Q-Tune

Chromatic Instrument Tuner Pedal

BUILD DIFFICULTY
★★★☆☆ Intermediate

DOCUMENT VERSION
1.0.3

PROJECT SUMMARY

Q-Tune is a DIY chromatic instrument tuner pedal based on the ESP32 microcontroller and a 2.8″ color LCD display. It is designed for builders who enjoy assembling their own gear with high accuracy and a responsive UI.

Q-Tune can be housed in a 125B or 1590B enclosure and uses standard through-hole components wherever possible, making it accessible to both beginner and experienced builders. Q-Tune kits come pre-installed with the Q-Tune software. The latest version of the software can be installed using the Q-Tune website: https://q-tune.com/install

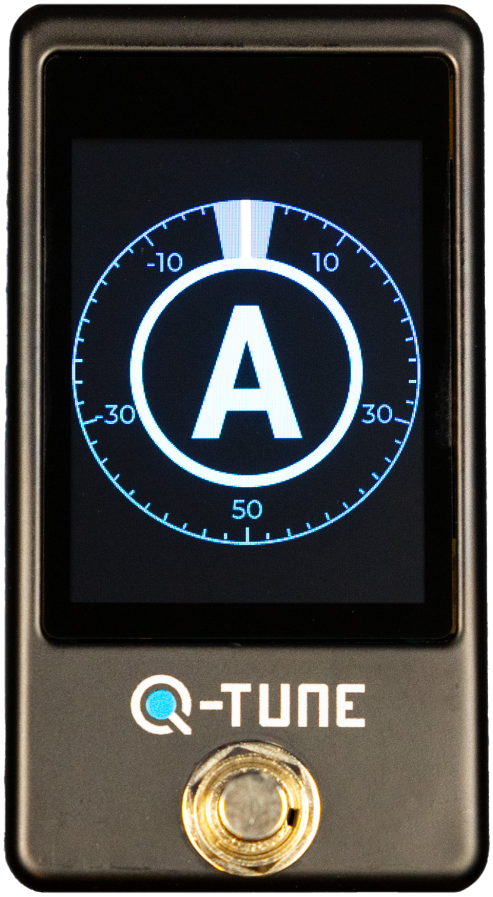


TABLE OF CONTENTS

INTRODUCTION

If this is your first pedal, congratulations for jumping into the fun of building your own pedals! Thank you for choosing Q-Tune.

If you're a seasoned DIY-er, we hope you'll enjoy building your very own high-quality tuner pedal.

Here are a few things to review before starting:

- **You're going to have to get your hands dirty.** Nothing comes pre-assembled, and you'll need to learn the skills to put it all together. Not to worry, we'll help you through every step, but be open to learning a few things along the way.

- **This will take time.** Plan on about two to four hours start to finish. It may even take you longer if it's your first time. Make sure not to rush. If you're feeling frustrated or overwhelmed, take a break and come back in a couple hours or even the next day.

- **No direct technical support is offered.** There are several DIY forums and Facebook groups with thousands of members who enjoy troubleshooting and teaching. Some possible things to review can be found on the Troubleshooting page. Feel free to participate in the project's discussions and even file an issue if you discover a bug with the software. But, please be sensitive to the fact that the maintainers of this project are participants just like you.

- **There is no implied guarantee of a final product.** We've provided the ingredients and the recipe but you are responsible for putting everything together to make it work. We've tried to make the process as clear as possible, but it must be expressly stated that purchasing a kit is not a guarantee that you will end up with a working pedal.

It's recommended to read through all of the instructions before you start, particularly if you've never built a pedal before. If you familiarize yourself with the entire process ahead of time and you know what the goal looks like, each step will make more sense.

PACKING LIST

This is a list of all the parts that are included with the kit, grouped by value.

If you find that any parts are missing or damaged, please fill out the Missing Parts form.

Film Capacitors

Value	Name	Qty
100n (.100J63)	2	2
470n (.47J63)	1	1
		3

MLCC Capacitors

Value	Name	Qty
100n (104)	C12, 15, 16, 19, 22, 24	6
330 (334)	C3	1
470n (474)	C5	1
1u (105)	C4	1
10u (106)	C6	1
		10

Polarized Capacitors

Value	Name	Qty
4.7u	C13	1
10u	C8	1
22u	C21, C23	2
47u (low ESR)	C14, C17	2
100u	C11	1
220u	C18	1
		8

Resistors

Value	Name	Qty
47R	R11, R13, R21, R23	4
330R	R10	1
2k2	R12, R22, R24	3
7.5k	R6	1
10k	R8, RPDo	2
100k	R2, R3, R4, R9	4
150k	R7	1
220k	R5	1
2M	R1, RPDi	2
		19

Diodes

Value	Name	Qty
1N5817	D11, D12, D13	3
1N4148 (41 48)	D21 - D28	8
1N5226 (1N5 226)	D1	1
		12

PACKING LIST (Cont.)

Chips

Name	Type	Value	Qty
IC1	Op Amp	TL072	1
IC2	Op Amp	MCP6002	1
IC3, IC4	Timer	NE555	2
IC5, IC6	Relay	TQ2-L2-5V	2
IC7	5v Regulator	LP2950 (KY5050)	1
IC8	5v Regulator	LF50	1
			8

Screws

Name	Type	Value	Qty
Screw	Clip to EBD2	M2.5 x 5mm	4
Screw	For Nylon Mount	M3 x 5mm	2-4
Screw	For 125b only	M3 x 12mm	2
			8

Tools

Wrench	Hex Key	3 mm	1
Wrench	Hex Key	2.5 mm	1
			2

Sockets

(IC5, IC6)	Socket	40-pin Stick	1
(IC1 - IC4)	Socket	DIP-8	4
			5

Mounts and Standoffs

PCB Mount	Nylon	Side	2
Standoff	3D Printed	2mm	4
			6

Jacks & Switches

Input, Output	Audio Jack	Mono-Switched	2
DC	DC	9V DC	1
Foot Switch	Momentary	Non-latching	1
			4

Miscellaneous

Enclosure	125b or 1590b		1
PCB			1
Screen	LCD	ESP32	1
Wire	24 awg	8 inches	1
Tie Strap	for Momentary		1
			5

TOOLS NEEDED

Soldering Iron

Temperature-adjustable is recommended. The optimum soldering temperature is 700-725°F (371-385°C) for leaded solder or 750°F (400°C) for lead-free.

Solder

Preferably 63/37 or 60/40 leaded solder. Lead-free is more difficult to use, so if that's the only type you can get, its best to watch tutorials that are specific to lead-free solder.

Digital Multimeter (DMM)

Most cheap ones in the $10-30 range are fine for what we're doing. Make sure it has audible continuity testing (i.e., it beeps at the lowest resistance) and transistor hFE measurement. The AstroAI Digital Multimeter and Analyzer is a tremendous value for ~$25.

Wire Snippers

Also called nippers or wire cutters. The Hakko CHP-170 is the best you can get inexpensively.

Flat-Nose Pliers

Many general-purpose uses, but particularly tightening the nuts of jacks. Quicker than changing out sockets on a ratchet, though feel free to use a ratchet if you prefer.

Needle-Nose Pliers

These are used for bending leads on components and other general uses. Use the smaller type with a tip that's approximately 0.05″ (1.25mm) wide.

Screwdriver (Phillips)

Used for the enclosure and PCB mounting screws.

Painters Tape

Used to hold components or wires in for soldering.

12 mm and 9/16" Sockets

Used for tightening the audio jacks and momentary switch respectively. Generally 3/8" drive. While not necessary, DEEP sockets are easier to use.

Wire Strippers

Used to strip the ends of the 12 pin ribbon cable and momentary wires. The self-adjusting kind are amazing.

COMPONENT IDENTIFICATION

The components shown are not to scale but these illustrations should help you easily identify them.

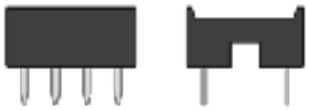
SILICON DIODE

ZENER DIODE

SCHOTTKY DIODE

RESISTOR

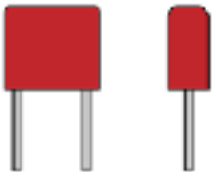
IC SOCKET



INTEGRATED CIRCUIT (IC)

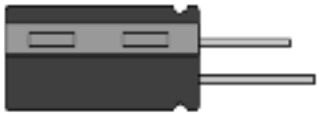
MULTI-LAYER CERAMIC CAPACITOR (MLCC)

FILM CAPACITOR



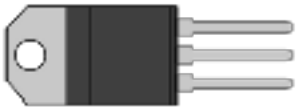
(Color may vary)

ELECTROLYTIC CAPACITOR



5V REGULATOR PKG: TO-92

5V REGULATOR PKG: TO-220



RESISTOR IDENTIFICATION

The resistors in the kit use a 5-band identification code. For convenience they are listed here but verify with a multimeter if you have any doubt.

47R

yellow-violet-black-gold-brown

330R

orange-orange-black-black-brown

2.2K

red-red-black-brown-brown

7.5K

violet-green-black-brown-brown

10K

brown-black-black-red-brown

100K

brown-black-black-orange-brown

150K

brown-black-green-orange-brown

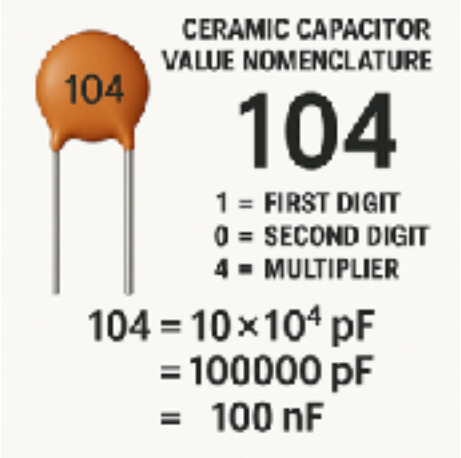
220K

red-red-black-orange-brown

2M

red-black-black-yellow-brown

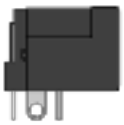
CAPACITOR IDENTIFICATION



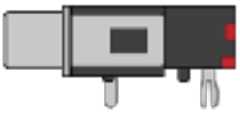
HARDWARE IDENTIFICATION

The hardware comes unassembled, so you'll need to sort & identify each of the pieces.

