

Leach Low TIM 2x200 Watt Amplifier

2U Rack Mount Assembly Guide v1.0

Change History

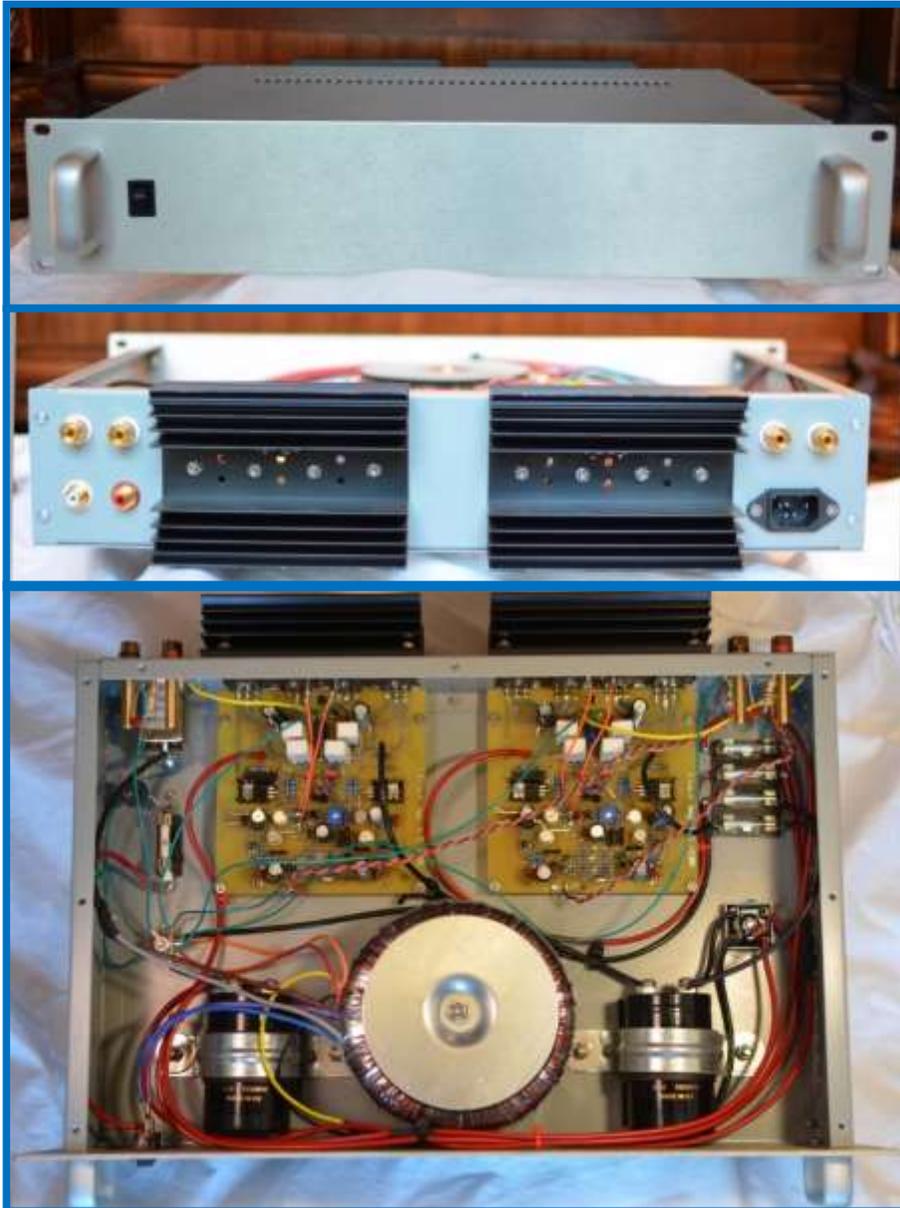
Version	Description	Author	Date
V1.0	Initial Release	Charles Robison	2015-01-04

Contents

1.	Introduction.....	4
1.1	About Myself	5
1.2	About This Guide	5
2.	Disclaimer	6
3.	Ordering the Parts	7
3.1	Main Board	7
3.1.1	Ordering the PCB.....	7
3.1.2	Ordering the Rest of the PCB parts.....	9
3.2	Chassis.....	10
3.3	Building Accessories	10
3.4	Now we wait.....	11
4.	Assembling the PCB.....	11
4.1	Transistor Matching.....	11
4.2	Zener Diode Matching	12
4.3	Assembly of Main Board PCB	13
5.	Assembling the Chassis	16
5.1	Mechanical.....	17
5.1.1	Countersink the Chassis Screws.....	17
5.1.2	The Bottom Panel	17
5.1.3	The Heat Sinks	19
5.1.4	The Front Panel	21
5.1.5	The Rear Panel	21
5.1.6	Connecting the Rear and Bottom Panel	23
5.1.7	The Bias Diodes	27
5.1.8	Assembly of Rear Panel	28
5.1.9	Finishing assembly	29
5.2	Electrical.....	29
5.2.1	AC Power	29
5.2.2	DC Power.....	30
5.2.3	Signals	31
6.	First Power One.....	33
6.1	Testing.....	33
6.2	Setting Bias Current.....	34
6.3	Is there a hum in the loudspeaker	34
6.4	Hints if problems are encountered	35
7.	A Final Word	36
	CITATIONS:	37

1. Introduction

In this guide we are building the Leach Low TIM leach 200Watts x 2 channel Amplifier within a 2U rack mount chassis. It is a two channel amplifier which means it can drive two speakers otherwise known as a stereo amplifier. There is a lot of snake oil sales in the audio industry and if you're like me, I don't mind spending the money on my equipment but I definitely want my money's worth. It's very difficult to tell what you get for your money so this led me to building my own equipment. The amps design is intended to fit the needs of most home audio enthusiasts and packs quite a punch. I've seen a few folks change the original design to add more power transistors or otherwise increase the output power but I believe this is completely unnecessary. I'm driving a huge 15" sub to its limits without any problem using one channel and I've also driven my Klipsh Floor speakers to deafening levels without overdriving the amp. If you want more power out of these amps, use a pre-amp to drive the inputs higher. I use the McIntosh C-15 Pre-amp and it will drive these Leach amps as hard as I need. Here are some pictures of the amp we are building in this guide:



1.1 About Myself

I studied under Dr. Leach at the Georgia Institute of Technology and took nearly every course he offered from 2000 – 2002. In Fact, I built my first low TIM amplifier as course work with his guidance. It was a single channel, boxy design made of clear acrylic and looked absolutely terrible... you know, being built in a dorm room and all. Sometime around 2005 I blew up my amp doing some experiments (not audio related so I never should have done them) and I vowed one day to build a new set of amplifiers. I'm in the process of building 3 amps in the rack form factor so I can use them in my music/theater room and have them fit into a media rack discretely in the corner of the room. I've finished one and partially done with the other two at the time of writing this document.

