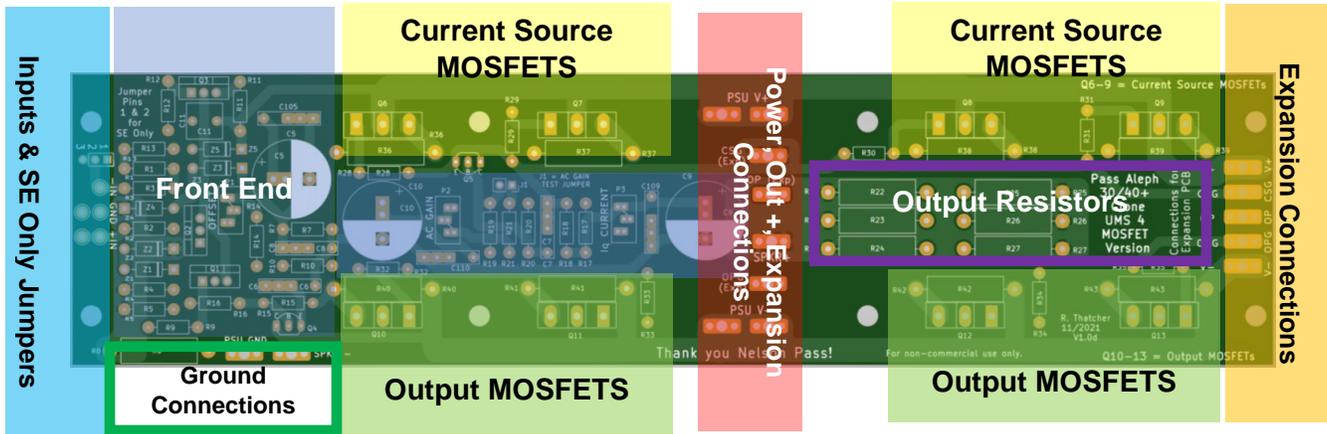


This project uses line/mains voltages and has a power supply with large capacitors. The voltages in this amplifier can kill – even at miniscule current. If you are not competent / confident with working with these voltages, please seek advice from either a qualified electrician, or an audio DIYer who is competent and experienced in this area. Always work safe and work smart!

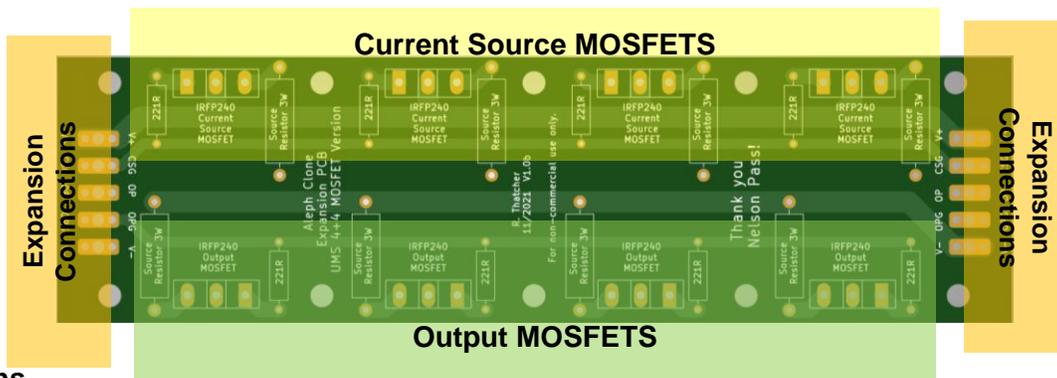
The original schematic has been updated and a new PCB layout devised. The PCBs for this project are offered without any warranty, guarantee provided, or liability taken.

Version / Date	Revision History
V1.0a – 16 May 2022	Original Release – based on Aleph 60 build notes V1.0e with modifications for Aleph 2
V1.0b – 12 April 2022	<ul style="list-style-type: none">• Added pinout diagrams for ZTX450 & 2N4401• Added fuse calculation• Updated transformer notes• Updated BOM based on current stock levels

Getting to know the Main Amp PCB



Getting to know the Expansion PCB



PCB Revisions

PCB	Version / Date	Revision History
Main Aleph PCB	V1.0d – 11/2021	Group Buy #1 Version <ul style="list-style-type: none"> Q4 & Q5 CBE / EBC markings are backwards on PCB.
Main Aleph PCB	V1.0f – 3/2022	Group Buy #2 Version Q4 & Q5 corrected CBE / EBC markings on PCB Added C20 & C21
Expansion PCB	V1.0b – 11/2021	Group Buy #1 & #2 Version

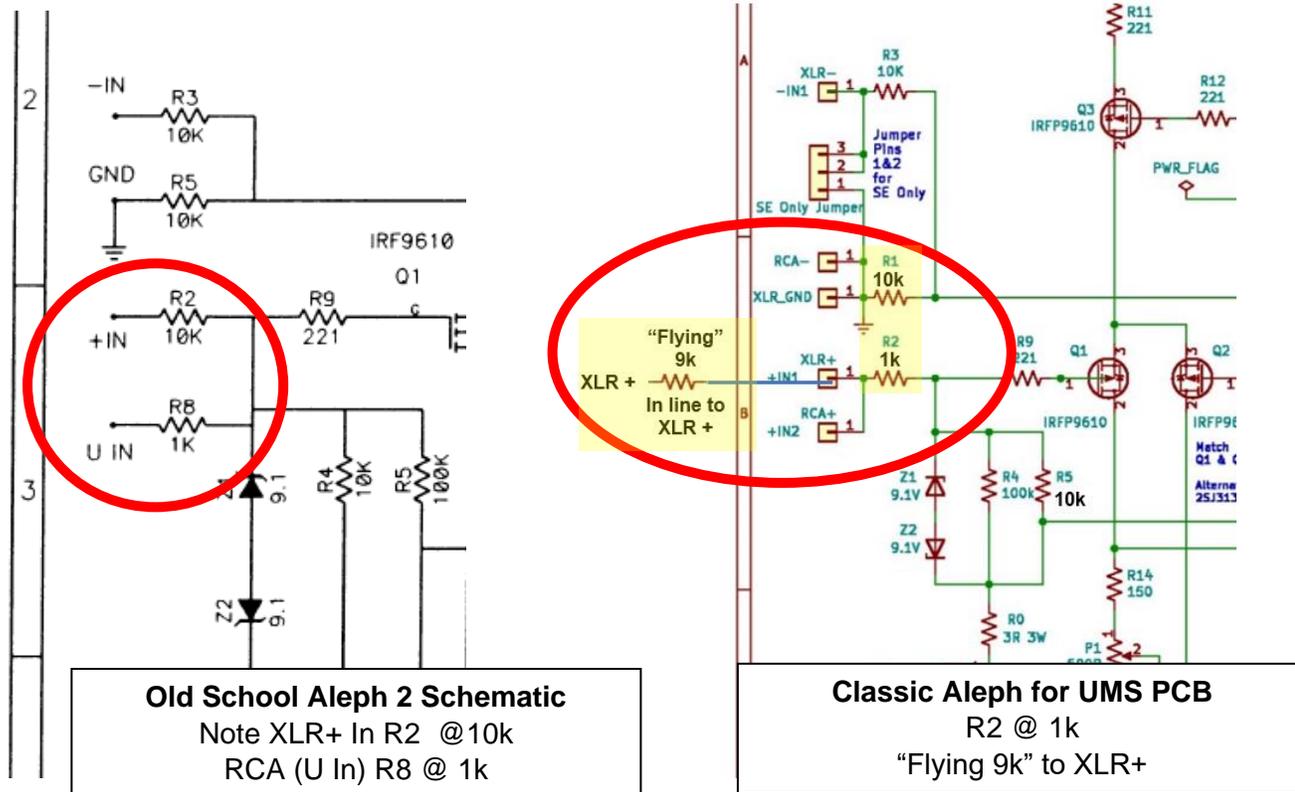
Resistors

Resistors are either 1/4W or 3W rated.

- For 1/4W resistors use your favorite brand metal film resistors. Yaego, etc.
 - Higher wattage is OK. Less is not OK
 - Hole spacing on PCB is 10mm for 1/4 W resistors.
- For 3W MOSFET Source Resistors and Output resistors use 3W metal oxide, and avoid wire wounds unless non-inductive.
- Solder all 3W resistors on amp PCBs and Power supply PCBs so they are elevated from the PCB. Put a spacer underneath them (like a piece of cardboard) to create an air gap of a few mm, and remove it after soldering them in place.
- For Ground / Hum Breaker resistor (R0), any brand 3W Metal Oxide resistor will work. Value is not critical, 2R7 to 4R will be fine.