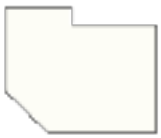
DC JACK



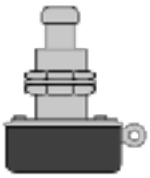
IO JACK



PCB MOUNT



FOOT SWITCH



SCREEN STANDOFF CLIP

JST CABLE

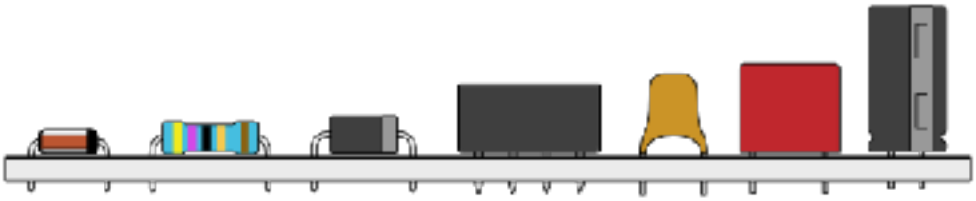
40-PIN SNAP APART SOCKET

PCB: OVERVIEW

Time to start building!

The general principle for PCB population is that you work in stages from shortest components (i.e., lowest profile) to tallest so that when the PCB is upside-down, everything is making contact with the work surface and held in place.

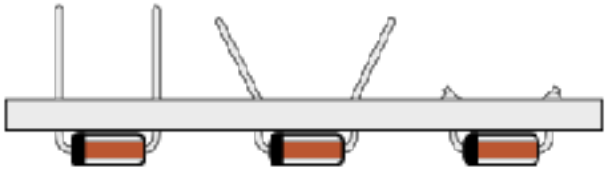
So, you will start by populating the small diodes (the lowest-profile components), followed by the resistors & Schottky diodes, sockets, MLCCs, film capacitors, and finally the electrolytic capacitors.

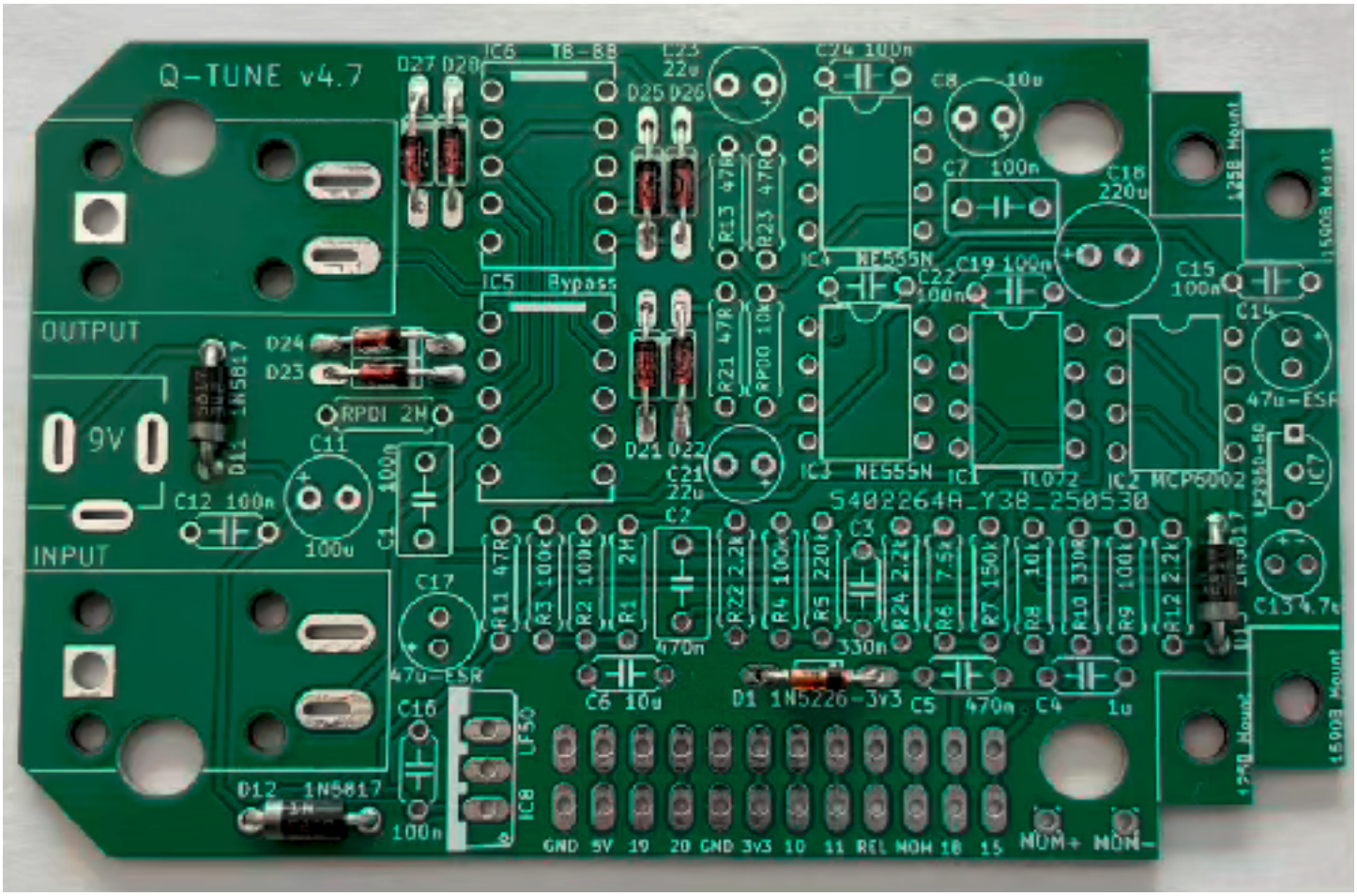


PCB: DIODES

Using the parts list above, populate the smaller diodes by pushing them through the holes and bending the leads outward at an angle to hold them in place. **Diodes are polarized**, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

Flip the board over, solder all the diodes, then cut the leads. You'll use this same technique for most of the other components as well. If this is your first time soldering, watch tutorial videos on YouTube and make sure you get it down before you begin. You don't want to practice or experiment on this board!

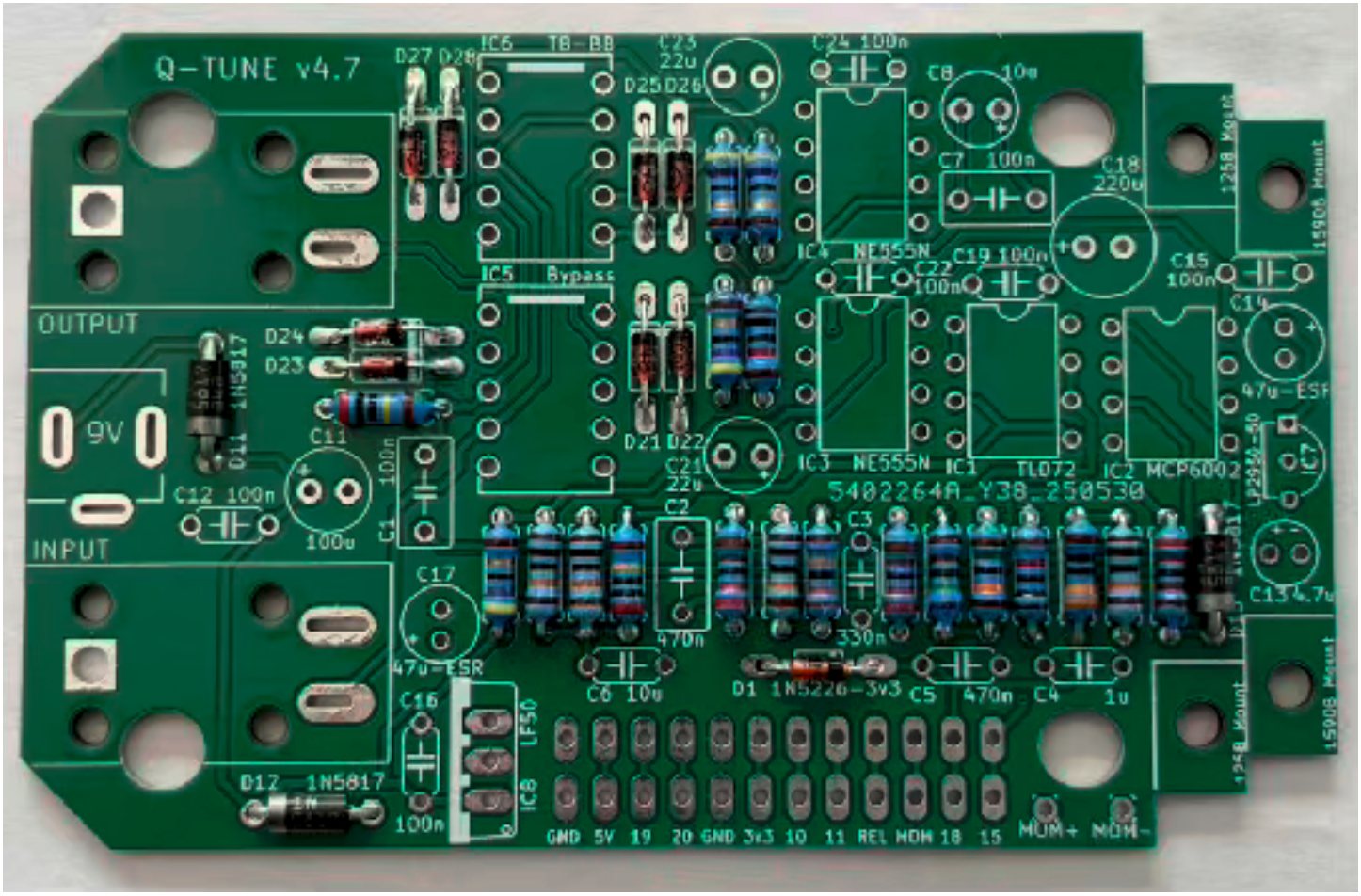




PCB: RESISTORS

Next, you'll populate the resistors. Resistors are not polarized so they will work in any direction. Some builders like to line them up in the same uniform direction with the tolerance band on the same side, but that's a personal preference for you to decide.