1.2 About This Guide

This guide is one of a few files which you will need to assemble your amplifier. There should be the following files included:

- Leach Amplifier Assembly Guide.pdf - This Guide
- Leachamp_4.5 CDR.pcb - This is the PCB file you want to use to build this 2U leach amp. ExpressPCB format
- Leachamp_4.5.pcb - This is the original leach amp PCB from Dr. Leach but in ExpressPCB format
- Leach Amp ExpressPCB Layout.pdf – Use this to ensure the PCB layout for the Leachamp_4.5 CDR.pcb is correct
- Leach Amp PCB BOM.pdf - This is a Bill of Materials file. Use this to order parts for the Main Board
- Leach Amp Chassis BOM.pdf - This is a Bill of Materials file. Use this to order parts for the chassis
- Leach Amp Building Accessories BOM.pdf - This is a Bill of Materials file. Use this to order the tools and accessories you will need to build your amp.
- Leach Amp board components.jpg - This is the PCB board layout drawing you should use for reference as you begin to solder your board.
- Leach Amp Chassis Layout.pdf - This is the chassis layout for each component.
- Leach Amp Chassis Measurements.pdf – This is the approximate measured location for each component in the chassis.
- Leach Amp Rear Panel Layout & Measurements.pdf - This is the combined layout and approximate measured location for each component which resides on the rear panel.

I'm starting this guide late in the process so will do my best to cover all the steps even one's I've long since completed. I want to create a single all incorporated guide to building your amplifier. There is obvious need to include Dr. Leach's work and information. If you had not heard, we unfortunately lost Dr. Leach a few years ago and in the spirit of keeping his work alive, I want to ensure the information from his website is preserved. I'm not sure how long Georgia Tech will keep his site up, but making sure the data is kept alive is paramount. I will cite his work whenever possible and will do whatever I can to make sure all credit is to Dr. Leach. I am giving the information and work within this guide away for free to all. You may distribute as you like. If you have any feedback or have better ideas about citations & credit feel free to email me at leachamp2u@gmail.com.

2. Disclaimer

I've made every effort to ensure everything in this document is as accurate as possible but I simply cannot guarantee you will succeed or that I have not swapped a part in the links I provide, forgot a trace, forgot to add a part, recommended a bolt that is a little too long, etc. In fact, I had to solder 'blue wires' where I left a trace off the PCB. I've corrected the board layout before posting this but it just goes to show how something can always be overlooked. I'm working on my 4th digit in hours committed to this project over the last few months trying to ensure a polished guide. There is a lot of information to convey.

This has been designed and tested to work on a US power grid. I'm not sure what effects 50Hz will have on the design nor have I made recommended purchasing power cords, etc which comply with different country regulations.

I take no responsibility if you get hurt or killed trying to wire, test or otherwise follow the instructions in this guide. These are high voltages that you are working with and dangerous tools you are using so use extreme caution at all times. Wear protective gloves, shoes, mask, glasses & ear plugs at all times. These instructions and this information is free of cost and free of liability for me so please approach it as such. Last point, I'm an engineer not an English major. That will be obvious as you work your way through these instructions. Grammatical corrections will fall on deaf ears. I know the internet is touchy with your/you're their/there etc. I know the difference and I don't care if I got it wrong somewhere.

Now onto the fun...

3. Ordering the Parts

For this section, I have a BOM (Bill of Material) document for each section Main Board, Chassis and Building Accessories. Use these BOM documents to easily place your orders. The links are active and should make it very easy. Depending on how old these documents are, how much the websites have changed and if parts have EOLd (End of life) then you might have to seek out a replacement part. I recommend you place all the orders at one time to save shipping costs as well as catch duplicate orders because some parts are called out twice between the Main Board and Chassis BOMs.

Each 2 channel amplifier will cost you approximately \$500 not including tools & accessories and somewhere around 100 - 200 hours to build (I was probably more meticulous than necessary so you might be able to do much faster). It is always cheaper per amp to build more amplifiers at the same time so see if you can build with a friend to save some cash. For Example, pricing just the PCBs, it cost \$400 to build 6 PCBs but it costs \$200 to build just 2 PCBs.

Purchasing notes:

- Use components that will last.
 - I used all Stainless Steel screws, nuts and washers
 - Use Silver Mica capacitors when you can get deals on them or if you can afford them
- Check your order and how they will fit.
 - I organized each subsection with the parts and put them in sandwich bags to make sure they do not get lost
 - Some parts such as the Power Cord inlet have to have the mounting screw holes drilled out just slightly to fit a #6 screw.

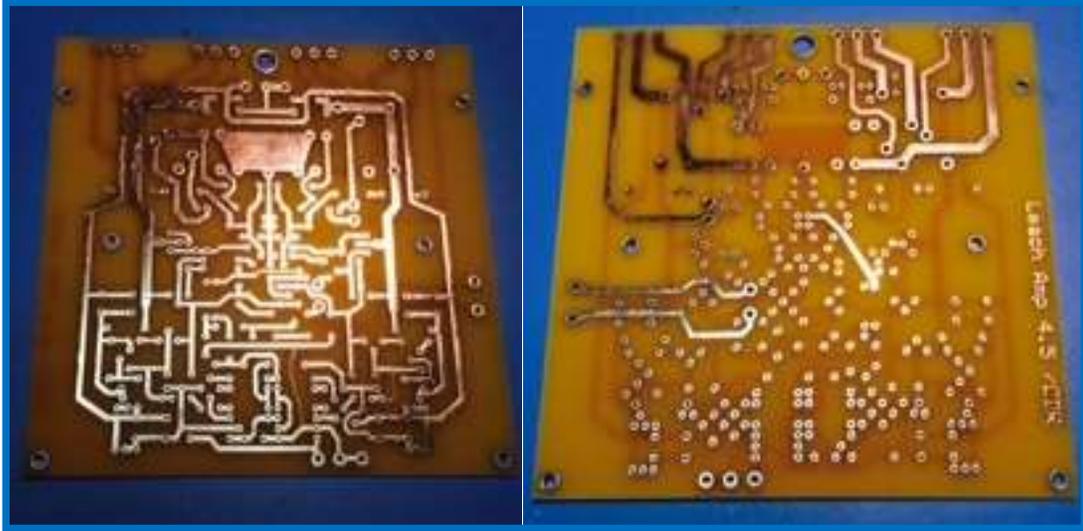
3.1 Main Board

Open 'Leach Amp PCB BOM.pdf' and use it to purchase each item for the Main Board. Follow the instructions below carefully.