Aleph 2 – Input Resistors for XLR & RCA

- Aleph 2 (and Aleph 5) circuits have different resistors for XLR In (R2 @ 10k) and RCA In (R8 @ 1k).
- This “Classic Aleph for UMS” PCB is based on the Aleph 30 / 60 circuit which uses a single resistor for both inputs.
- To adapt this PCB to the “classic” Aleph 2 values.
 - Install 1k in position R2
 - Install a 9k resistor “flying” / in-line with the XLR+ wire from the XLR terminal to the PCB input.
 - Diagram on the left (black and white) is the “old school” Aleph 2 input schematic.
 - Diagram on the right is the “Classic Aleph” PCB schematic with values shown for Aleph 2 with the “Flying 9k” shown. Classic SAAB lovers can take special joy in this “Flying 9k” option.



Trim pots & Adjustments

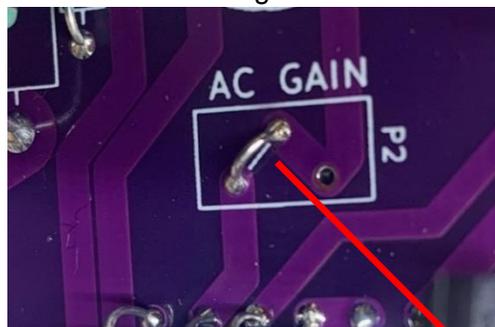
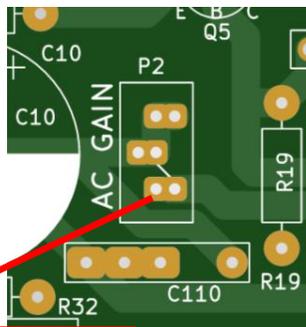
Trimpot positions are included for adjusting output DC Offset, AC Gain, and bias current. In the original circuit these were fixed resistors. It is recommended to install the trim pots to make easy adjustments to your build. The values below are chosen so that midrange of the trimpot value (factory default) + associated resistor equals approximately the original Aleph 2 resistor value. This allows for range for adjustment.

BUILD NOTE: Set trimpot prior to installation

Parameter	Original Aleph 2 Value	Trimpot + Resistor	Initial Trimpot value Set before Installing
DC Offset	R14 = 392R	P1 @ 242R + R14 @ 150R = 392R	P1 = 242R
AC Gain	R21 = 681R	P2 @ 481R + R21 @ 221R = 681R	P2 = 460R
Iq Current	R19 = 56k2	P3 @ 26k2 + R19 @ 30k = 56k2	P3 = 26k2

Optional – Build with fixed resistors and no trim pots

You can use the original base Aleph 2 resistor values in R14, R19, and R21. You will need to jumper the trim pots by soldering in a clipped resistor lead from your discard pile as a jumper wire between the trimpot pads on the PCB. You must use the correct base resistor value (R14 = 392R, R21 = 681R, R19 = 56k2), not the values in the BOM. The PCB is marked with a white line showing where to install the jumper.



Install Jumper here.
See the white mark on the PCB for guidance.

Example hardwired jumper on a prototype build

Bias Current Measurement

Option 1 – Measure voltage of each source resistor (R36-39 + Current Source MOSFET source resistors on Expansion PCB and R40-43 + Output MOSFET source resistors on Expansion PCB), divide by source resistor values. Add them up. Example: R36: $500\text{mV} / 1\text{R} = 0.5\text{A}$. Measure the rest and add up the values.

Option 2 – use a clamp ammeter on positive and negative voltage rail wires from power supply.

Initial target value is 3.0A

Use Option 1 to verify the current is spread evenly on the MOSFETS. Once verified, Option 2 is good for overall bias adjustments.

AC Gain Setting Procedure

See Posts 2 & 3 here for guidance

<https://www.diyaudio.com/forums/pass-labs/38033-proper-current-source-adjustment.html>

Post 3 - From Nelson:

If you set the amplifier driving a sine wave into a load (let's say 16 Vrms into 8 ohms at 100 Hz), you can measure the current variation of the gain N channel Mosfets (whose Sources attach through power resistors to the - supply rail) with a cheap AC voltmeter placed across one of these Source resistors.

With R21 taken out of the circuit, you will get one AC value across the Source resistor (say 470 mV, for example). As you put a value for R21 in the circuit, this will decline, and when it measures ½ the value without R21, you have reached 50%.

If it measures 1/4 the value, the current gain of the Aleph source is 75%, and this figure is too high for a standard Aleph. Most listeners like the Alephs at 50% or lower, so I recommend between 50% and 100% of the AC voltage value compared with no R21.

NOTE: The PCB has an AC Gain Setting jumper that allows for pulling R21 out of circuit temporarily by removing the jumper.

Step	Procedure	Measurements	
		Left Channel	Right Channel
1	Remove AC Gain Jumper		
2	Set the amplifier driving a sine wave into a dummy load (let's say 16 Vrms into 8 ohms at 100 Hz).	V = _____ into 8R Freq = _____ Hz	V = _____ into 8R Freq = _____ Hz
3	Measure voltage on source resistors on Amp / V- Side (R40-43) Set meter to read AC Volts. Confirm it's AC, not DC.	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV
4	Calculate 50% of voltages in Step 3	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV
5	Install AC Gain Jumper		
6	Set the amplifier driving a sine wave into a load (let's say 16 Vrms into 8 ohms at 100 Hz)	V = _____ into 8R Freq = _____ Hz	V = _____ into 8R Freq = _____ Hz
7	Measure Source resistors on Amp / V- Side. Adjust P2 to match Step 4 values Set meter to read AC Volts. Is it set for AC, not DC?	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV	R40 = _____ mV R41 = _____ mV R42 = _____ mV R43 = _____ mV

Capacitors

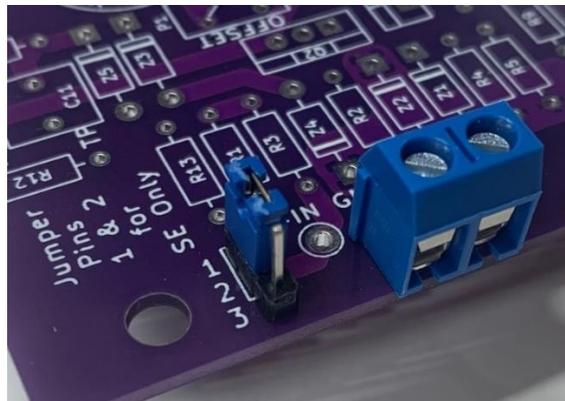
Positions	Value	Size & Notes
C5, C9, C10	220uF 27+V	5.0 or 7.5mm Lead Spacing, up to 16mm Diameter Use your favorite brand here
C105, C109, C110	0.1uF Film Cap	OPTIONAL - Bypass for each of the above Electrolytic Caps 5.0, 7.5, or 10.0 mm Lead Spacing
C8	10pF Mica	5.0, 7.5, or 10.0 mm Lead Spacing
C6, C7	1nF / 1000pf / 0.001uF	5.0, 7.5, or 10.0 mm Lead Spacing
C11	1.0uF to 4.7uF will do, value is not critical	5mm LS 7.2x7.2 Film
C20, C21	1,000 uF or greater 50V or greater	5.0 or 7.5mm Lead Spacing, up to 16mm Diameter Use your favorite brand here

Minimum DC Voltage rating for capacitors should at minimum be rail voltage. Greater is OK, lower is NOT OK. 50V is a standard value that would work.

The exception is C5/105, C9/109, C10/110. For these caps the minimum Voltage rating = $\frac{1}{2}$ rail voltage + 4V. $45V / 2 + 4V = 26.5V$. Therefore >27V for Aleph 2. >27V or higher voltage rating is OK. Lower is not OK. 35V is a standard value that would work.

