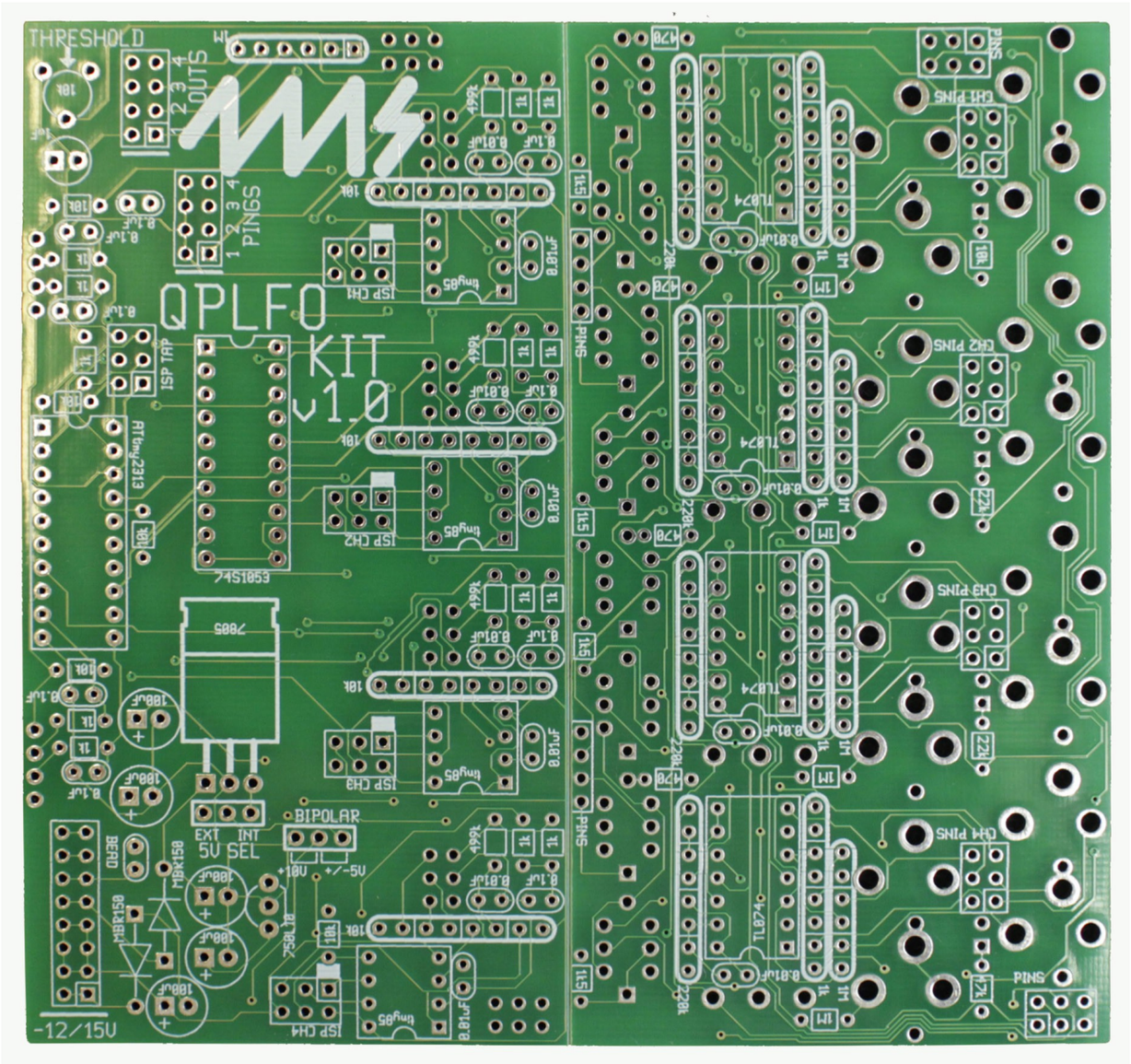
Quad Pingable LFO

Kit Builder's Guide for PCB v1.0

4mspedals.com



QPLFO

This guide is for building a Quad Pingable Low Frequency Oscillator (QPLFO). It's an intermediate-level kit. You should have basic soldering skills, and a basic familiarity with identifying electronics components. *Note:* The QPLFO PCB is actually two PCBs that are V-scored so they can be snapped apart. It's easiest to assemble the board in one piece, and then snap the boards apart afterwards.