3.1.1 Ordering the PCB

3.1.1.1 Overview

The Main PCB board can be the most expensive part of the amp depending on how you procure it. I recommend you use the board layout that I have provided in a file named 'Leachamp_4.5 CDR.pcb' and should be attached with this document. You place the order with ExpressPCB.com through their local application which you will need to install. The application runs a check on the board layout, contacts expresspcb.com's special servers, validates the price, provides you shipping options and takes care of payment. After payment is finalized, your boards will arrive in a few weeks. This is when going in with a friend can really save you some cash. Here is what the board will look like when they arrive:

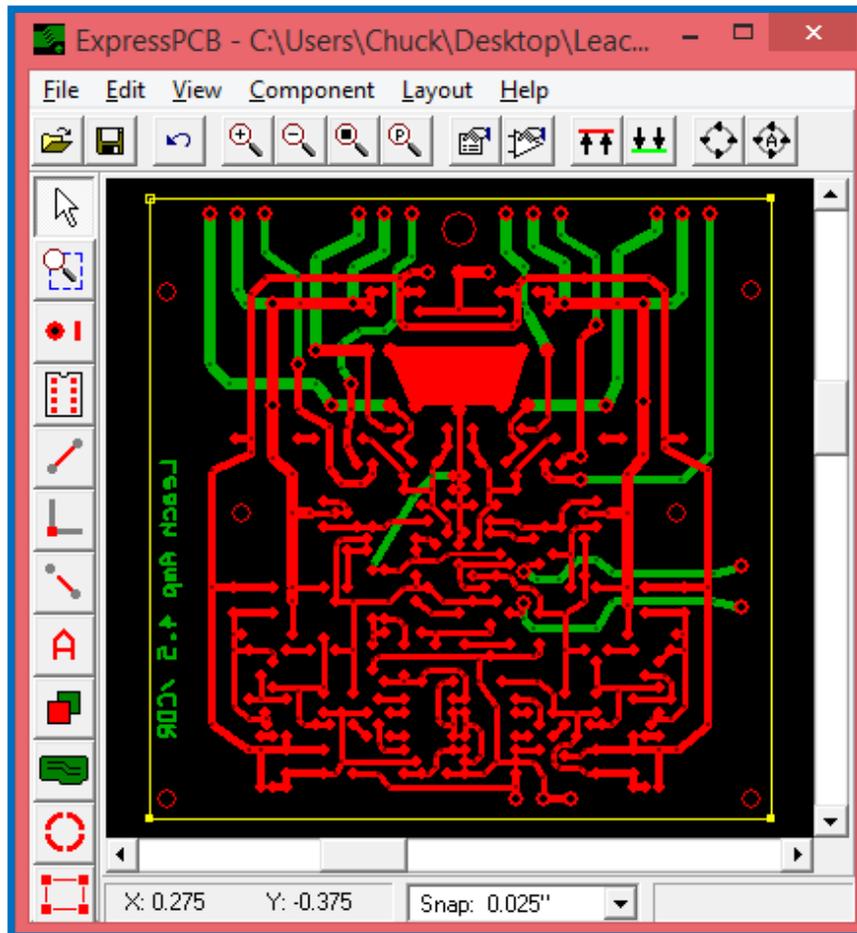


3.1.1.2 Step by Step

- Go to <http://www.expresspcb.com> and click the 'download' link in the upper right side. The icon looks like the icon seen here:



- Next, choose your OS version. I'm going to assume you have Vista, Win7 or Win8 for this guide.
- Double click or otherwise open 'ExpressPCBSetup.exe'
- Click 'Next' then 'Next' then 'Install' (Read and agree to any terms if you need or desire) and finally click 'finish'
- You will notice two new icons on your desktop: 'ExpressPCB' and 'ExpressSCH'. Double click 'ExpressPCB' to launch.
- Click 'OK' to bypass the welcome wizard.
- Now open the 'Leachamp_4.5 CDR.pcb' and you should see this:



- Click 'Layout' from the menu then 'Update Pricing File' then 'Update'
- Now Click 'Layout' from the menu then 'Order Boards Via the Internet...' Click 'Next' twice then fill out the form with your name, email and phone number and click 'next'
- Fill out your address then click 'Next'
- Order the 'Standard Service' Board then 'Next', pick the quantity of boards you need (YOU WILL NEED **2 BOARDS** FOR EACH AMP YOU BUILD IN THIS GUIDE), enter your credit card number and submit. At the time of writing this, two standard service boards costs \$112 including shipping.
- You can close ExpressPCB and move on to the next step of ordering the rest.

3.1.2 Ordering the Rest of the PCB parts

Again, make sure to open 'Leach Amp PCB BOM.pdf' and use it to purchase each item for the Main Board. Go down each item, click the part number to open a web browser to the merchant and part. Add the appropriate quantity in the 'Total' column and add to cart. Make sure you are remembering that you need to build 2 Main Boards for each amplifier if you want to have a two channel amplifier. Please verify that the part description matches the item in your cart. Also, in the first steps of assembling your amp, you will be matching transistors and Zener diodes so if you want to add a few Q1 – Q4 and a few D13 – D16 so you get a closer match by having more to choose from, it might not be a bad idea. I'll leave that up to you to decide. The links are almost certainly correct but there could be one or two mismatches due to oversight. I also recommend you wait to checkout until you have done this for each BOM PCB, Chassis and Building Accessories to save on shipping costs. Some of these parts

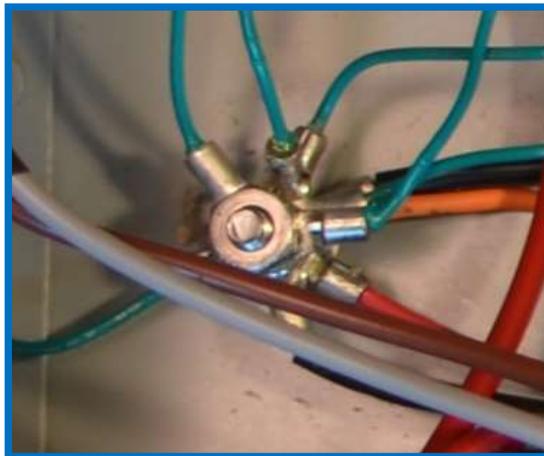
were on sale when I purchased and either the price might be different or link may have changed. If you find problems with the BOM please send me a note and I will update the pdf - leachamp2u@gmail.com.

3.2 Chassis

Open 'Leach Amp Chassis BOM.pdf' and use it to purchase each item for the chassis. Go down each item, click the part number to open a web browser to the merchant and part. Add the appropriate quantity in the 'Quantity' column and add to cart. You are only building one chassis so there is no 'Total' column. Please verify that the part description matches the item in your cart. The links are almost certainly correct but there could be one or two mismatches due to oversight. I also recommend you wait to checkout until you have done this for each BOM PCB, Chassis and Building Accessories to save on shipping costs. Some of these parts were on sale when I purchased and either the price might be different or link may have changed. If you find problems with the BOM please send me a note and I will update the pdf- leachamp2u@gmail.com.