Connections to PCB

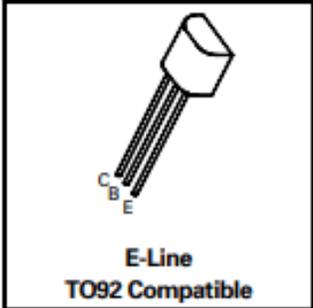
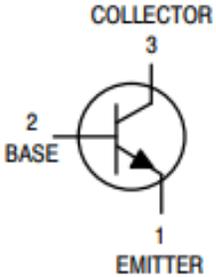
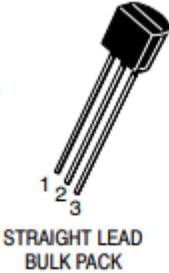
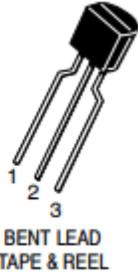
You have several options for wire connections to the board including Quick Disconnect Spades, bare wire, or Euroblock type 5mm / 5.08 mm connectors.



Semiconductors

The Aleph 2 clone will use 3 of the 4 MOSFET positions on the top and bottom of the main PCB and 3 of the 4 MOSFET positions on the top and bottom of the Expansion PCB. There will be a total of 24 IRFP240's in 2 channels of Aleph 2 configuration.

Positions	Semiconductors	Matching & Notes
Q1 & Q2	IRF9610 or SFP9610 Alternate: 2SJ313 Alternate: FQP3P20	Must be matched
Q3	IRF9610 or SFP9610 Alternate: FQP3P20	No need to match
Q4 & Q5	ZTX450 Alternate 2N4401	No need to match NOTE for PCB version 1.0d: Q4 shows "C B E" on PCB rev V1.0d. This is backwards, it should read "E B C". For ZTX 450, refer to the shape outline on the board. NOTE for PCB version 1.0d: Q5 shows "E B C" on PCB rev V1.0d. This is backwards, it should read "C B E". For ZTX 450, refer to the shape outline on the board. Markings have been corrected on PCB version 1.0f
Q6-Q9	IRFP240	Qty 3. Must be matched Must populate Q6 position. Populate 2 of the 3 remaining positions on PCB and populate associated gate and source resistors. All 6 Current Source MOSFETs per channel must be matched. Therefore Current Source MOSFETs on Main PCB must be matched with Current Source MOSFETs Expansion PCB.
Q10-Q13	IRFP240	Qty 3. Must be matched Must populate Q10 position. Populate 2 of the 3 remaining positions on PCB and populate associated gate and source resistors. All 6 Output MOSFETs per channel must be matched. Therefore Output MOSFETs on Main PCB must be matched with Output MOSFETs Expansion PCB.
Expansion PCB – Current Source MOSFETs	IRFP240	Qty 3. Must be matched – see notes above about matching. Populate 3 of the 4 positions on PCB and populate associated gate and source resistors.
Expansion PCB – Output MOSFETs	IRFP240	Qty 3. Must be matched – see notes above about matching. Populate 3 of the 4 positions on PCB and populate associated gate and source resistors.

ZTX450 Pinout	2N4401 Pinout
 <p>E-Line TO92 Compatible</p>	 <p>TO-92 CASE 29 STYLE 1</p>  <p>STRAIGHT LEAD BULK PACK</p>  <p>BENT LEAD TAPE & REEL AMMO PACK</p>

Distributing MOSFETs on the Heatsinks

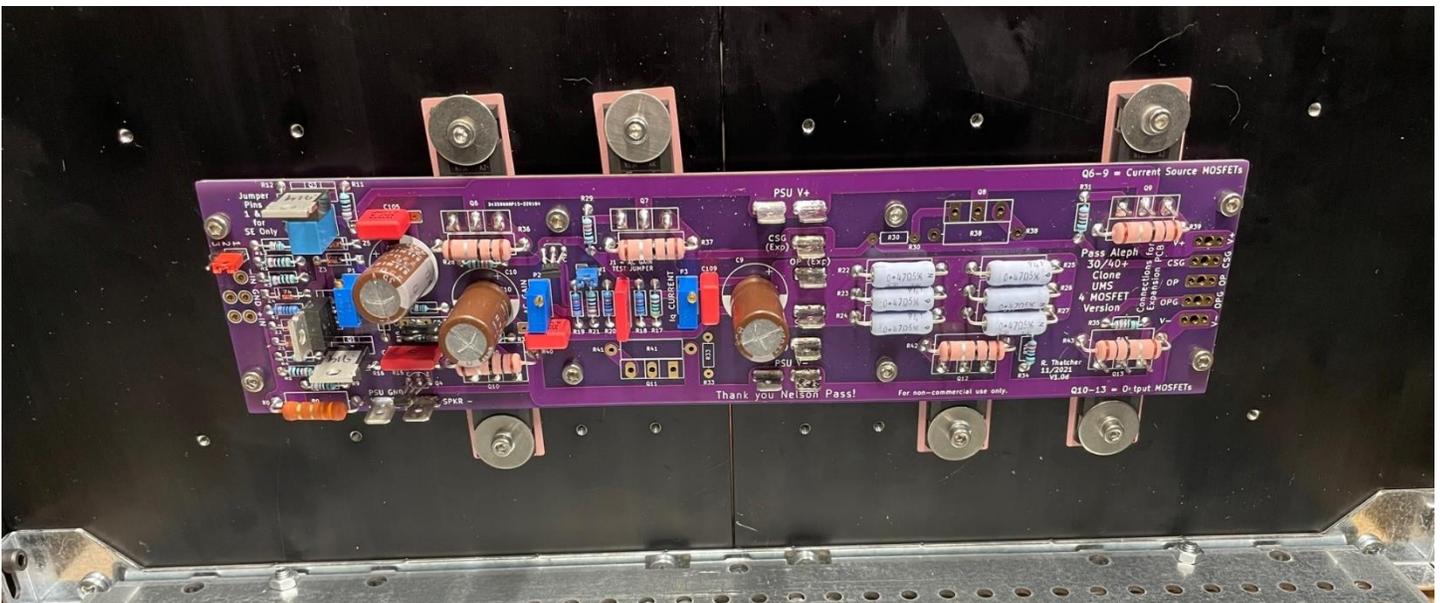
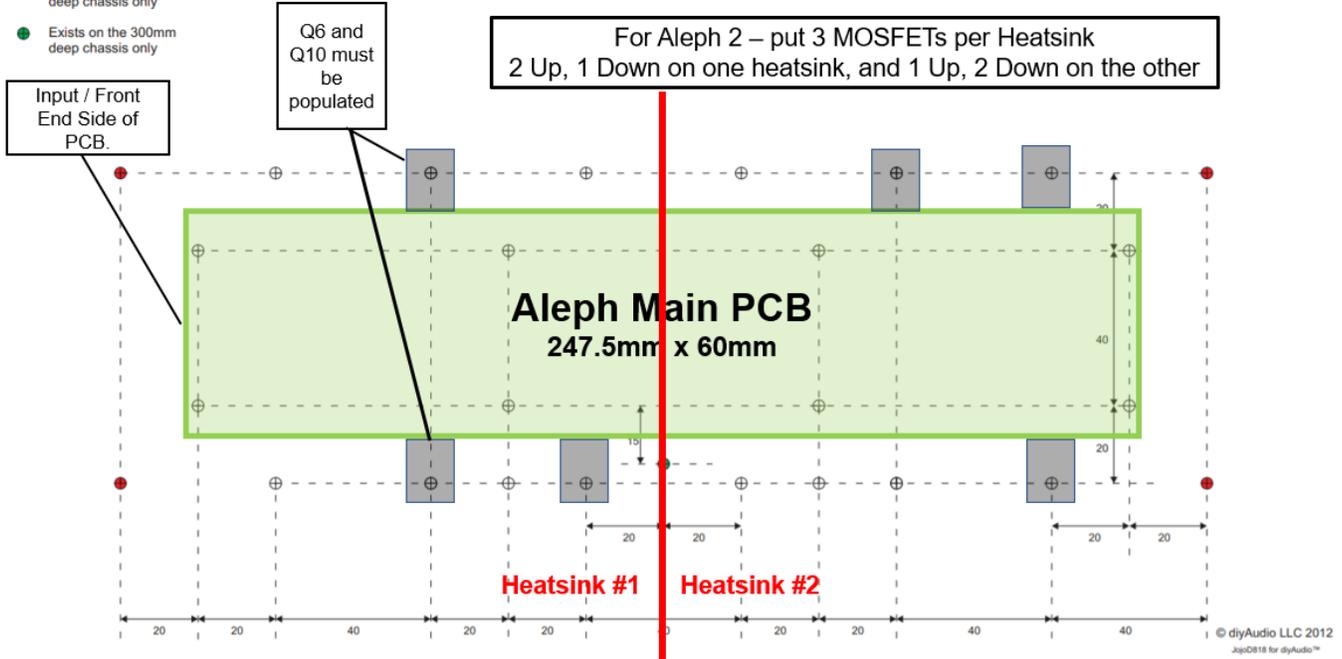
The following notes assume the use of a 5U Deluxe Chassis

The 5U Deluxe chassis uses 2 heatsinks per side. Arrange MOSFETs on the main PCB and Expansion PCBs so that each heatsink has 3 MOSFETs each. Here are examples of possible arrangements.