Tools Needed:

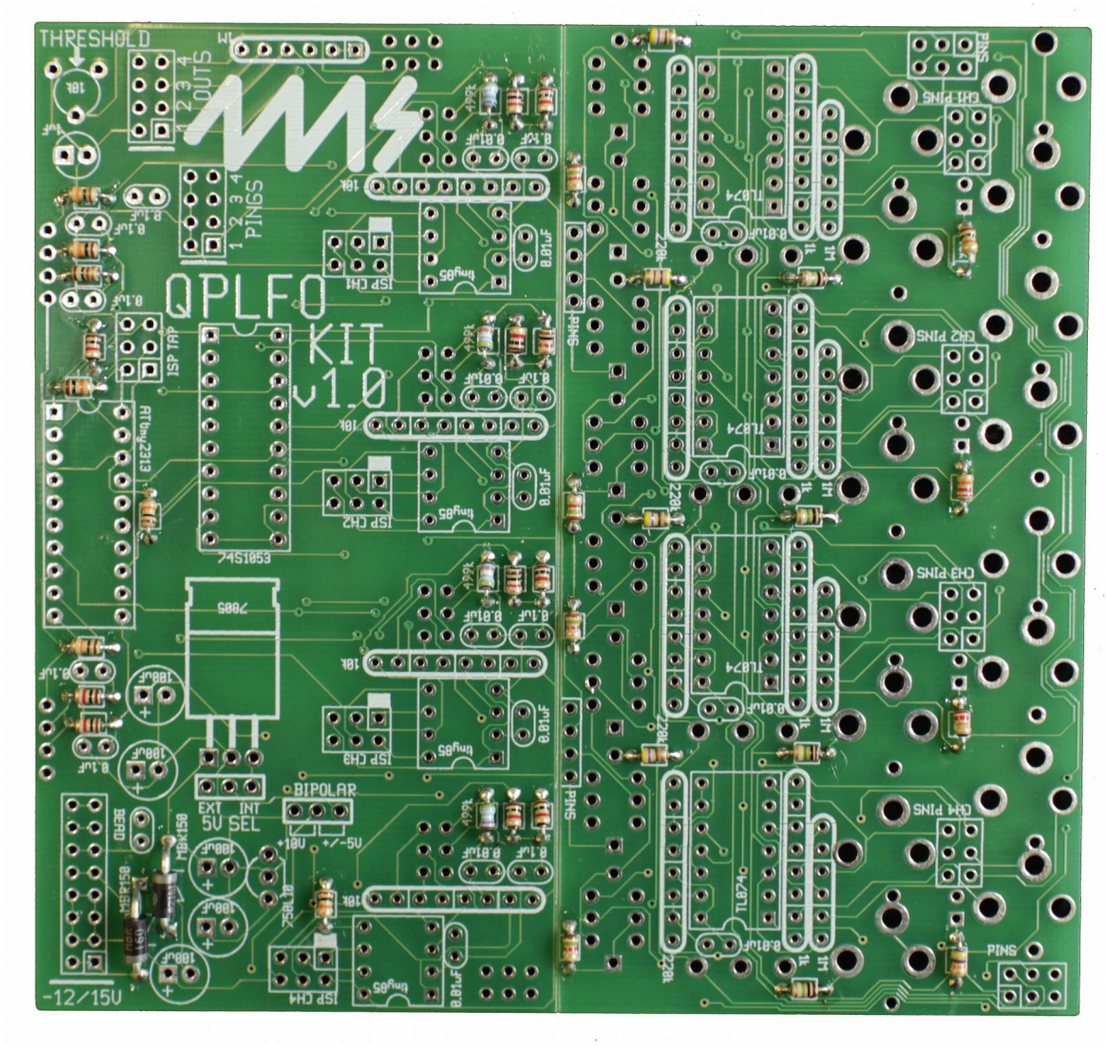
- Soldering iron, solder
- Flush snips
- Needlenose pliers (for removing a component if you make a mistake)
- 5/16" and 3/8” socket driver (optional: pliers will work too if you're careful)
- Multimeter (for reading resistor values if you don't know the resistor color code chart)

Step 1: Resistors & Diodes

Insert and solder the 38 resistors and 2 diodes. There are thirty-four 5% resistors (tan body) and four 1% resistors (blue body). Due to variations in components, you may find that some resistors are larger than others in your kit. The size is irrelevant, only the color bands matter. The diodes must go in with the black or grey band facing the tip of the arrow. (see photo in step 4 for a more detailed view of this orientation)

After soldering, snip the leads nearly flush to the PCB (you'll be snipping the leads flush on all the components after soldering).