Don't try to do all of the resistors at once. You'll want to stop periodically, flip the board over, solder everything, then cut the leads using the wire snippers to make room for more. Generally you don't want to do more than 5 to 10 resistors at a time or the bottom of the board will get too crowded. One approach is to install all the resistors of the same value at once.



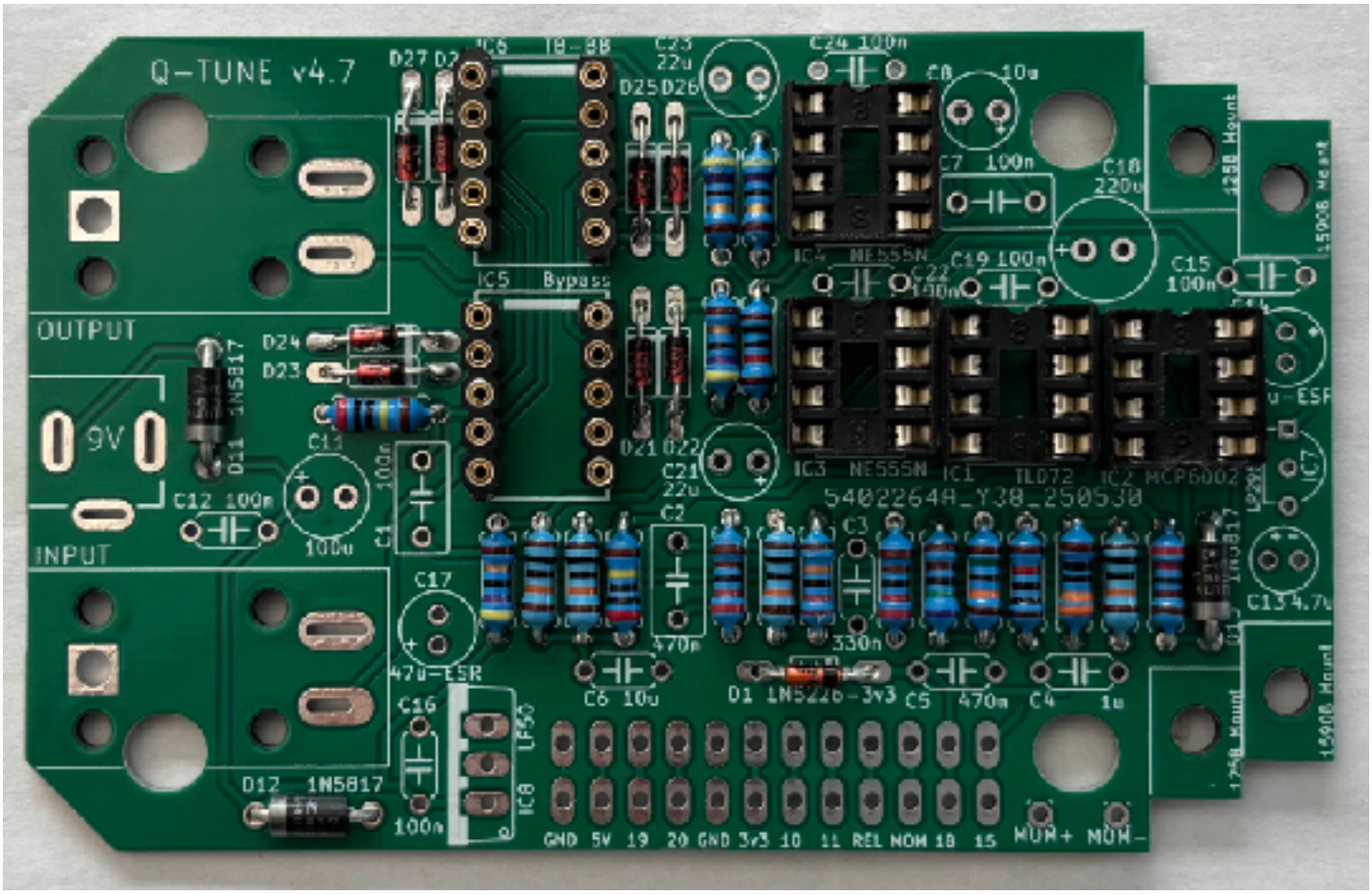
PCB: SOCKETS & ICs

Next up are the sockets. You can't bend the leads of the sockets like you can with the other components so they won't stay in on their own until they are soldered. Pay attention to the markings on the PCB noting which side is pin #1. Orient the socket with the half-circle notch on the side of pin #1.

Again, it's much easier to do all of these at once with gravity holding them in place for you. You'll want to do them before you do any of the taller components.

For the relay sockets, you'll cut the supplied 40-pin socket header into sections of 5 pins each. For best results, use a sharp utility knife to cut them. Wire cutters can also be used but may not result in a clean cut. Prepare 4 sets of 5-pin sockets. To ensure they are soldered perpendicular to the PCB, it is recommended to put 2 socket segments on a relay and install and solder it with the relay lining it up. Then remove the relay.

Don't insert the ICs into the sockets just yet. We will do this in a later step, after the entire board has been populated with the tallest components (the polarized capacitors).

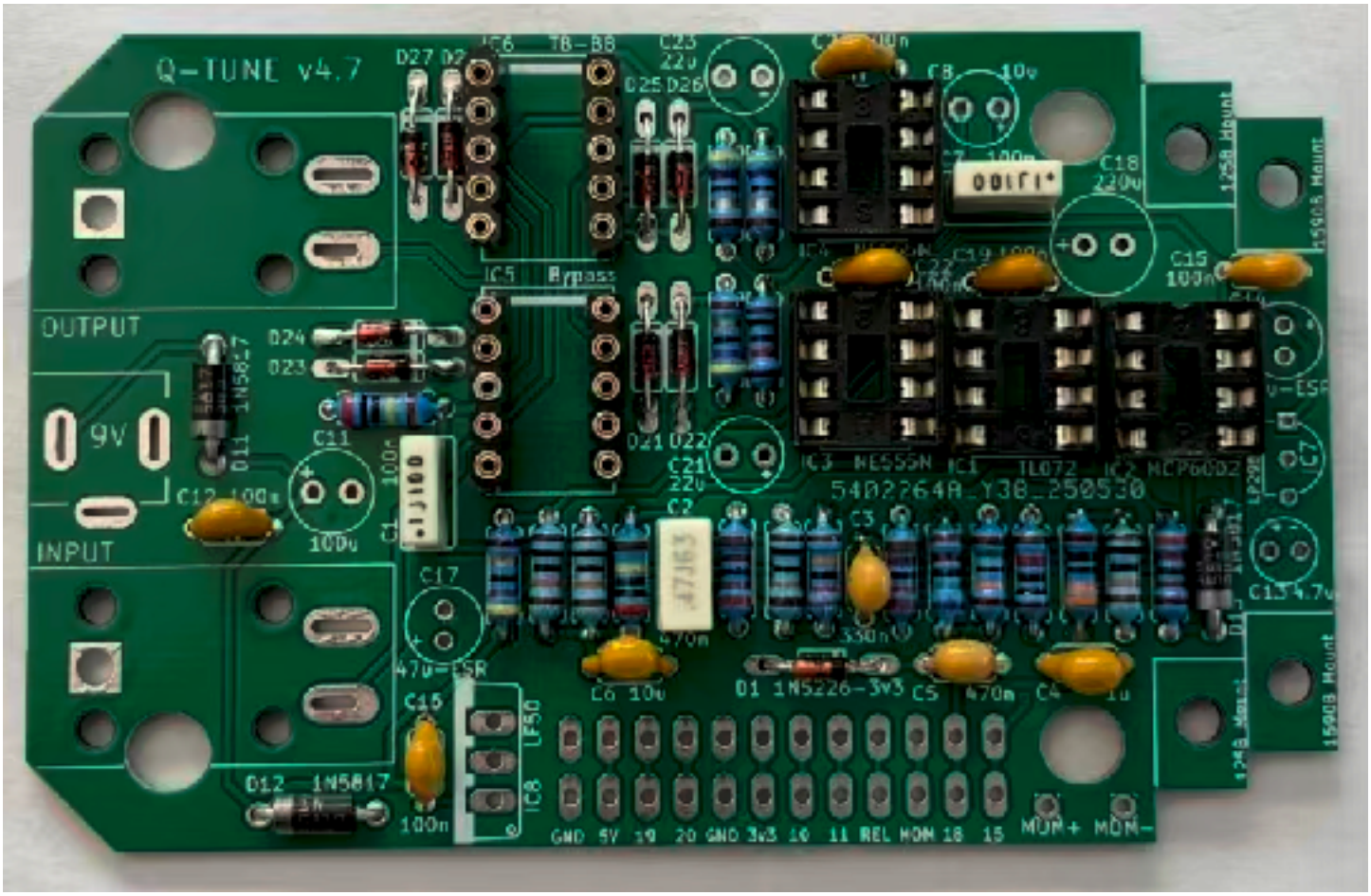


PCB: CAPACITORS (NON-POLARIZED)

After the sockets come the MLCC and the box film capacitors. These are several different heights, but there aren't as many, so you can do as many as you like at the same time, matching heights. Bend the leads at an angle to hold them in place before soldering.