Scale = 1:1
All units in millimeters (mm)

- Exists on the 400mm deep chassis only
- Exists on the 300mm deep chassis only

This specification makes it easy to experiment with different amplifier designs using the same chassis and a universal power supply. Transistors mount using M3 hex socket bolts and the boards with M3 standoffs. Some holes serve a dual purpose and for one board may be a board mounting hole and for a different board the hole may be used for transistor mounting, depending on the board design and orientation. We hope that members developing their own PCB's will use this standard so all members will benefit.

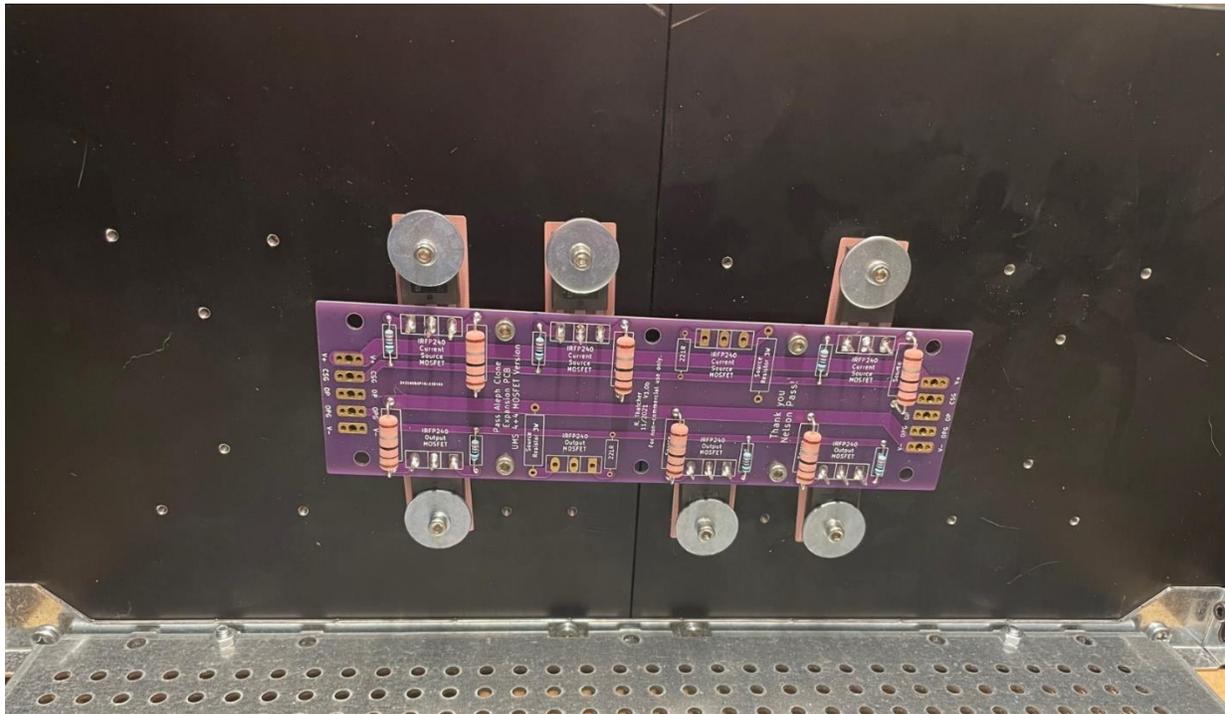
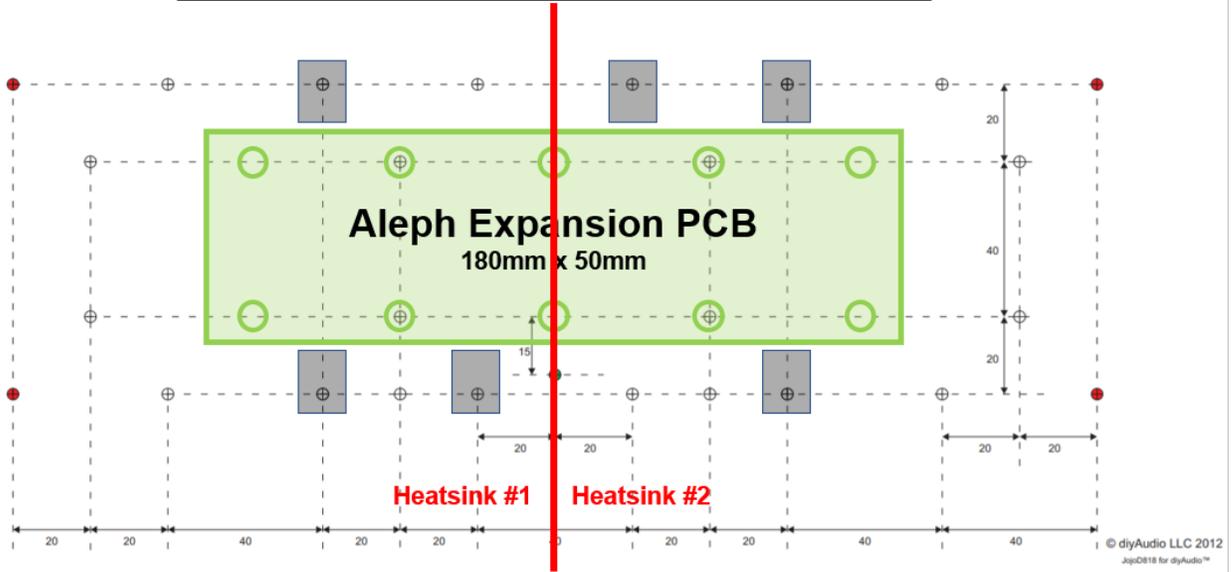


Scale = 1:1
All units in millimeters (mm)

- Exists on the 400mm deep chassis only
- Exists on the 300mm deep chassis only

This specification makes it easy to experiment with different amplifier designs using the same chassis and a universal power supply. Transistors mount using M3 hex socket bolts and the boards with M3 standoffs. Some holes serve a dual purpose and for one board may be a board mounting hole and for a different board the hole may be used for transistor mounting, depending on the board design and orientation. We hope that members developing their own PCB's will use this standard so all members will benefit.

For Aleph 2 – put 3 MOSFETs per Heatsink
2 Up, 1 Down on one heatsink, and 1 Up, 2 Down on the other



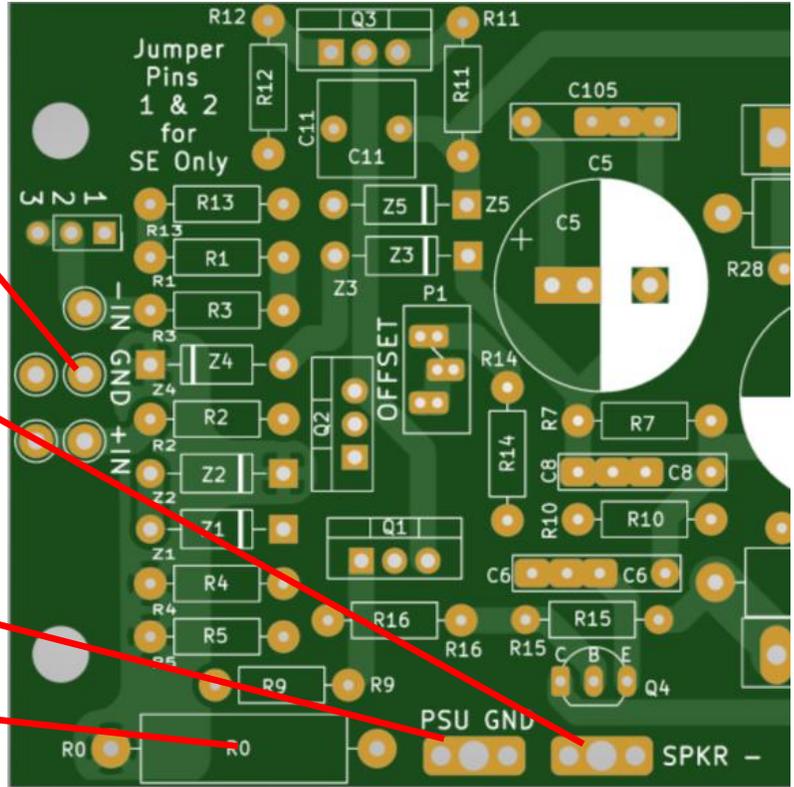
Grounding

Input Signal Ground(s) attach here.