3.3 Building Accessories

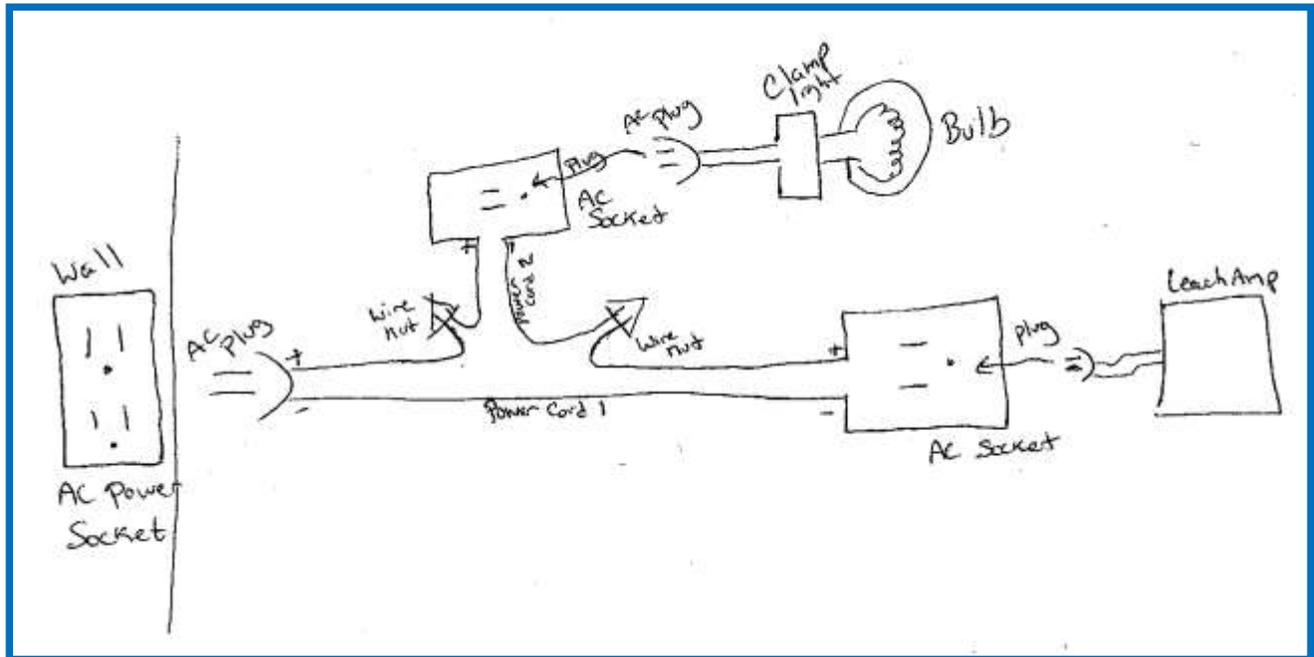
Open 'Leach Amp Building Accessories BOM.pdf' and use it to purchase what you need to build your amplifier. Bottom line is that I have no idea what kind of tools and equipment you have to build the amp so I've just listed everything I've used. I do recommend you purchase the items under the 'Electrical recommended' and the 'Mechanical recommended' groups. Pick and choose what you want under 'Electrical optional.' The Ring terminals make the wire terminations clean at such places as the ground lug. See picture below



You will need hookup wire under 'Electrical optional' but I'll let you choose the colors and length. Per Dr. Leach you will need #22 stranded wire for connecting D1 – D4 to the Main Board (I actually used the orange solid wire), #18 stranded wire for all the 120AC/+40V/-40V wires between the transformer, main power capacitors, rectifier, fuses & Mainboard. *"I prefer wiring the chassis with 3 colors of wire, e.g. red for +Vcc, yellow for -Vcc, and black for ground. It is probably cheaper to use a single color."* [1] I also used solid wire from the input to the PCB to get a tight twisted pair that would keep shape (orange #20 and Blue #22). Finally use #20 stranded wire for all the rest.

I've only added the tools here so you can see what I used the most. They are intended to be seen as examples and I in no way want to get into the tool recommendation business. Some of these are the tools I used, some are just some I found on the internet because I couldn't find the one I use. Some of these tools are

non-obvious so let me explain those. The Breadboard and power supply are used to do the ‘Transistor Matching’ portion of Dr. Leach’s construction details. I built the power supply from an old Desktop PC power supply. There is a ton of these designs online but here is an example: <http://www.instructables.com/id/ATX--%3E-Lab-Bench-Power-Supply-Conversion/>. The variac is used to control the voltage coming into the amp to prevent you from burning it up during testing. I didn’t have one handy so I build a current limiter instead. The basic premise is that you can limit the current coming into the amp and therefore prevent major damage if you mess up (It took me nearly 20 hours to debug a grounding issue which caused me the burn out quite a few resistors. The current limiter gave me more time before the resistor started smoking between each test). A diagram of how to build it is below. Note that the light will not turn on until the amp is turned on and it will dim quickly as the amp warms up. I will go over this in more detail later during the ‘Testing’ portion of this guide.



3.4 Now we wait

All The parts should now be on their way. As they come in, arrange them as I have... Put the Main Board components together, the chassis parts together & the building accessories together. Verify you have them all and you are ready to move on to the assembly section.

4. Assembling the PCB

This is where I will begin to heavily leverage Dr. Leach’s work. Again, his website has been kept online for a few years after his unfortunate passing and I’m not sure how long the university will keep it up so I’ll be adding the content here and citing it appropriately.

4.1 Transistor Matching

It is very likely you will receive your Q1 – Q4 transistors before all your parts arrive so start this step as soon as you can. My recommendation is to use the breadboard, benchtop power supply, multi-meter, resistors of the calculated value and your transistors for this step. The curve tracer is not necessary. Now, here is Dr. Leach’s instructions for this step:

“For minimum dc offset at the output, Q1 through Q4 should preferably have matched current gains. The current gains can be measured with a curve tracer or with a multi-meter which has this capability. Ideally, all four transistors should be matched. If this cannot be achieved, the second choice is for Q1 and Q3 to be matched and

Q2 and Q4 to to be matched. The third choice is for Q1 and Q2 to be matched and Q3 and Q4 to be matched. The typical dc offset at the amplifier output is less than 50 mV. If you are concerned about how well the transistors are matched, I have seen amplifiers built without matching the input transistors, and they had no dc offset problems.