MLCCs and box capacitors are not polarized so they will work in any direction. However, to keep things neat, it's recommended to put them all facing the same way and, if possible, with the value markings left legible after building.

Note: Depending on the type, the box film capacitors may have their value printed on either the top or the side. Usually the red ones have it printed on the side while the blue, yellow, or gray ones have it on the top.



PCB: LCD WIRES

This kit includes a JST 12-pin cable assembly that connects between the PCB and the ESP32 development board. Not all of the wires are used for communication between the two devices, but for better organization pads are provided for each wire. It's easiest to just solder all 12 wires.

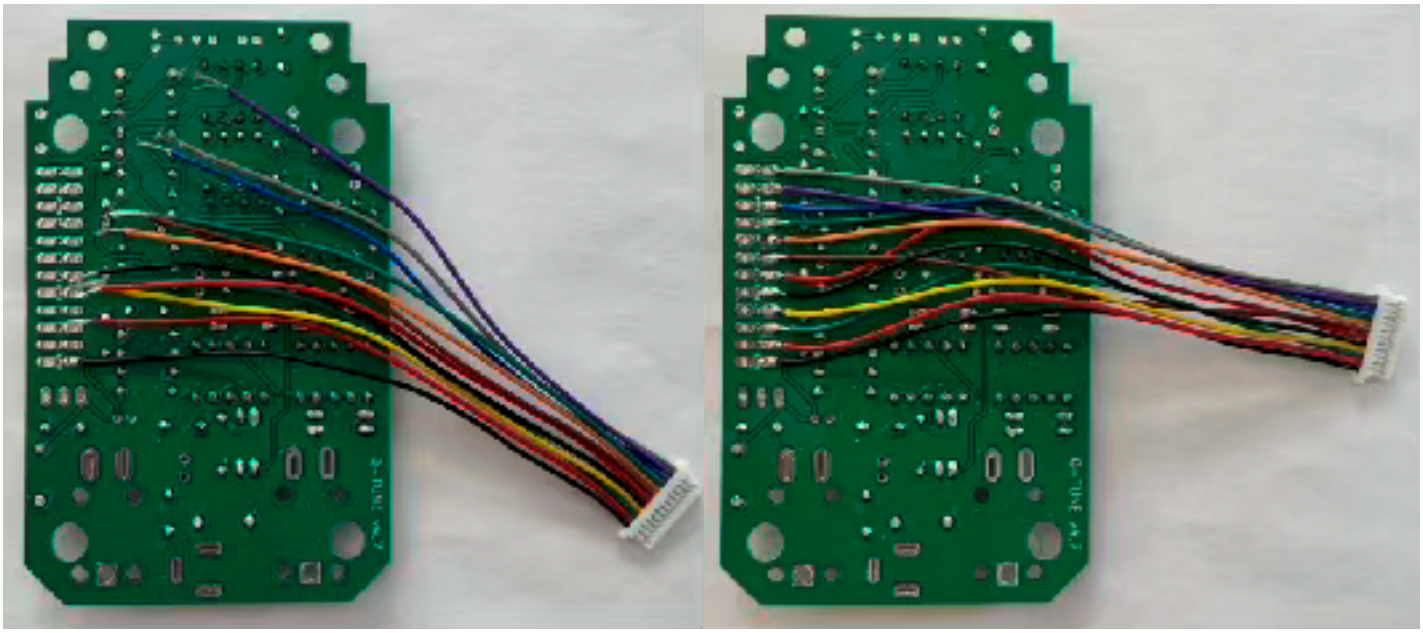
Preparation

Cut the wires so they are between 8-10 cm long; short enough to not be a nuisance but long enough so that with the PCB and ESP32 dev board disconnected, you can lay them next to each other "unfolded" like a book.

After cutting the wires, carefully strip each wire with about 1/8-3/16" (3-5 mm) of the wire exposed. Twist and (optionally) tin each wire before proceeding. The BLACK wire, is GND. The rest are in order.

Installing the Wires (on the BOTTOM of the board)

Placing each wire into the pads and soldering can be a bit tricky. To make the installation easier, you can (optionally) use some painter's tape to tape the wires down to your work surface; this forces the wires to be next to each other and neatly organized. Lift the taped wires from your work surface and carefully align each one in order into the pad holes. Use the holes closest to the inside of the board paying close attention to orient the colors with the corresponding pins on the ESP32 development board. The outer pads are for convenience in troubleshooting. Be sure to trim the wires after soldering so that none of them short out with each other.

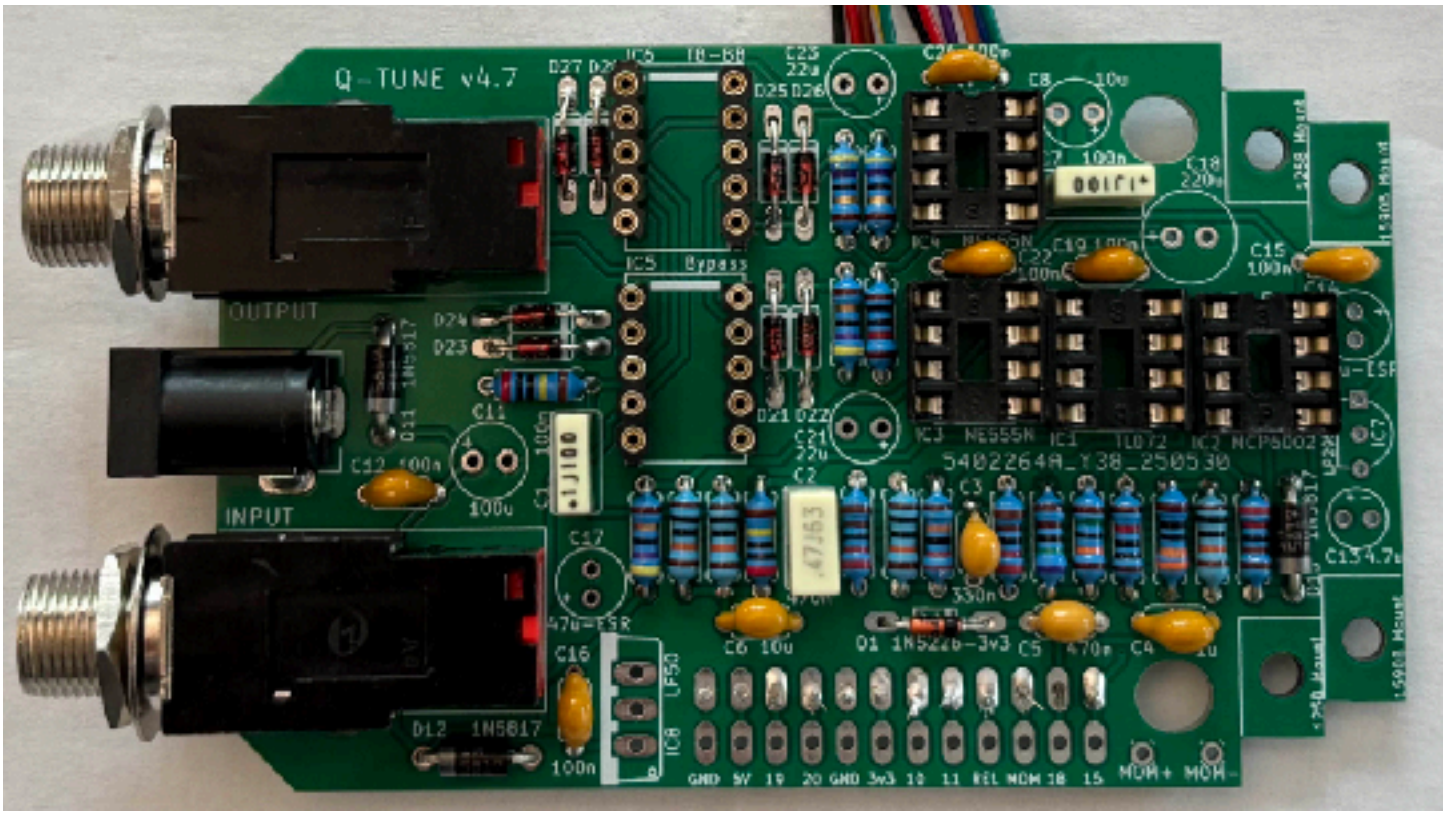


PCB: AUDIO AND DC JACKS

You're getting close! You can install these in any order, but it might be easiest to install the DC jack first. Install and align the DC jack as close to straight as possible. If you need help keeping it aligned you can use some painter's tape to hold it in place before you solder it.