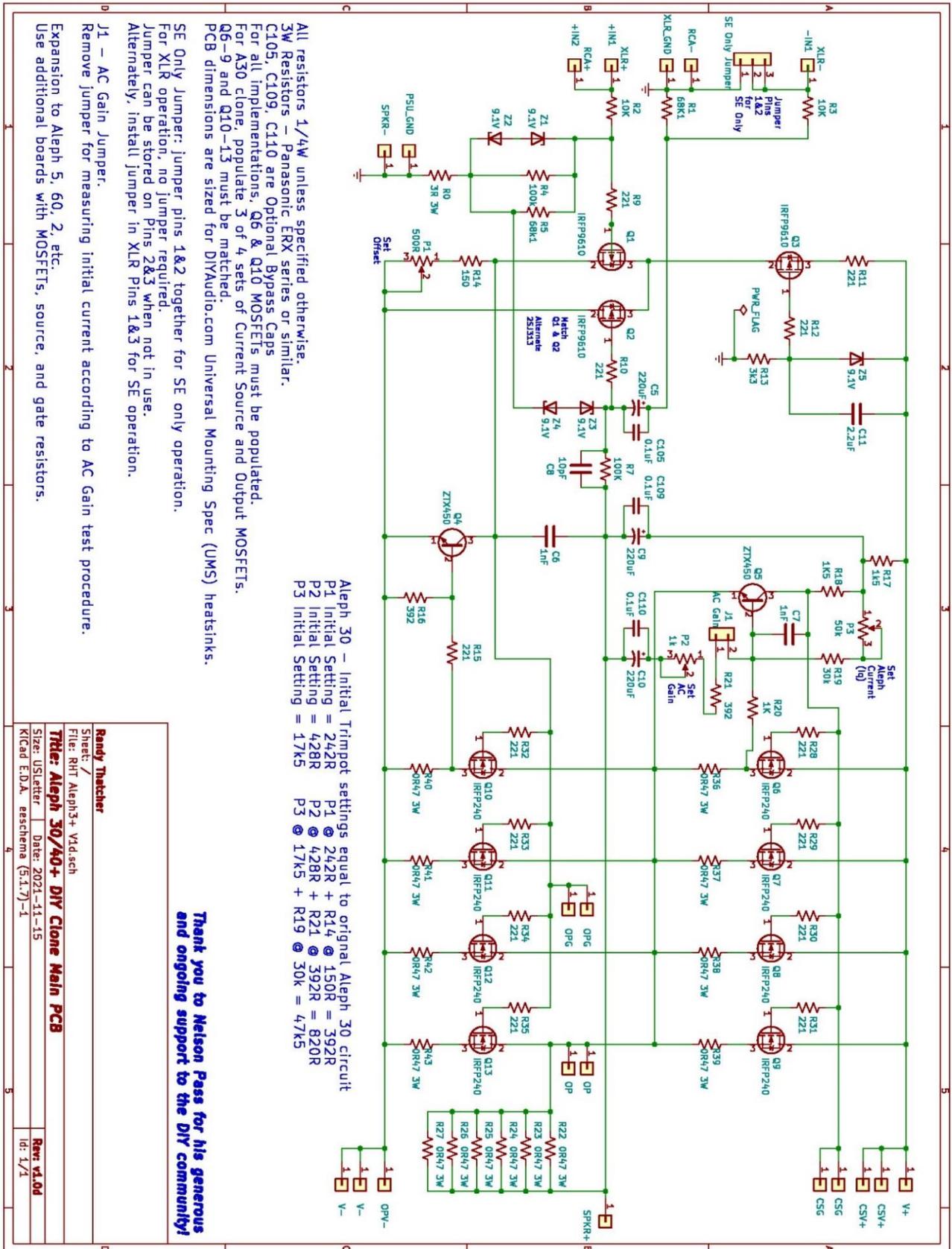
Option 1 – Connect Power supply / star ground to Negative Speaker Terminal.
Option 2 – Connect from here to Negative Speaker Terminal.
See which option sounds better in your setup

Power Supply Ground attaches here.