If you do not have access to a curve tracer, the circuits in Figure 1 can be used to match the transistors. These circuits can be easily assembled on a solderless electronic breadboard. The bias current in each transistor is set at approximately 1.6 mA. Matched transistors will have equal base currents. The expected current should be in the approximate range from 4 uA to 20 uA.

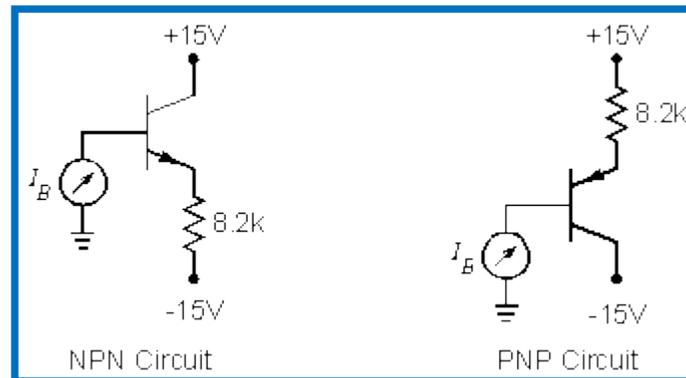


Figure 1. Transistor matching circuits.

When using these circuits to match transistors, the power supply voltages should be held constant. Don't turn the voltage down or off when changing transistors. If the measured current appears to be unstable, the transistor may be oscillating or picking up a RF signal. In this case, a 0.1 ufd capacitor from base to emitter should cure the problem. You may notice some temperature drift of the current as the transistors warm up. You can use 9 V batteries in place of the 15 V sources if you change the 8.2 kohm resistors to 4.7 kohm.

If you do not have a multimeter that reads microamps, you can put a 51 kohm resistor in series with the base and measure the voltage across the resistor. Matched transistors will have equal voltages. The expected voltages should be in the approximate range of 0.2 V to 1 V.” [2]

4.2 Zener Diode Matching

Unfortunately I cannot remember exactly how I handled this step because it was done nearly 18 months ago. Here is Dr. Leach's instructions for this step:

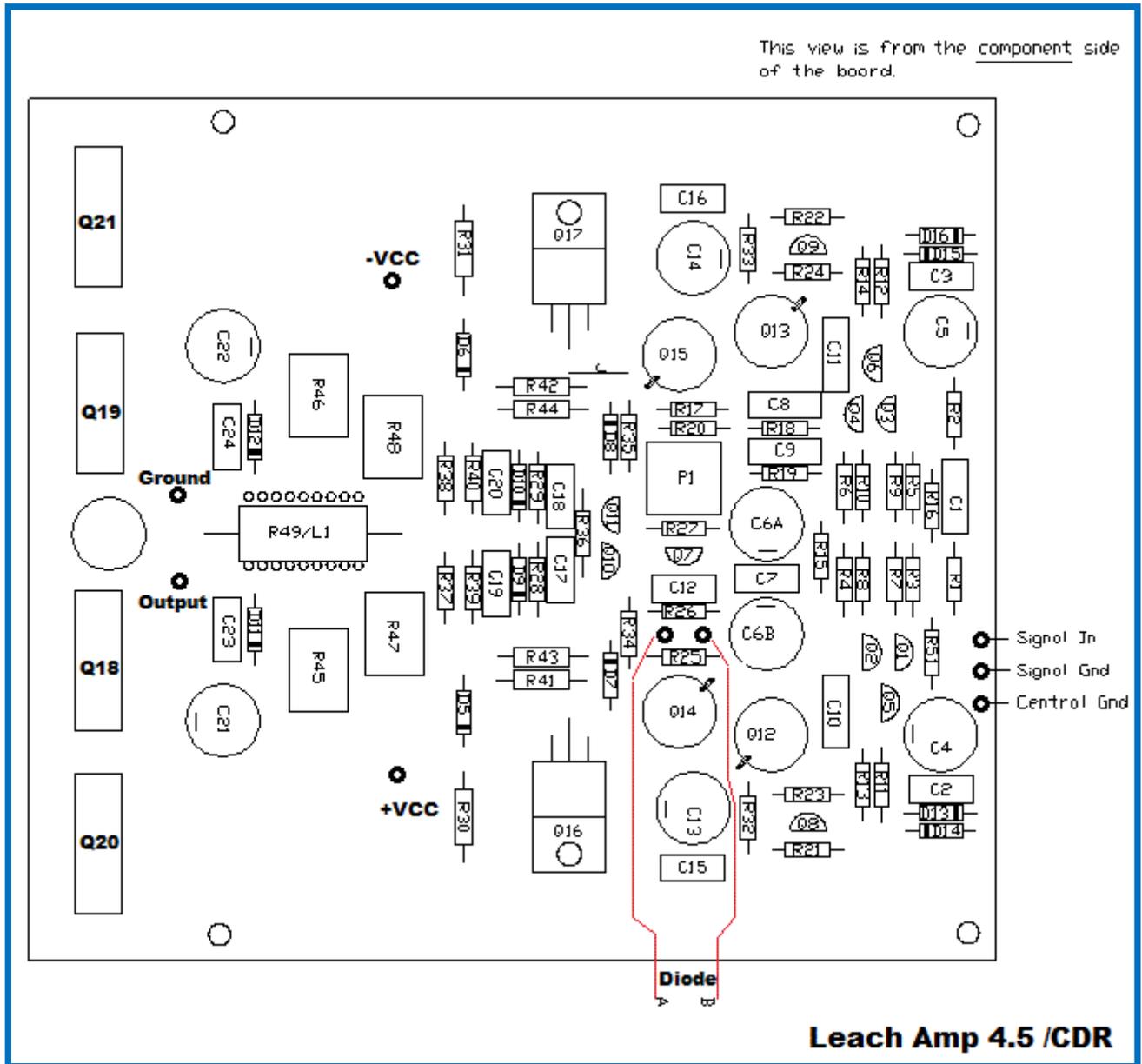
“The bias currents in the diff amps are regulated by zener diodes. In each diff amp, two 20 V diodes are used in series to form an equivalent 40 V zener. Although a single diode could be used, the error tolerance is less when two diodes are used in series. If you are building a stereo amp, you need 8 diodes. It is probably cheaper to buy a single pack of 10. I prefer to measure the zener voltage of each and pick series combinations of diodes that give equal reference voltages on each circuit board. It is not necessary to have exactly +40 V and -40 V. But the two voltages on each circuit board should be as close as possible to minimize dc offset problems.