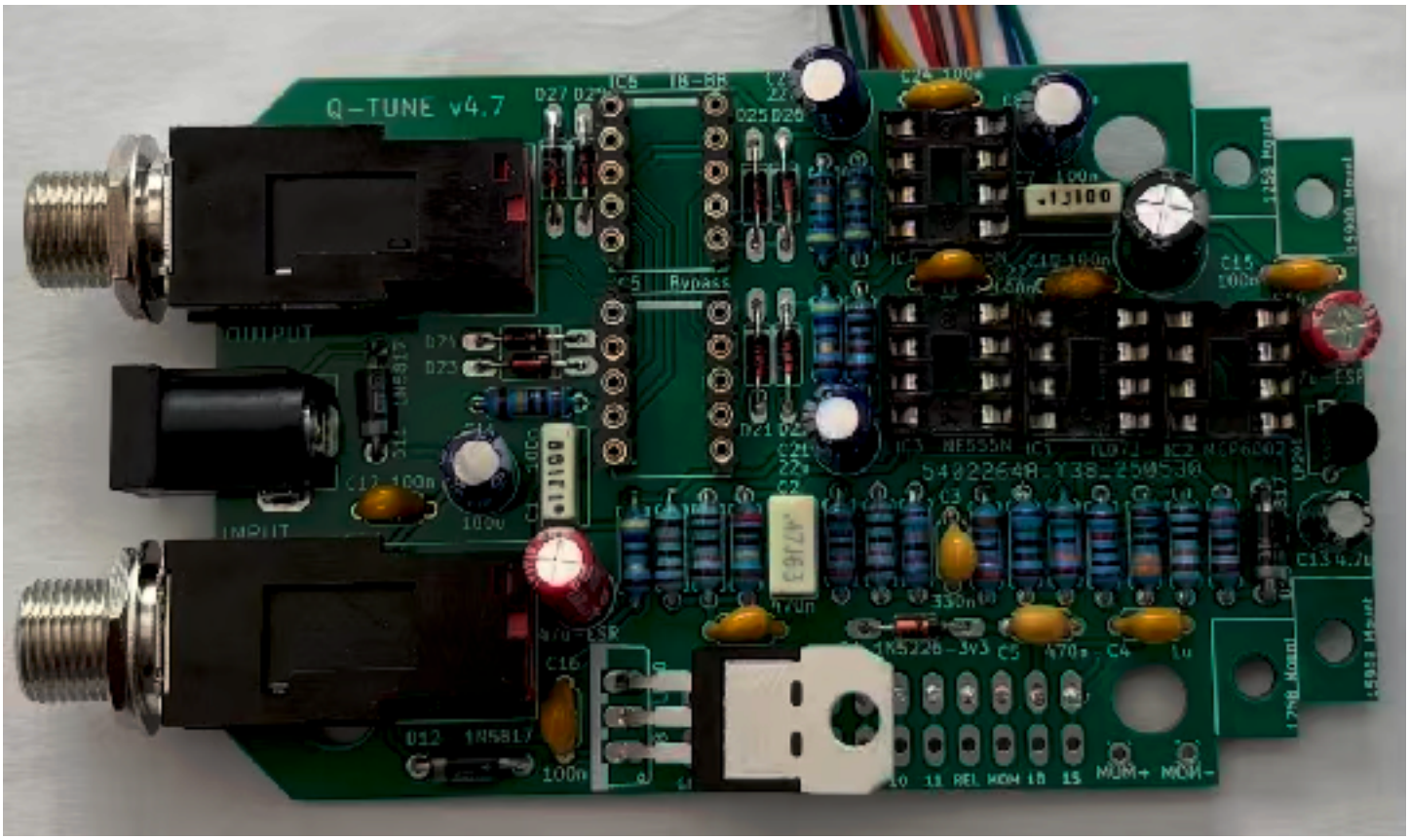
Get the two input/output jacks. The PCB has alignment holes for the 4 nubs sticking out of the jacks. Press each of the audio jacks in place and make sure they are pressed right up against the board before soldering. Solder one pin and double-check the jack is still pressed against the board before soldering the other two pins.

After soldering, clip the 9 pins on the back as close to the PCB as possible, else it will interfere with the screen during final assembly.



PCB: CAPACITORS (POLARIZED)

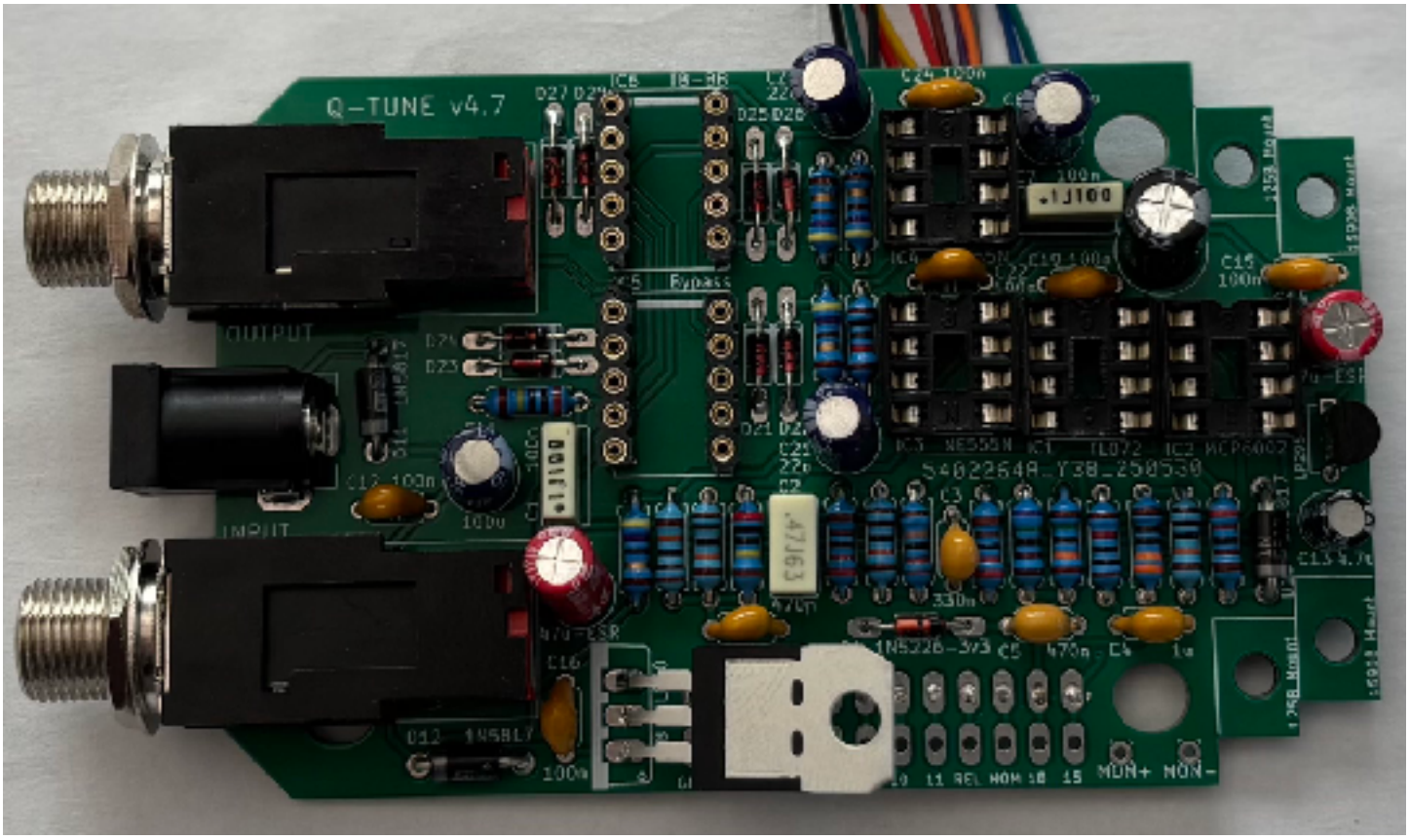
Populate the electrolytic capacitors. These are the tallest components so we save them for the last. They are polarized (i.e., they will only work in one direction) so pay particular attention that you install them correctly. The lighter vertical mark indicates the negative side. The longer leg is positive and fits in the square pad.



PCB: VOLTAGE REGULATORS

There are two voltage regulators used. The bigger one (IC8) is an LF50 used to power the ESP32 and LCD screen; it is rated for 500 mA. The ESP32 and LCD screen nominally requires ~130 mA. The smaller LP2950 (IC7) is used to power IC2 (MCP6002) that gains up the signal into the ESP32 for frequency detection. Everything else (IC1, IC3, IC4, IC5, IC6) are generally powered by the 9v input voltage.

Install both voltage regulators (notice the LP2950 says KY5050 on it). Make sure to fold IC8 down over the ribbon connections like shown.