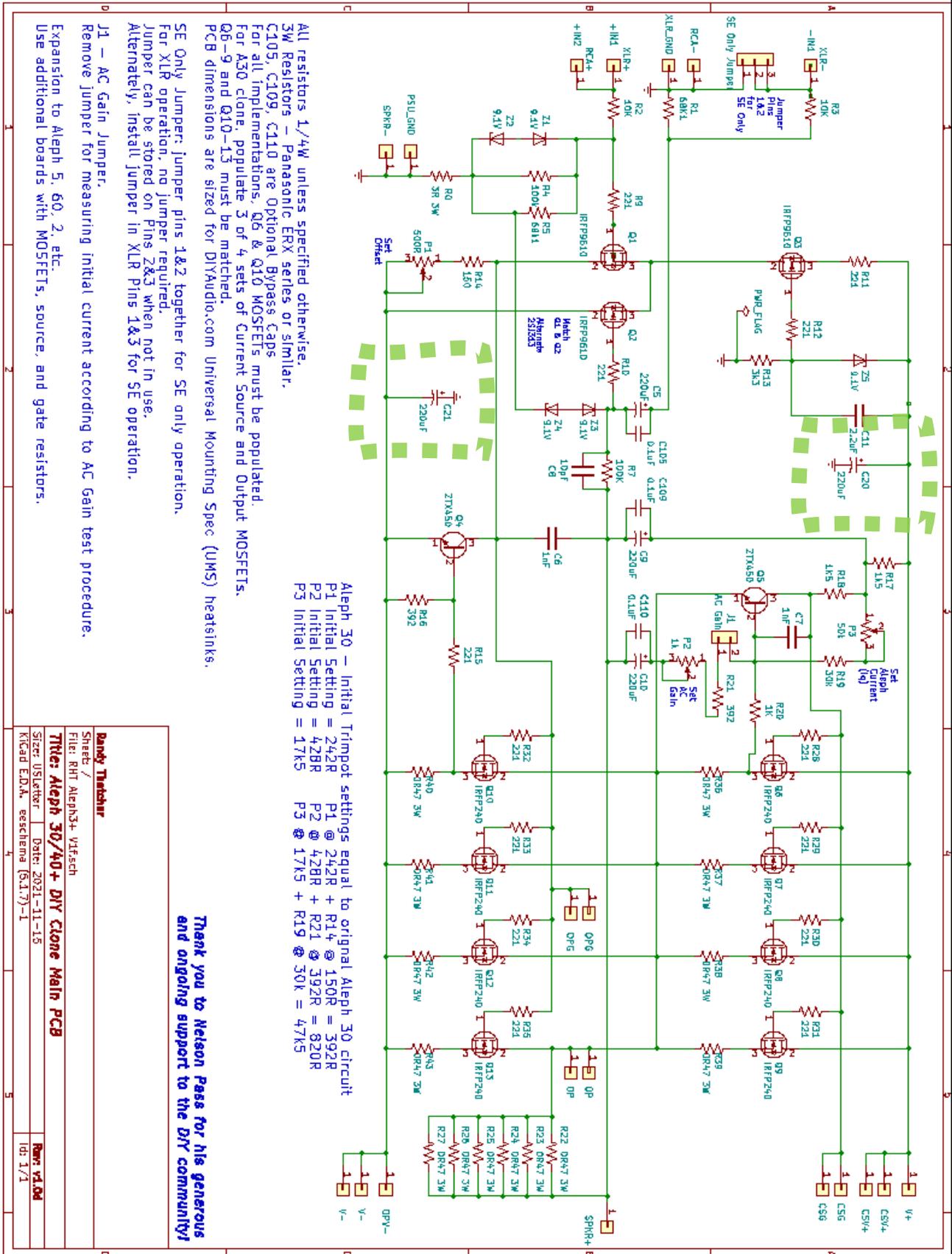
R0 = hum breaker resistor



PCB Schematic – Main PCB – Version 1.0d



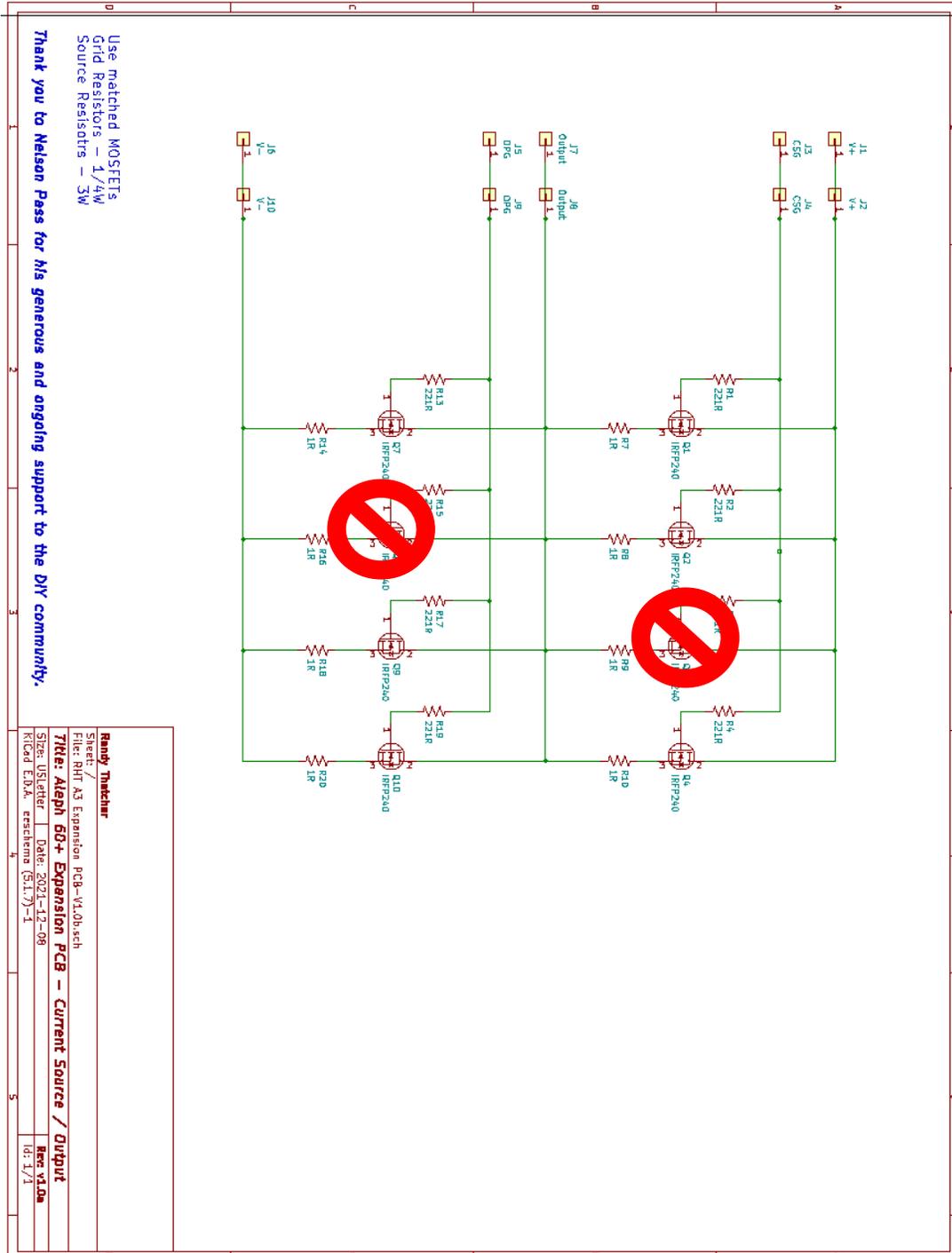
PCB Schematic – Main PCB – Version 1.0f



Dandy Thatcher	
Sheet: /	Files: RHT Aleph3+ v1f6ch
Title: Aleph 30/40+ DIY Clone Main PCB	
Sheet: USLetter	Date: 2021-11-15
Kicad: EDA, eeschema (5.1.7)-1	Rev: v1.0f
	Id: 1/1

Expansion PCB Schematic

Aleph 2: populate 3 of the 4 positions for Output and Current Source MOSFETs and associated resistors. The “do not populate” markings below are staggered to distribute the MOSFETS evenly on the heatsink(s)



Aleph 2 BOM

BOM for Aleph 2 Main & Expansion- The quantities below are for 6 MOSFET output (Aleph 2)

BOM Version: v1.0a **NOTE: This BOM is based on the old school Aleph 2 schematic + Aleph reloaded mods. The PCB schematic above shows Aleph 30 values. Use values from the BOM below for your Aleph 2 build.**

This table contains example part numbers and part recommendations. Any good quality similar parts will work with no detriment to the sound.

Per Chan	2 Chan	PCB Designation	Description	Add'l Detail / Comment	Digikey
1	2	R0	2R7-4R 3W		A131577CT-ND
3	6	R1, R3, R5	10k 1/4W		10.0KXBK-ND
2	4	R2, R20	1k 1/4W		1.00KXBK-ND
1	2	Install 9k "flying" in line with IN+ from XLR	9k 1/4W	Old school Aleph 2 schematic uses different resistor for XLR vs RCA Input. Current PCB was designed around Aleph 30/60.	
2	4	R4, R7	100k 1/4W		100KXBK-ND
18	36	R9-12, R15, R21, R28-35 + Exp. PCB gate Rs	221R 1/4W	Refer to the "Distributing MOSFETs on heatsinks" section above when placing R28-35 and expansion PCB resistors.	221XBK-ND
1	2	R13	3k3 1/4W		MFR-25FBF52-3K3-ND
2	4	R14, R16	150R 1/4W		150XBK-ND
1	2	R17	4k75 1/4W		
1	2	R18	1k5 1/4W		RNF14FTD1K50CT-ND
1	2	R19	30k 1/4W		MFR-25FBF52-30K-ND
6	12	R22-27	0R47 3W	Panasonic ERX or similar Metal Oxide	A138094CT-ND
12	24	R36-43 + Exp. PCB source Rs	1R 3W	Panasonic ERX or similar Metal Oxide	A138411CT-ND 13-FMP300FRF73-1RCT-ND
5	10	Z1-5	9.1V Zener		1N5239B-ND
3	6	Q1-3	IRF9610	Q1&2 Matched	IRF9610PBF-ND
3	6	Q1-3 Alternate Option	SFP9610	Q1&2 Matched	
3	6	Q1-3 Alternate Option	FQP3P20	Q1&2 Matched	
2	4	Q1-2 Alternate Option	2SJ313	Q1&2 Matched	
2	4	Q4-5	ZTX450		ZTX450-ND
12	24	Q6-13 + Exp PCB MOSFETs	IRFP240	6 x Current Source MOSFETs matched – per channel 6 x Output MOSFETs matched – per channel Total of 4 matched sextets per 2 channel amp	IRFP240PBF-ND
3	6	C5, C9, C10	220uF 35V	5.0 or 7.5mm Lead Spacing, up to 16mm Diameter	604-1066-ND
3	6	C105, C109, C110 (Optional)	0.1uF Film Cap	5.0, 7.5, or 10.0 mm Lead Spacing	1928-1538-ND
1	2	C8	10pF Mica	5.0, 7.5, or 10.0 mm Lead Spacing	338-1061-ND
2	4	C6, C7	1nF / 1000pf / 0.001uF	5.0, 7.5, or 10.0 mm Lead Spacing	1928-1384-ND
1	2	C11	1.0uF to 4.7uF will do, value not critical	5mm LS 7.2x7.2 Film	399-12660-ND 495-1127-ND
2	4	C20, C21	1,000uF or more, 50V	5.0 or 7.5mm Lead Spacing, up to 16mm Diameter (PCB V1.0f only)	493-4612-ND 493-10898-ND 493-4613-ND
1	2	P1	500R Multi-turn pot	Initial Setting = 242R	3296Y-501LF-ND or 3296W-501LF-ND
1	2	P2	1k Multi-turn pot	Initial Setting = 460R	3296W-1-102RLFCT-ND 490-2874-ND
1	2	P3	50k Multi-turn pot	Initial Setting = 26k2	3296Y-503LF-ND or 3296W-503LF-ND
2	4	Jumpers	2.54mm jumper		1849-09200-71-BBGB00-ND
1	1	Jumper Header	2.54mm jumper header pins	Buy a long strip and cut for J1 and SE Only Jumpers	2057-PH1-15-UA-ND
12	24	Heatsink pads	Pads for MOSFETS		Keratherm (DIYAudio Store)
1	2	Main PCB			Group Buy
1	2	Expansion PCB			Group Buy
		OPTIONAL: QD's		Quick Disconnect Spades	36-1287-ST-ND
		OPTIONAL: QD Connectors	3 Position Terminal Block	Screw disconnect for inputs	A98077-ND
		OPTIONAL: QD Connectors	5 Position Terminal Block	Screw disconnect for Expansion connectors For use in Aleph 2 Builds	23-0395443005-ND

Chassis

The DIY Audio 5U Deluxe Chassis is a good candidate for the Aleph 2 Mono Amplifier. A 500mm deep chassis is also a good candidate. The 500mm Deep chassis has more heatsink than the 400mm deep Deluxe chassis, but the 500 deep version will require drilling and tapping the heatsinks. It is NOT "UMS" compliant.