The zener voltage of a diode can easily be measured by connecting a resistor in series with it on an electronic breadboard and connecting a bench power supply across the series combination. The power supply voltage should be set so that the current through the diode is 3.3 mA. If you don't have a current meter, the voltage across the resistor should measure $3.3 \times R$, where R is in kohms. For example, the voltage across a 2 kohm should

be 6.6 V. Once the current is set, the dc voltage across the zener diode can be measured with a voltmeter. For example, if you have +15 V and -15 V supplies, connect the diode in series with a 3 kohm resistor across the two outputs and measure the diode voltage. If you get the diode backward, the resistor will get hot.” [3]

4.3 Assembly of Main Board PCB

On the next page, there is a picture I’ve modified from Dr. Leach. It shows where to place the components on the PCB. I’ve modified it to reflect the changes I made to the PCB as well as make it a little more legible. In addition, the full resolution version will be packaged with this guide (Leach Amp board components.jpg).



Now we are ready to start soldering parts to the board. Note that we are using a non-polar (also called bipolar) capacitor for C6 when looking at Dr. Leach’s instructions below. This means you will install the jumper labeled J as well as the jumper across C6B. DO NOT solder Q18, Q19, Q20 or Q21 yet. We do this in a later step. I highly recommend you use a Weller or equivalent high end soldering iron. I used a Metcal PS2E-01 which makes this job a snap. Dr. Leach’s instructions:

“Start assembly of the circuit boards by soldering the smallest components first, working up to the largest components last.

If you do not know how to solder, please find someone who can teach you. Do not use a soldering iron that gets too hot. I like the temperature regulated Weller irons, but they get a little pricey. If you use a non-temperature regulated iron, it should have a power rating of about 30 watts maximum. The principle reason for poor solder joints is not enough heat. But too much heat can damage a component or make a pad on the circuit board peel up.

To get a good solder joint, simultaneously apply the tip of the iron and the solder to the circuit board so that they touch each other and touch both the wire being soldered and the solder pad, i.e. all 4 are in contact. When the solder begins to flow, remove the solder and hold the iron on the joint until the solder flows and bonds to the wire and the pad. Then pull the tip of the iron up so that it slides up the wire. A good solder joint is smooth and shiny. It doesn't have any waves in it and it doesn't look like a drop of water on a waxed car. Figure 2 illustrates what a correct solder joint looks like. It also shows one where too much solder is used and one where not enough heat is used. In the latter case, the solder is bonded to the pad on the circuit board but not to the wire. When you cut the wire, cut it just above the solder joint. Do not cut into the solder.

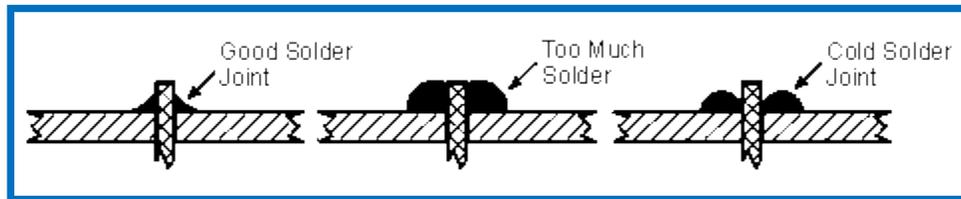


Figure 2. Good and bad solder joints.

The assembly procedure is approximately as follows:

- *Install and solder the short-circuit jumper labeled J on the circuit board layout near Q17. If you use a single nonpolar capacitor for C6 (as opposed to the two polar capacitors C6A and C6B as described in the Parts List), solder a short-circuit jumper in the holes for C6B.*
- *Install and solder the resistors. A resistor lead bender should be used to bend the leads for the proper length. No more than about 5 resistors should be soldered and clipped at a time. Check the value of all resistors with an ohmmeter before soldering them to the circuit board. I saw one amp which had 33 kohm resistors where I specified 3.3 ohms. Another amp had a 12 kohm resistor where there should have been a 1.2 kohm resistor. This caused a bad dc offset problem.*
- *If there is a problem with the VBE multiplier bias circuit, it could cause R36 to smoke and even catch on fire. To minimize damage to the circuit board in this event, I recommend stripping two 1/4 inch pieces of insulation from some hookup wire and putting them on the leads of R36. This will cause R36 to stand up off the circuit board to minimize damage in the event that it smokes. If it does smoke, you can lose expensive output and/or driver transistors, so be careful and don't make errors.*
- *Install and solder Q1 through Q11. Don't bend their leads before putting them on the board. Their cases should be about 3/16 inch off the board. Figure 3 shows the pinouts of all of the transistors.*

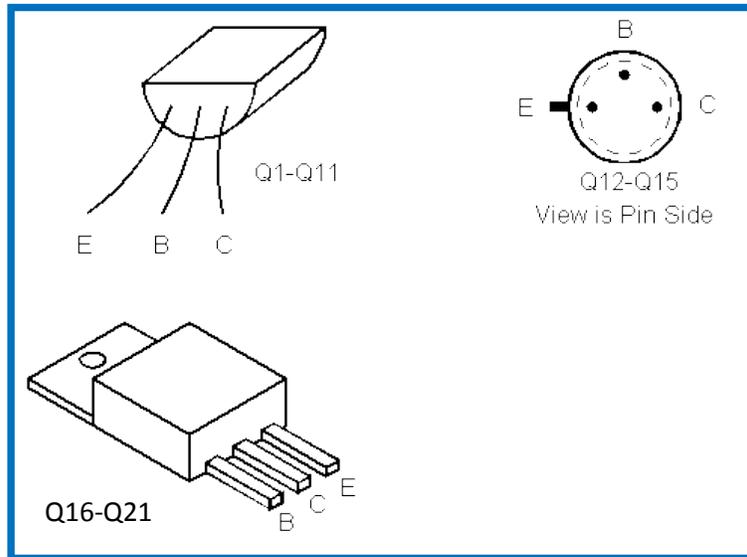


Figure 3. Transistor Pinouts.

- *Install and solder Q12 through Q15. These transistors should be flush with the board.*
- *Install and solder Q16 and Q17. First, use needle nose pliers to bend the transistor leads at 90 degrees so that they will mount on the circuit board with the hole in the metal tab on the transistor mating with the hole in the circuit board. Then put the transistor and its TO-220 heat sink on the circuit board and secure them with #4 machine screws and nuts. Solder and clip the transistor leads. It is not necessary to use any insulating wafer between the transistors and heat sinks. If you use the clip-on version of the TO-220 heat sinks, don't trust the clips without the screws to make good thermal contact between the transistors and the heat sinks. Screw them down to the circuit board.*
- *Install and solder L1/R49. This is a 2 W resistor with an inductor wound around it. Figure 4 illustrates this part. There are 11 to 12 turns of wire around R49. Instructions for winding the inductor are given in the Parts List.*