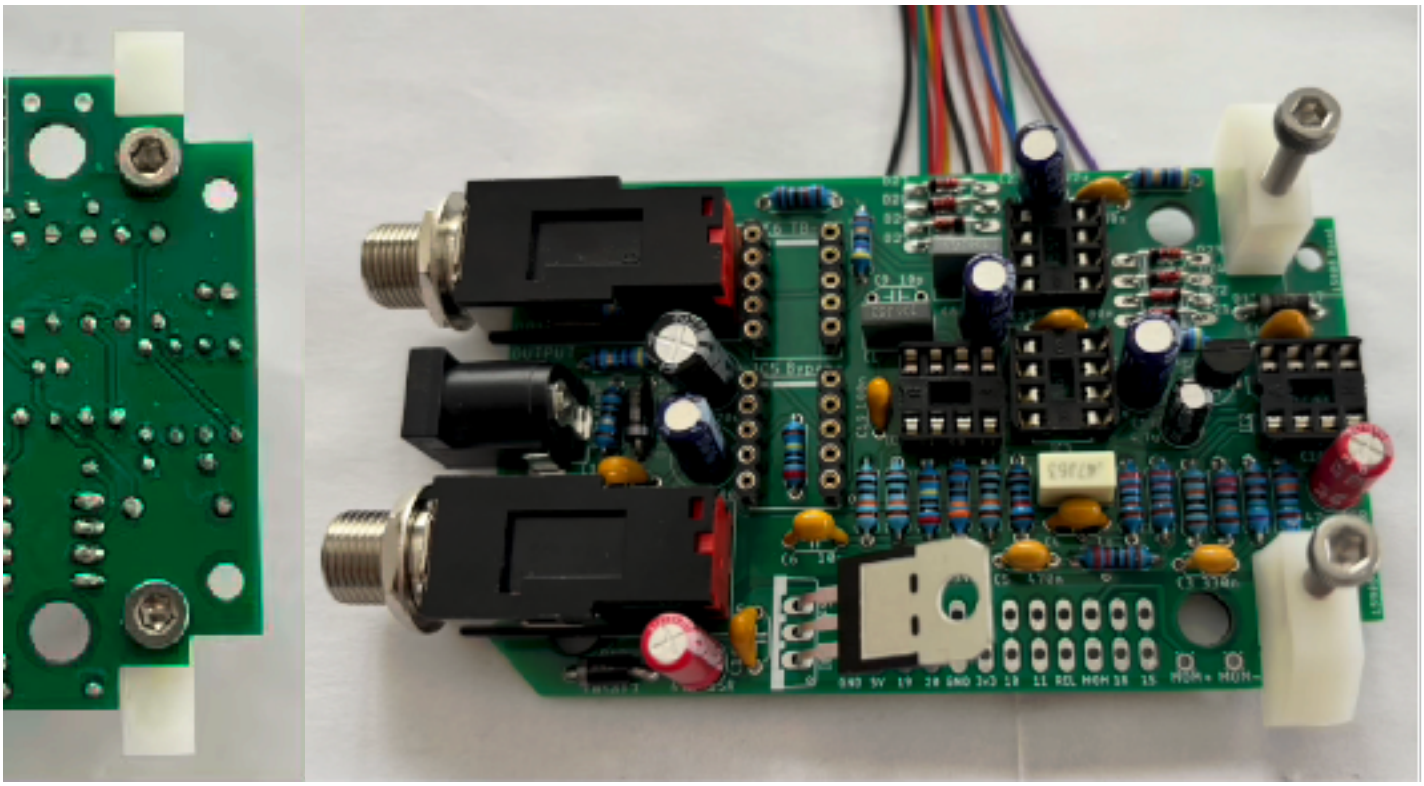


PCB: BOARD MOUNTS

The two nylon PCB mounts need to be prepared before using them. The mounting holes are not threaded. To cut new threads, use the provided hex wrench (or your own) and the larger M3 x 5mm screws. Install the screws by turning them clockwise into the holes as shown on both top and bottom of the nylon mounts

Now remove the screws and attach the mounts to the board as shown below. Depending on which enclosure you have, you will install the nylon mounts in a different location (1590B shown below).

Install additional screws into the top of the nylon mount. These will be adjusted later (125B) to press against the enclosure lid. For 1590B, screw them all the way in.

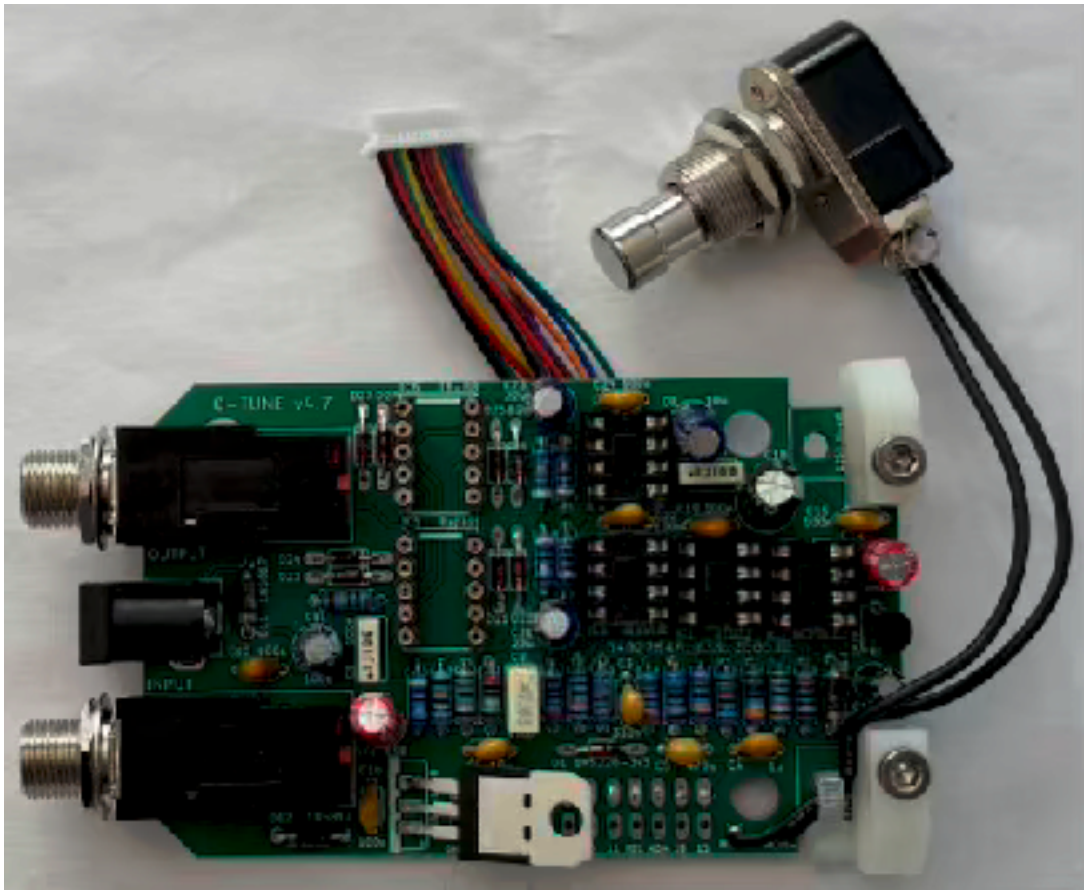


	Screws for Nylon Mount to bottom of PCB	Screws between Nylon Mount and Enclosure Lid
125B	M3 x 5mm long	M3 x 12mm long
1590B	M3 x 5mm long	M3 x 5mm long

MOMENTARY FOOT SWITCH

Prepare the two short wires provided for the momentary foots witch. Solder one to each leg of the momentary foot switch and the other sides to the top (this is important) of the PCB into the MOM+ and MOM– pads. The momentary switch is NOT polarized, either leg is fine. Then use the supplied tie strap to tie the wires to the unused mounting hole as a strain relief (these wires will break off at this junction eventually if not strain relieved). Nylon mounts for 1590b shown below.

Check the vertical placement of the switch in the enclosure and adjust the lower nut to end up with the height you prefer. A popular preference is to have the top of the shaft threads fully covered with the outside nut. This gives a nice and finished look.



DIAGNOSTIC POWER UP

We've intentionally left out the ICs and the relays and not connected the ESP32 LCD. It's time to do some basic checks with your multimeter.

Do not connect power to your PCB for these first tests.

Switch your multimeter to continuity mode and if possible, enable the alarm/beep so that when both probes are connected, you hear a sound. For these tests we want to make sure there aren't shorts (i.e., hearing a beep between these test points is bad).

1. Check neighboring pads on the X1 connector area. You shouldn't have continuity between any of the pads. If you do, double-check your work, fix your soldering job of the wires, and re-test.

2. Feel free to check other connections as you see fit using the schematic as a guide.

Connect 9V DC power (center pin negative) to the DC Jack and turn power on to the board.

With your ground probe connected to the GND of X1 (or any easily-accessible ground point, like either of the input/output 1/4" jacks), use the positive probe to check the following:

1. 9V on Pin 8 of each NE555 (IC3 & IC4)

2. 5V on Pin 8 of MCP6002 (IC2)

3. 9V on Pin 8 of TL072 (IC1)