Power Supply

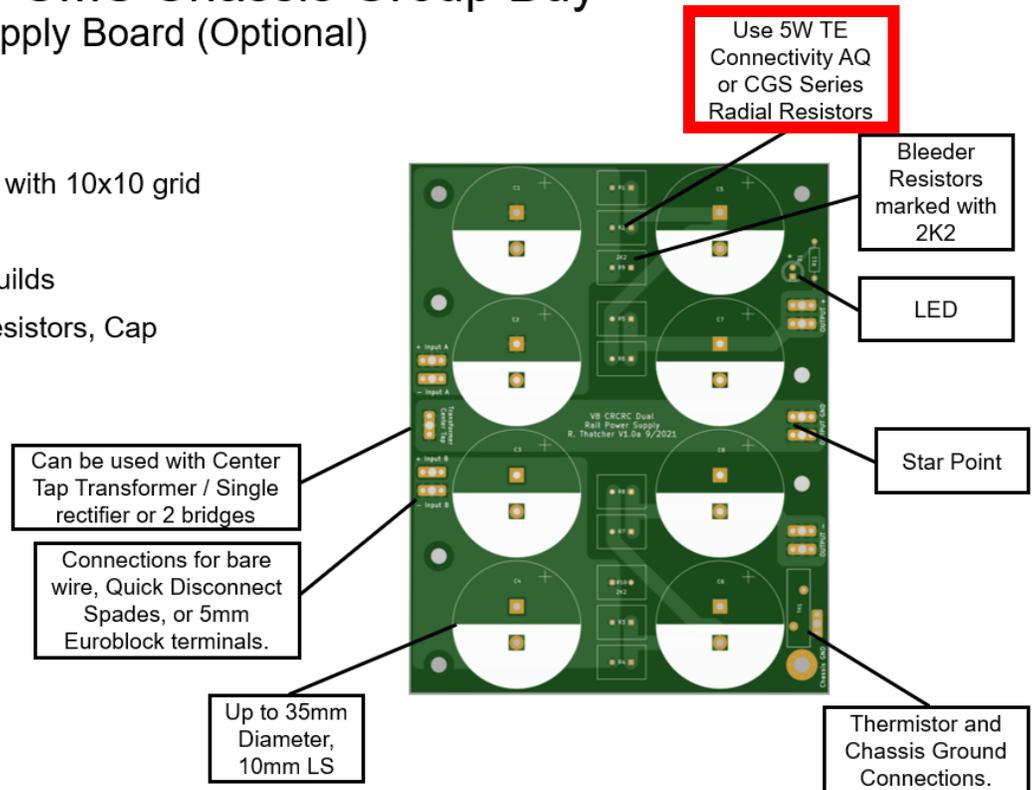
Transformer

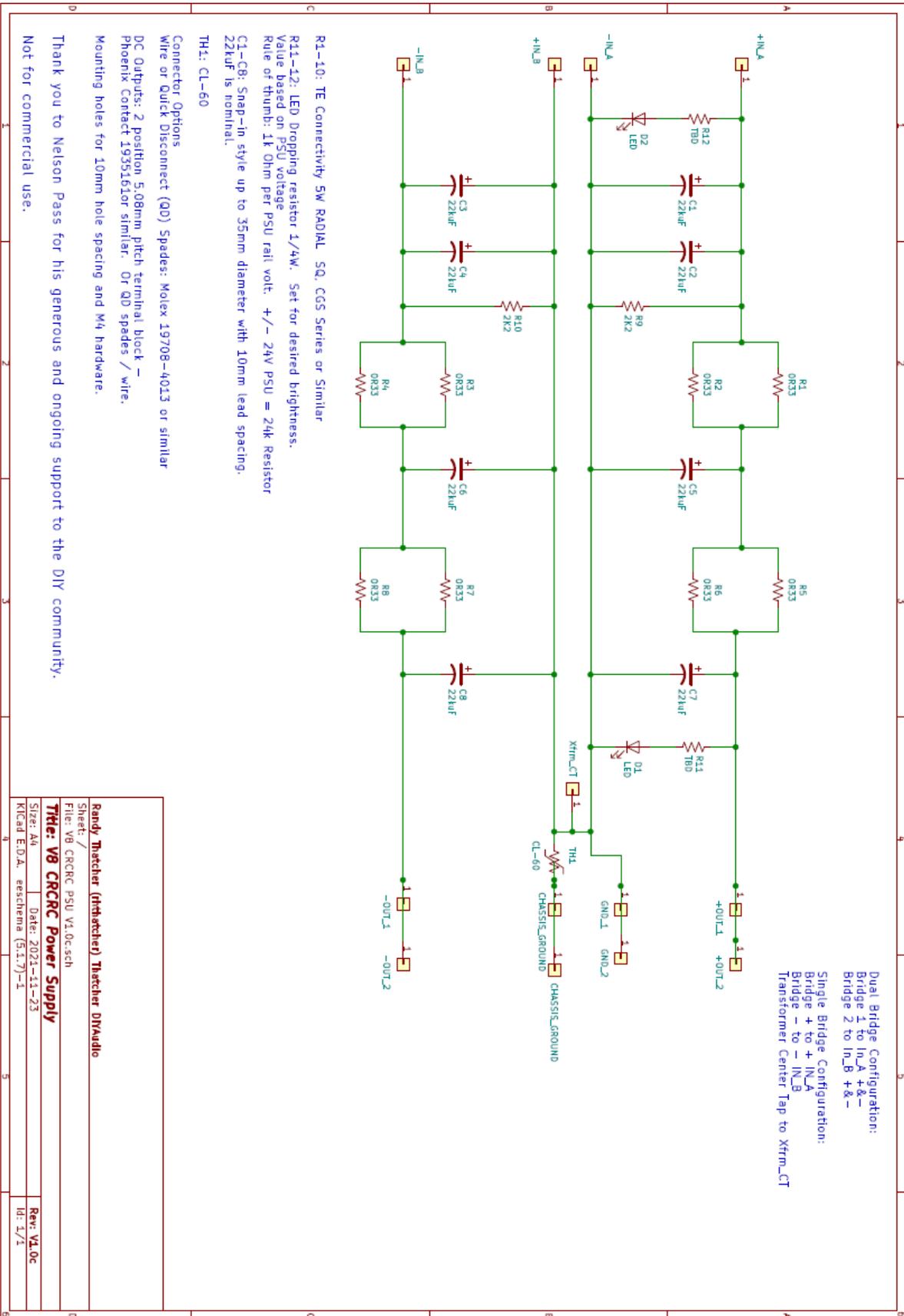
Per Monoblock: Recommend 600VA or higher Transformer (example: Antek AN-6435)
35+35V secondaries
Or alternate transformer with 600VA or higher with 37+37V secondaries

V8 CRCRC Power Supply – Recommended for Mono builds

Classic Aleph for UMS Chassis Group Buy V8 CRCRC Power Supply Board (Optional)

- Mounting holes compatible with 10x10 grid
- 115 x 146mm
- Recommended for mono builds
- 2 Caps, Resistors, Cap, Resistors, Cap





Power Supply BOM for V8 CRCRC Power Supply PCB

NOTE: BOM is for SINGLE CHANNEL. Multiply Quantity x 2 for 2 channels!

This table contains example part numbers and part recommendations. Any good quality similar parts will work with no detriment to the sound.