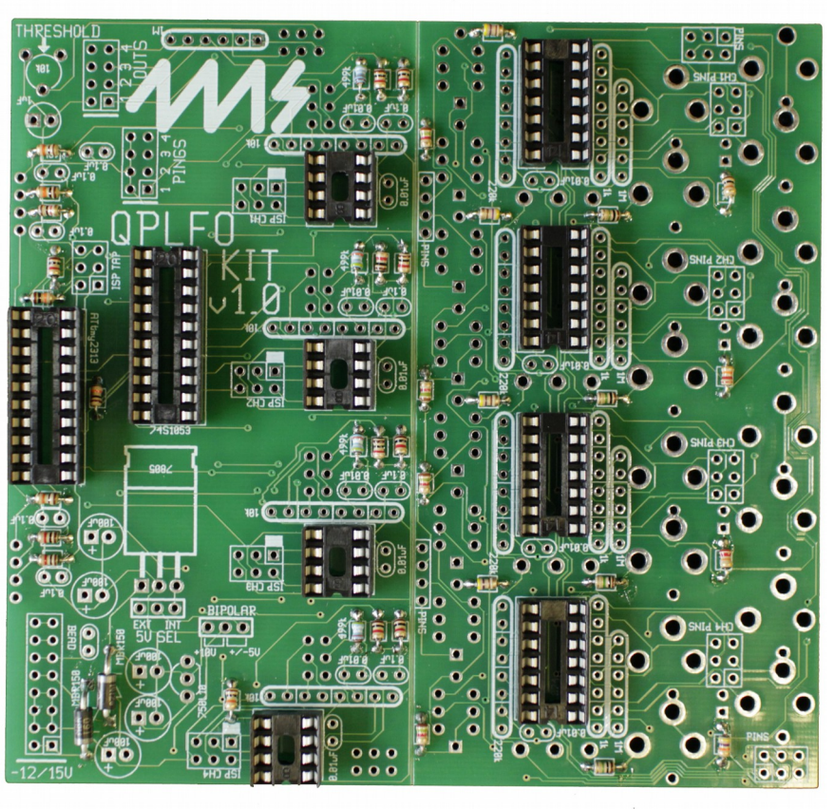
The 34 tan resistors (5%):	The 4 blue resistors (1%):
• *470 ohm x 4 (yellow purple purple gold)* • *1k x 13 (brown black red gold)* • *1k5 x 4 (brown green red gold)* • *10k x 6 (brown black orange gold)* • *22k x 2 (red red orange gold)* • *47k x 1 (yellow purple orange gold)* • *1M x 4 (brown black green gold)*	• *499k x 4 (yellow white white orange brown)* ***These are the only blue resistors in the kit*** The 2 diodes: • *MBR150 x 2 (black body)* ***Note orientation! Band=arrow tip***



Step 2: Sockets

Insert and solder the IC sockets. The notch in the sockets should line up with a matching notch drawn on the PCB.

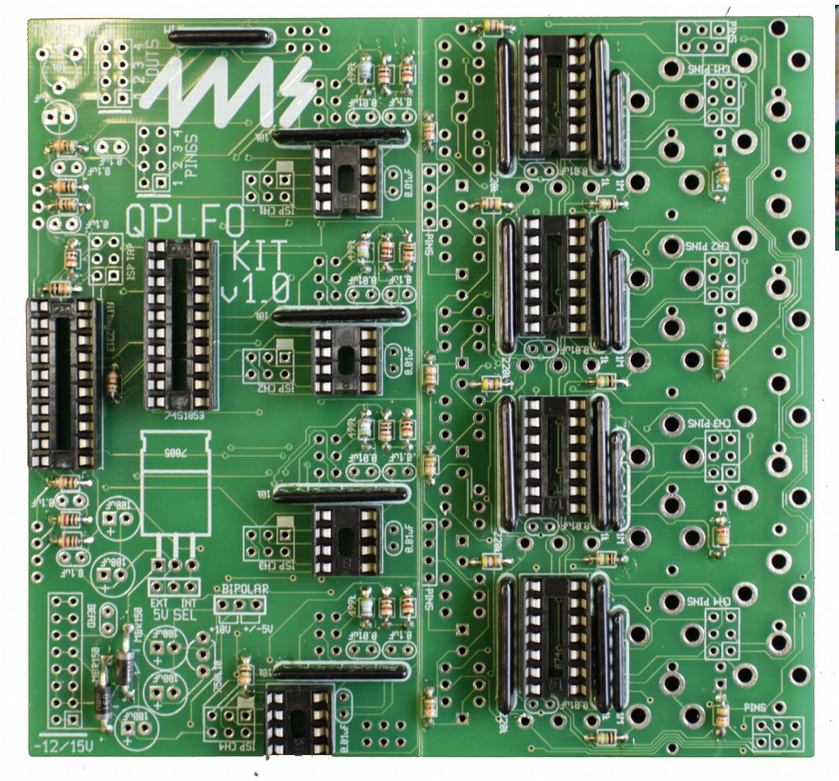
- *8 pin socket x 4*
- *14 pin socket x 4*
- *20 pin socket x 2*