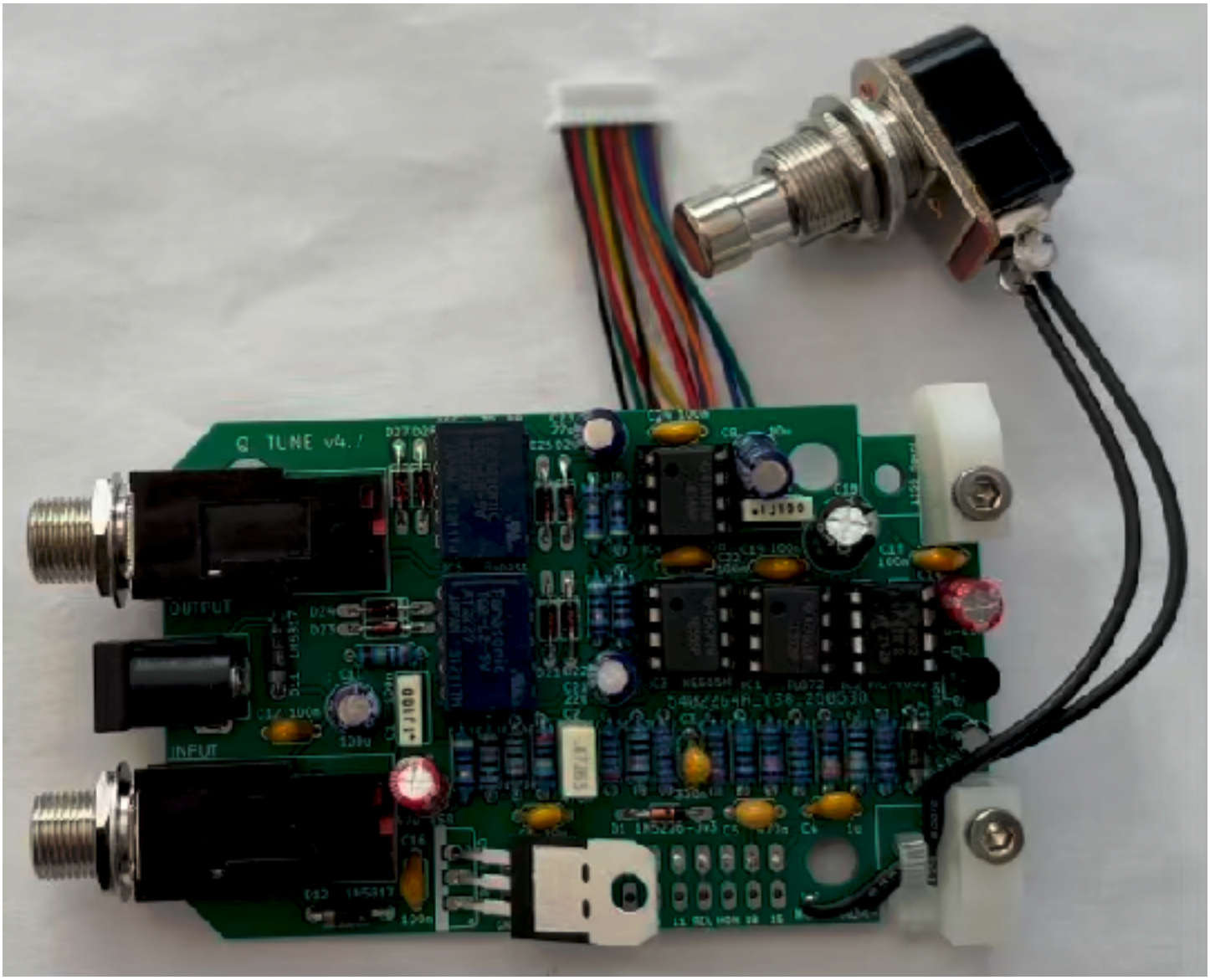
4. 5V on the X1-5V pad (which powers the ESP32)

If these tests all pass and you're seeing the proper voltages and connectivity, you're ready to move on.

ICs AND RELAY INSTALLATION

You're ready to install the ICs! Pay attention to Pin #1 and carefully line up the pins with the sockets. Make sure you take your time so you don't bend any of the pins and damage them. After they are aligned, carefully press them all the way down into the socket.

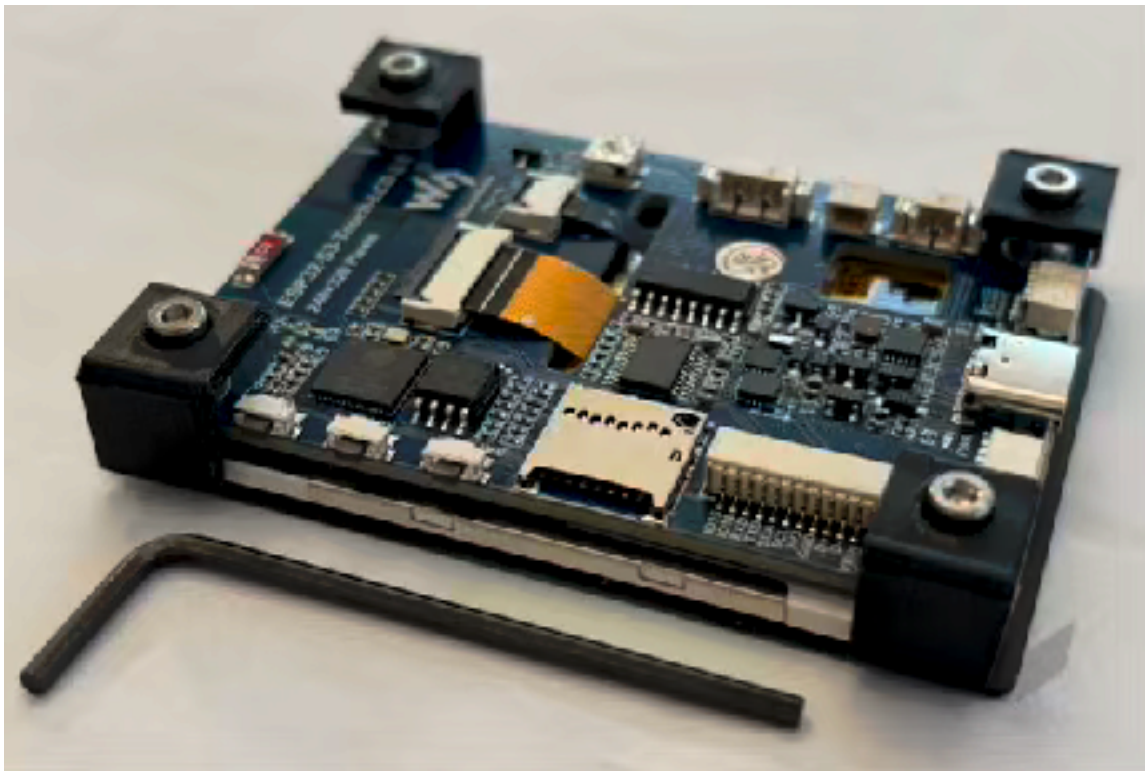
Install the two relays last and again, make sure the pins are lined up before pressing them down into the sockets.



ENCLOSURE INSTALLATION

Attach the 3D Printed standoffs to the LCD Screen

Using the 2.5 hex wrench, attach the 3D-printed standoffs to the LCD screen.



WARNING!

Carefully read and follow the instructions on the following page regarding the JST cable!

Connect the JST Wire Assembly

IMPORTANT: The pins inside the JST header on the back of the LCD screen can be bent or damaged easily. Please take your time to line up the JST wire connector and the socket. They are designed to only be attached in one orientation.

The provided ESP32 + LCD uses a JST SH 1.0mm 12-pin connector. Notice that the pins inside of the header socket are closer to the top of the header. Make sure that you have the cable assembly oriented and lined up in the same way. If you followed the previous steps when soldering the cables to the PCB the wires will be in the correct order.

With the cable assembly connector aligned as straight as possible and centered left and right, carefully press uniformly on both outer sides so that you push the connector straight in. If something doesn't feel aligned, **DO NOT force it as the pins can break**. Take your time, reset, and try again.

Screen and PCB Installation

You're so close now, but take your time on this step! The LCD screen and cover lens are not indestructible. **DO NOT FORCE** the LCD into the enclosure cutout at any point as you may risk chipping, cracking, or otherwise damaging the screen.

Carefully place the screen into the enclosure opening with the USB-C port pointing towards the opening in the enclosure. Ensure the screen is placed properly and then slide the PCB into place by pushing the audio jacks through the holes in the enclosure.

For a detailed view of how this is done, view this video:
https://www.youtube.com/watch?v=bvPk-TBj-Ws

Momentary Switch and Audio Jacks

Install the momentary foot switch by placing the flat washers on the outside of the enclosure and then tighten the nuts with a properly-sized socket.

FINAL TESTING & ASSEMBLY

Woohoo! You've made it and are ready to power on your tuner for the first time. Before assembling the lid onto the enclosure let's make sure everything is working.

Connect a guitar cable to the input and output with the output running to an amplifier. Connect a 9V power source (center negative) to the DC jack.

Test the Bypass Relay

Press the foot switch to go into bypass mode and you should hear a good clean signal into your amplifier. This test confirms your bypass relay switch is properly working.

Test the Tuning

Press the foot switch again to return to tuning mode. Pluck a string on your instrument and make sure the needle of the tuner is picking it up.

Navigating User Settings

Even though it's easiest to **use the touch screen** to enter Q-Tune's settings (by tapping the Q logo), Q-Tune is designed to be fully operational just with the momentary foot switch.

Momentary Foot Switch Actions

Long Press
- Enters user settings from the tuning screen
- Also activates the currently-highlighted menu item

Single Press
- Advances forward one menu item

Double Press
- Advances backward one menu item

Test the Bypass Mode Relay

Enter user settings. Go into the **Tuner** settings screen. Advanced to the **Bypass Mode**. The tuner will unmute when you enter this screen so you can hear the difference between True & Buffered Bypass.

The buffer is designed to be as transparent as possible so if you only have the tuner plugged in with a short cable, you may not hear any difference. If the tuner is on a pedalboard as the first pedal with many following, you're more likely to hear the benefits of switching to Buffered Bypass mode. Hearing that difference in that case confirms the True/Buffered Bypass relay is working properly!

CONGRATULATIONS!

You've successfully built your very own instrument tuner pedal! Explore the settings and make it work just like you want.

We'd love to hear from you and your experience building the pedal. Take a picture and/or video of your build and tag us in your social posts on YouTube, Instagram, or whatever your preferred platform is. Use the #qtunepedal hashtag.

Thank you for choosing Q-Tune! We hope you'll find years of enjoyment and stable tuning.

–Boyd & Ryan
www.q-tune.com

ADDENDUM

Op Amps

Two op amps are used in the pedal. The first is a two-channel op amp for unity input and the buffer. The second is an op amp for gain into the ESP32.

The provided op amps provided work great, but you may wish to experiment with alternatives.

Input and Buffer

MPC6002
Default for Input and Buffer circuit. 6v maximum supply.

MCP6022
Lower noise alternative.

Gain into ESP32

TL072
Default for Gain into ESP32. A great all around op amp. 6v minimum supply.

TLV2372
Improved headroom over TL072.

OPA2134
Lower noise than TL072, similar headroom.

OPA1642
Lower noise than TL072. SOIC-8 only.