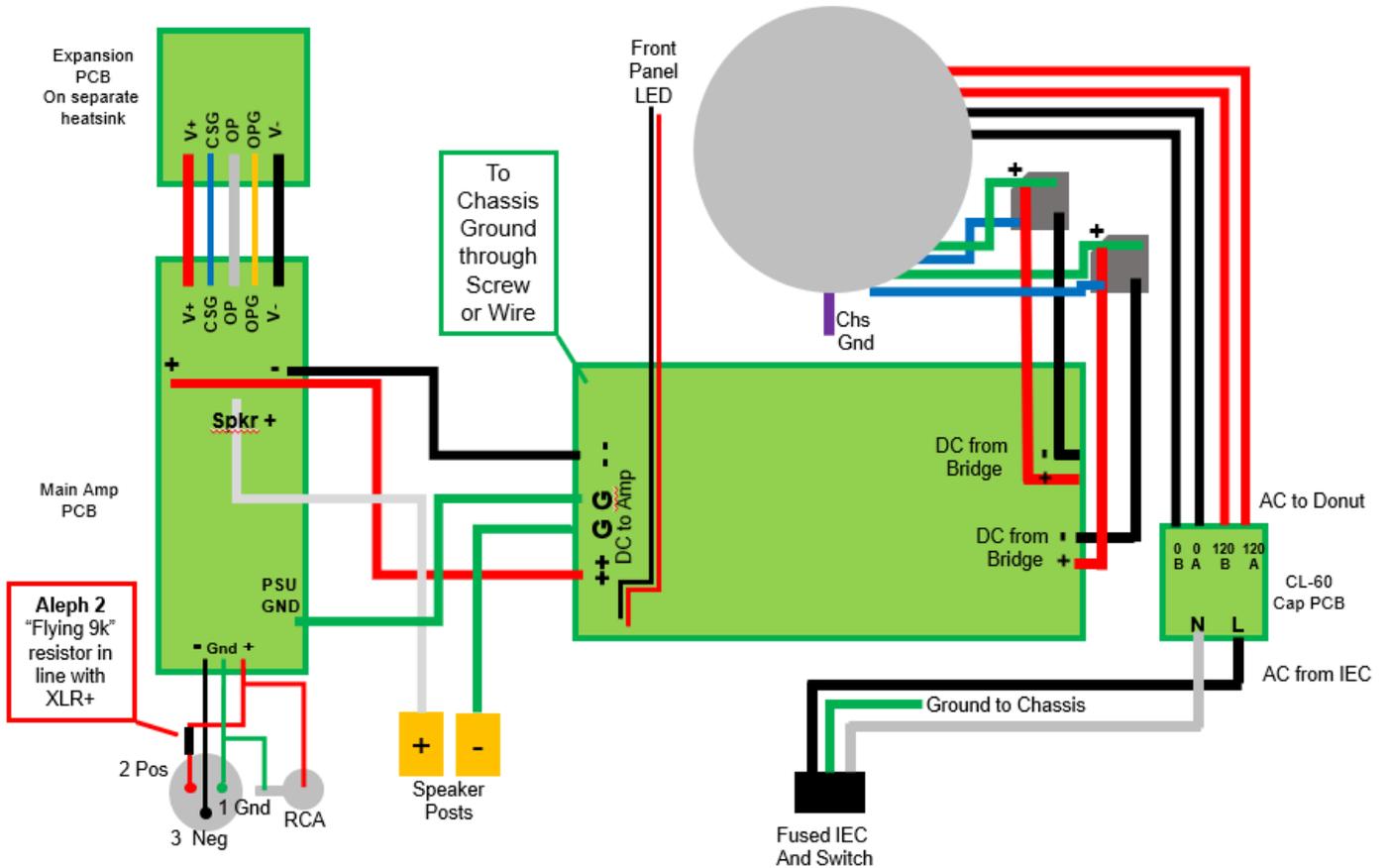
ID	Qty	Value	Digikey Part Number	Comment
Power Supply Board				
R1-8	8	0R22 – 0R33 5W	A103692-ND A137379-ND A102472-ND	Use TE Connectivity 5W Radial Resistors for this PCB 13.00mm x 9.00mm 5mm Lead Spacing
R9-10	2	2k2 5W (3k3 is also OK)	A102461-ND A131251-ND	Use TE Connectivity 5W Radial Resistors for this PCB 13.00mm x 9.00mm 5mm Lead Spacing
R11, 12	2	47k 1/4W		"Rule of Thumb" - 1k Ohm per PSU volt Increase R for dimmer LED.
C1-8		22k uF, 50V	338-1599-ND 338-2254-ND	10mm Lead Spacing, up to 35mm Diameter. Voltage rating must be greater than rail voltage!
C1-8 (alternate)		27k uF, 50V		
C1-8 (alternate)		33k uF, 50V		
TH1	1	CL-60	KC006L-ND	
D1, 2	2	Blue LED	732-5019-ND	This is a Pass clone – blue is required! LEDs are both on positive rail. Use one for on-board and another for front panel
Other	5	Screw Terminal Blocks 2 position	277-1667-ND	OPTIONAL - For connection to amp PCB and/or Bridges
Other	10	Quick Disconnect Blades	WM14275CT-ND 36-1287-ST-ND	
CL-60 / AC Cap PCB				
C1	1	3300pF, X1 Safety Rated	399-9513-1-ND	
TH1-2	2	CL-30		
Other	3	Screw Terminal Blocks 2 position	277-1667-ND	
Other	6	Quick Disconnect Blades	WM14275CT-ND 36-1287-ST-ND	
Rectifiers / Snubber PCBs				
Rectifier Bridges	2		GBPC3510-E4/51GI- ND 641-1380-ND	
Snubber C	2	FILM 10000PF / 10nF / .01uF	495-4975-1-ND	
Snubber C	2	FILM 150nF / .15uF	495-77011-1-ND	
Snubber R	2	Metal Film 1/4W - Value TBD		Use Quasimodo test jig to determine value 20R is typical for Antek in dual secondary configuration.
MISC				
Fuses	TBD	TBD Amp Slow Blow		Calculate Fuse size based on Transformer VA / Mains voltage, then go to next standard size. Example: 600VA / 120V = 5A Fuse
Thermal Switch	1	THERMOSTAT 70DEG C SPST-NC	1862-1126-ND	

Amplifier Wiring – Aleph 2 Monoblock Configuration with V8 Power Supply PCB

- Twist wires!!!
- If using shielded transformer, attach shield wire to Chassis.
 - Option 1: Direct to Chassis
 - Option 2: Connect to “Chassis Ground” termination point on PSU PCB
- Confirm transformer wiring pairs (120A / 0 A, 120B / 0B, secondary pairs)
- Expansion PCB Wiring
 - There are connections in middle and end of main PCB to Expansion PCB. Choose the wiring that works best in your chassis configuration
 - There are connection on both ends of Expansion PCB

Expansion Board Connection	Wire
V+ (Positive Rail) OP (Output) V- (Negative Rail)	Fat Wire
CSG (Current Source Gate) OPG (Output Gate)	Thin Wire

Classic Aleph for UMS Chassis Group Buy Aleph 2 Monoblock Wiring using V8 CRCRC Power Supply – 120VAC Mains



Final Test / Checkout Sheet

	Left	Right
Power Supply V+		
Power Supply V-		

	Initial / Cold		Warm (After ~1 Hour)		After Adjustment	
	Left	Right	Left	Right	Left	Right
DC Offset (mV) Target < 100 mV						
Current Source MOSFET 1 current <i>Main PCB</i>						
Current Source MOSFET 2 current <i>Main PCB</i>						
Current Source MOSFET 3 current <i>Main PCB</i>						
Current Source MOSFET 1 current <i>Expansion PCB</i>						
Current Source MOSFET 2 current <i>Expansion PCB</i>						
Current Source MOSFET 3 current <i>Expansion PCB</i>						
Total Current Source Current (Amps) Target = 3.0A						
Output MOSFET 1 current <i>Main PCB</i>						
Output MOSFET 2 current <i>Main PCB</i>						
Output MOSFET 3 current <i>Main PCB</i>						
Output MOSFET 1 current <i>Expansion PCB</i>						
Output MOSFET 2 current <i>Expansion PCB</i>						
Output MOSFET 3 current <i>Expansion PCB</i>						
Total Output Current (Amps) Target = 3.0A						

See AC Gain setting section for testing / recording AC Gain.