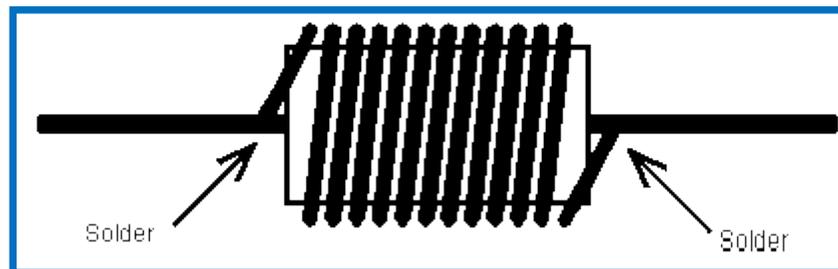
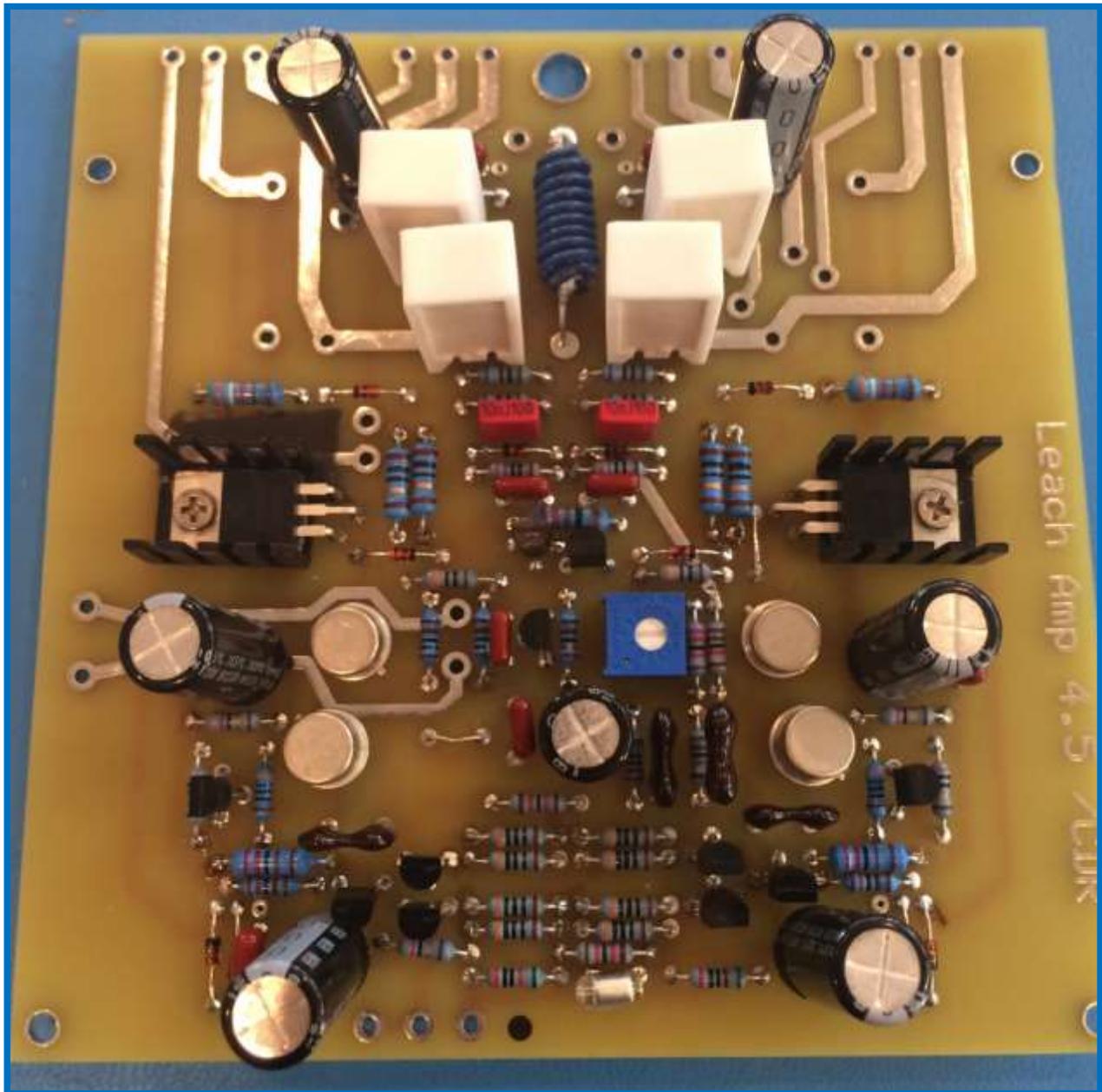


Figure 4. L1 wound around R49.

- *When all components are soldered to the circuit board, the flux should be removed with a solvent such as Stripper brand circuit board spray cleaner and a soft bristle brush. If this is not done, noise problems can arise that are caused by random pulses of current through the flux. I once saw flux catch on fire when it bridged the trace of a power supply rail and a ground trace. [4]*

When you are done with this section, your board should look the same as the picture on the next page.



5. Assembling the Chassis

Now we move on to assembling the chassis. I recommend you print out all the layout and measurement pages now and keep them close by as you build. I have provided you the measurements for each component location but you still must dry fit all the components before drilling, cutting, marking especially if you deviated from the selected components from the BOM list (the pdfs with all the components). You might find that some areas are too tight for your hands or tools or there could be direct interference which could cause a short or could rub and eventually fail.

Notes on Measurements: These measurements are based on my amplifier after assembly. Each component you've ordered can be slightly different than mine so you MUST verify and locate the best place to mount before drilling.

5.1 Mechanical

5.1.1 Countersink the Chassis Screws

You should have ordered 22 #6-32 3/8" flat head stainless steel screws. Use these to countersink and replace the outside facing chassis screws on the top, bottom and both side panels that came with the 2U chassis.

Notes on drilling: I always mark the spot I need to drill with a pencil then use a spring loaded center punch to keep the drill bit from slipping. I nearly always used my benchtop drill press with new drill bits to drill the initial hole in the chassis followed by a larger bit to countersink the flat head Philips screws. This is done primary for aesthetics but also keeps the screw heads from binding or scratching other surfaces. It may take some practice to get the countersinking done correctly and due to drill bit taper variations, I can't make a good guess on the size of bit or size of tool you need to make the head flush. The pictures below show what the flat head screw, the countersink and what the final flush mount. (Sorry for the terrible picture quality). You will also need to countersink the rest of the components which are mounted to the inside bottom panel.