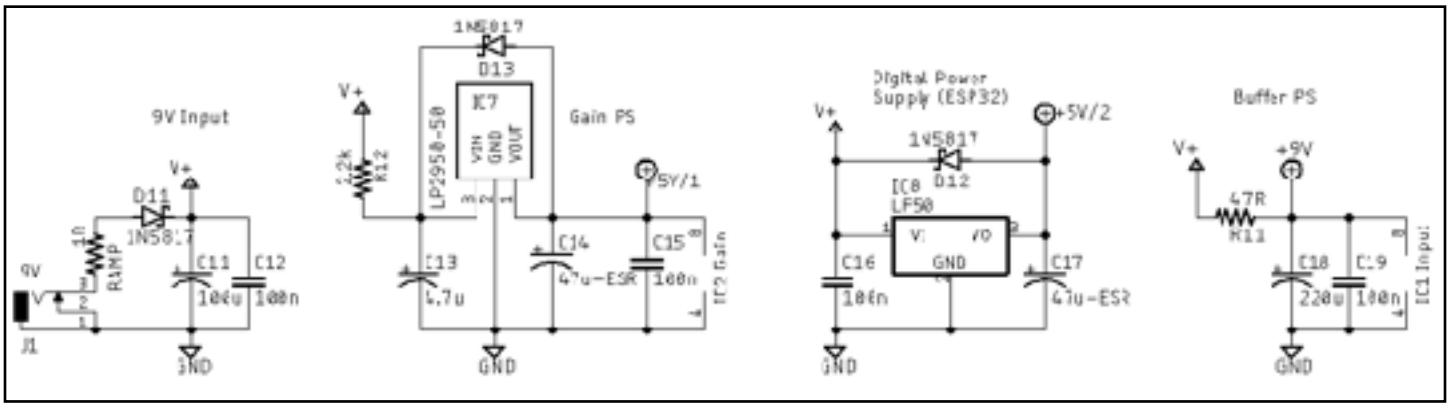
OPA1611
Ultra-low noise. SOIC-8 only.

Op Amp	Noise (nV/sqrtHz)	Headroom (V)*	~Cost (each)
MCP6002 @ 5v	28	5.6, published	$0.40
MPC6022 @ 5v	8.7	5.6, published	$1.80
TL072 @ 9v	18	7.68, measured	$0.60
TLV2372 @ 9v	39	8.80, measured	$1.30
OPA2134 @ 9v	8	6.72, measured	$5.00
OPA1642 @ 9v	5	5.84, measured	$2.00
OPA1611 @ 9v	1.1	5.0, published	$4.20

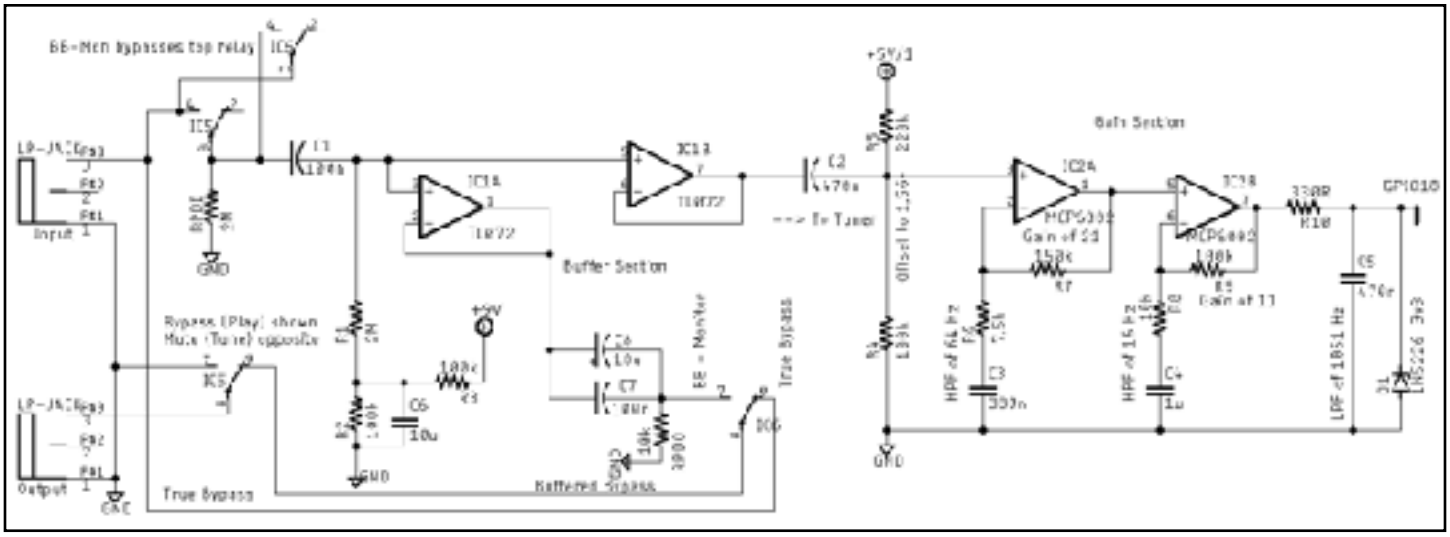
*Most guitar pickups output <1 Volt peak-to-peak.

SCHEMATICS

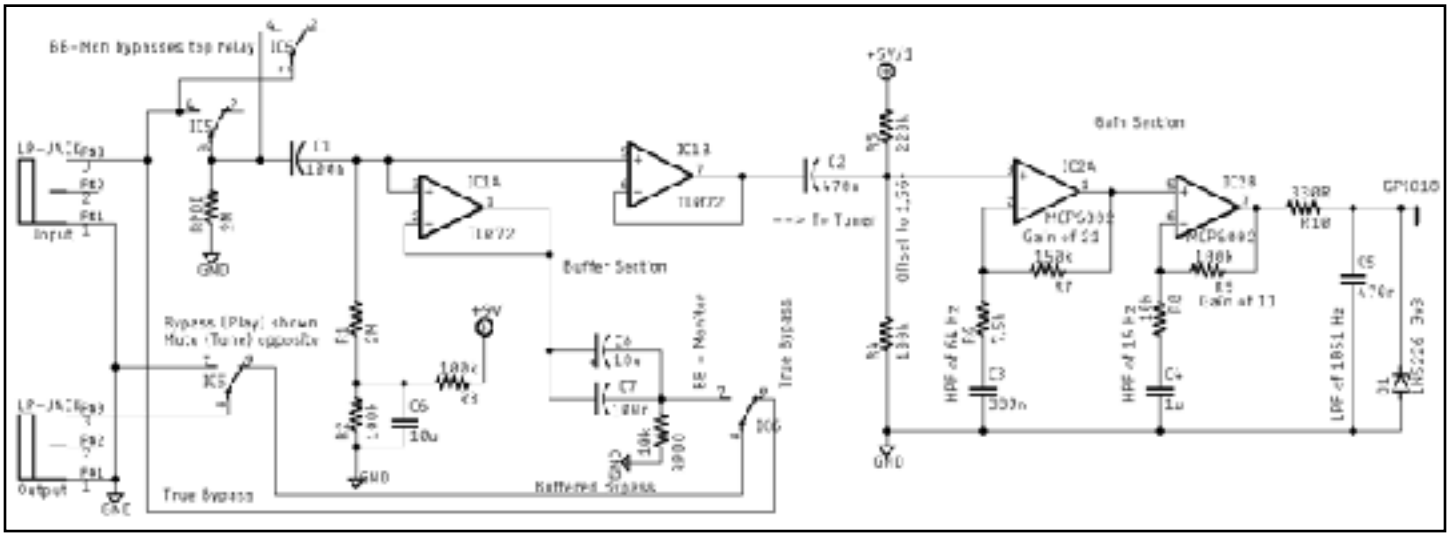
Power Supply



Relays & ESP32 Input



Signal



TROUBLESHOOTING

If you finish building the kit and find that it doesn't work right, visit the Troubleshooting page on the website.

SUPPORT

We do not offer direct support for Q-Tune beyond the provided documentation. Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error or that the included components are non-functional.

Where to get help

The best place to seek help is the Pedal PCB Forum's Third Party area designated specifically for Q-Tune: https://forum.pedalpcb.com/forums/qtune/

When posting a troubleshooting request, always include the following:

1. A thorough description of the problem you are experiencing

2. A photo of the inside of the pedal

3. A list of all the measured voltages of each of the pins, described on the **DIAGNOSTIC POWER UP** page

While we cannot offer direct, private support, we are definitely interested in you having a successful experience and a working tuner, so please reach out in one of the ways we mentioned above. It benefits the entire community when troubleshooting processes are public because then people who have the same issue in the future may come across it when searching. When you do get help, make sure to pay it forward! The best way to learn new skills is to help others. even if you've only built one tuner, you have more experience than someone who is brand new, so you have something to offer.

ACKNOWLEDGEMENTS / LEGAL

These kits are intended to be built by the customer. Molinello Music is not responsible for language that may be used by the customer in the marketing or resale of the finished product.

Any use of trademarks is for comparative advertising purposes only under fair use. It is not an endorsement of this product by the trademark holders.

The basic structure of this document is patterned after the excellent kit documentation made by Aion FX and borrowed with permission. Aion FX is otherwise not affiliated or a sponsor of this project. All additional content and graphics in this document are copyrighted by Molinello Music and may not be used without permission.

By using Q-Tune, you are expressly agreeing to the End User License Agreement published on the Q-Tune website available here: https://q-tune.com/eula

DOCUMENT REVISIONS

1.0.3 (06 June 2025)
- Adjusted text in the intro of the build doc.

1.0.2 (05 June 2025)
- Updated pictures for v4.7 PCB
- Added capacitor identification graphics
- Use clear dates for document revisions

1.0.1 (01 June 2025)
- Removed a blank page

1.0.0 (01 June 2025)
- Initial release