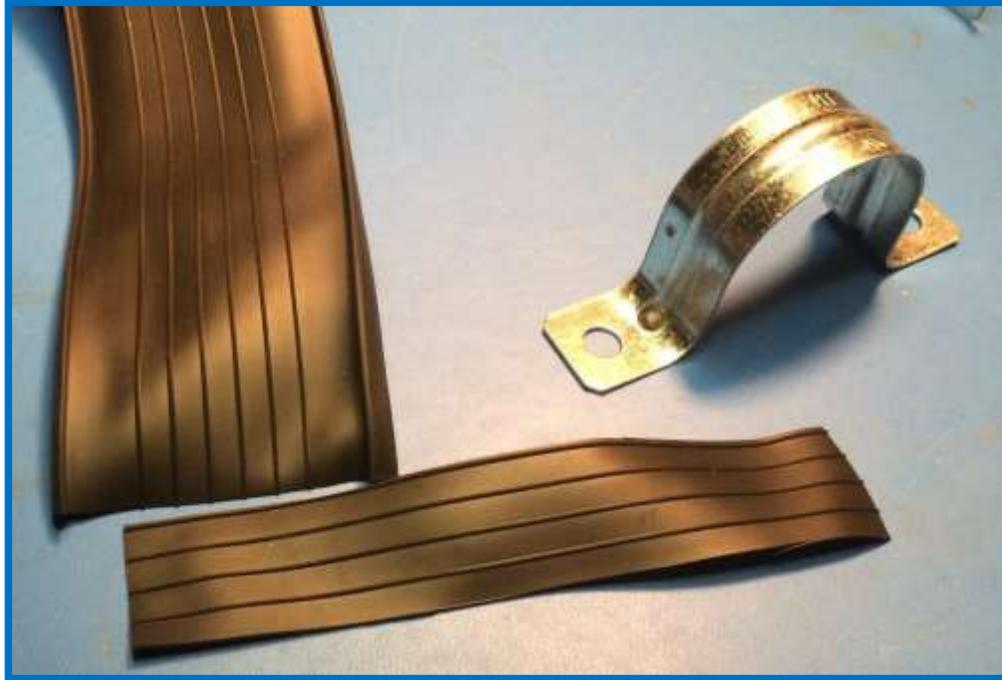


5.1.2 The Bottom Panel

First, make sure you've printed 'Leach Amp Chassis Measurements.pdf' and 'Leach Amp Chassis Layout.pdf' for reference. To determine which component the reference in the diagram stands for, use the Chassis BOM. Follow the steps below

- I recommend as Dr. Leach says in his guide to use masking tape on any surfaces that will be touching other metal surfaces to prevent scratching. I did not do this myself and there are noticeable marks on the back but they do not bother me as only the front will be exposed in my media rack.
- Dry fit each component before doing anything else. Use the measurements provided to make sure there is nothing binding or interfering. You might find it useful to roughly mark the location of the mounting holes for each component on the chassis bottom panel with a pencil.
- I started with the FH (fuse holder) so measure, mark, tap with spring loaded center punch and drilled with 1/8" drill bit. I did have to drill out the hole in the fuse holder so the #4-40 nut would fit into the recess. Do not drill that larger hole all the way through the fuse holder. On the underside of the bottom panel, countersink the screw head for a flush mount.

- Next, I drilled the ground lug just as step above using a 7/32" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount.
- Now, I placed CP1, CP2, CPH1, CPH2 & T1 on the bottom panel at the same time and ensure there is no interference. Mark the mounting holes in CPH1 & CPH2 using the measurements as a guide. If you need to move the capacitors further away from the transformer for clearance that should be fine. After you're happy with the locations, tap & drill the holes for CPH1 & CPH2 with a 7/32" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount.
- Now dry fit T1 and make sure there is proper clearance with the front panel. Mark, tap and drill the hole for T1 with a 7/32" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount.
- Now dry fit the RC1 regulator using measurements for guidance. Mark, tap and drill the hole for RC1 with a 7/31" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount. Use the #10 x 3/4" flat head screw and #10-32 nut to temporarily mount the regulator. Use the pencil to outline around the regulator making a square. Now remove the regulator and use a Dremel tool with grinding wheel to etch away the paint under the regulator from the bottom panel. Later we will be using thermal paste to bond the regulator to the chassis.
- Next, we want to mount the QFH. There are four mounting holes so I used just the first and last one. Dry fit, measure, mark, tap and drill the holes for QFH with a 5/32" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount.
- Now for the PCBs. It is critical to get the location of all four holes for each pcb in the right location so I placed the PCBs directly on the bottom panel in the location I've provided and penciled the exact spot of each mounting hole. Then mount the 1/2 stainless steel stand offs to the PCBs (four each) with the 4-40 nuts. Tap and drill the hole for PCB1 & PCB2 with a 1/8" drill bit. On the underside of the bottom panel, countersink the screw head for a flush mount.
- Using the Chassis BOM for screw size reference, permanently mount FH & QFH.
- Permanently mount RC1 by using thermal paste between the regulator and the bottom panel.
- For the ground lug, thread a #10-32 x 1" screw through the hole then a #10 lock washer and #10-32 nut to permanently mount. I didn't need to etch away the paint to make a solid connection to the chassis bottom due to countersinking the screw.
- Temporarily mount PCB1 & PCB2
- Do not mount T1 yet.
- Cut the weather seal to create CPG1 & CPG2. Cut the thick edges off each side then make long enough to wrap around the capacitor one time. This will keep the metal bracket from rubbing the capacitor and damaging it. Additionally, I made another strip to lay underneath the capacitor between it and the bottom panel to ensure a firm grip.



- Permanently mount CP1 & CP2 using the gaskets CPG1 & CPG2 and the brackets CPH1 & CPH2. Use the #10-32 x 5/8" bolt, #10 (3/16") x 1" washer and #10-32 nut to securely fasten.
- Put the #10 pan head 3/8" screw and #10 lock washer into both capacitor terminals for both CP1 & CP2.



5.1.3 The Heat Sinks

Fortunately, most of the drilling is already completed on the heat sink. We will use the existing holes to mount the power transistors to heat sink. We do need to CAREFULLY drill mounting holes and

the bias diode holes. In this section, we will only do drill for the bias diodes. The mounting holes will be drilled in the rear panel section.

For this heat sink, the optimal location for the diodes are directly center and down 1.25 – 1.3” from the top. You can use the pictures below for reference. The spacing between each diode is 0.25”. I’ll use Dr. Leach’s guide for the rest of this section below.



“The heat sinks should be marked, drilled, and deburred carefully. You can lose expensive transistors if a short circuit occurs in a heat sink or if a burr penetrates an insulating wafer between a power transistor and a heat sink. The heat sink for each channel is drilled so that the four output transistors mount in the channel with the four temperature sensing diodes in the center.

1. *Use a drill press, not a hand drill, to drill the holes. To prevent burrs, use sharp drill bits and drill into the side of the heat sink wall on which the transistors mount. Drill pilot holes first with smaller size bits. The hole diameter for the transistor mounting hardware is normally 1/4 inch. The hole diameter for the bias diodes must be determined with a drill index.*