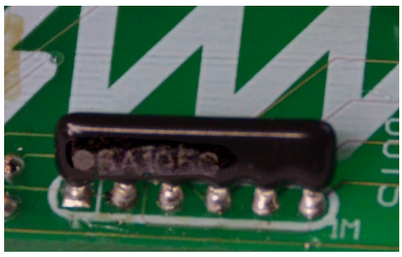


Step 3: Resistor Arrays

Insert and solder the 17 resistor arrays. All the arrays, except for one, can go in either way. The 1M bussed array on the top board (6A105G/L61S105) must go in with the dotted pin in the square hole (see photo). The dot can be seen over the leftmost pin if you hold the array so that you can read the writing.

- *1k isolated 8-pin array x 4 (“8B102G” or "L83S102...")*
- *10k isolated 8-pin array x 4 (“8B103G” or "L83S103...")*
- *220k isolated 8-pin array x 4 ("8B224G" or “L83S224”)*
- *1M isolated 6-pin array x 4 ("6B105G" or “L63S105”)*
- *1M bussed 6-pin array ("6A105G" or “L61S105”)*



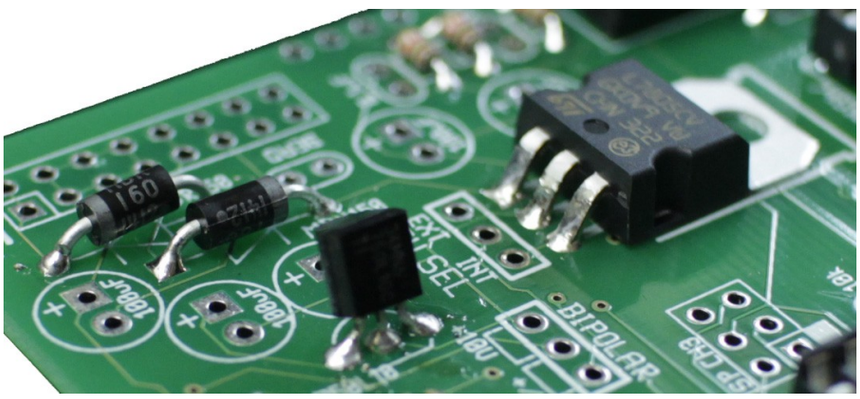


Above: 1M bussed 6-pin array, note the white dot oriented with the square hole on the PCB labeling

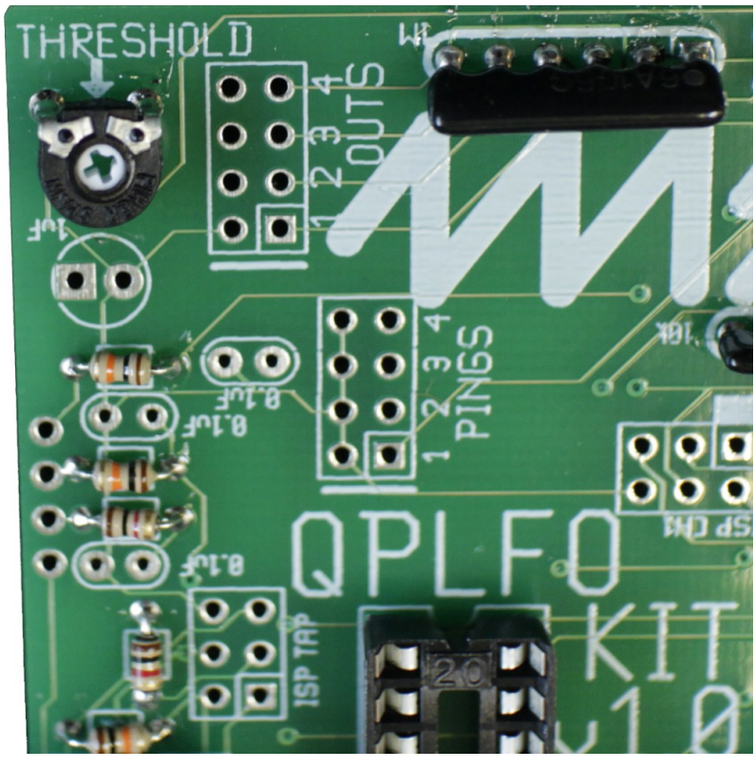
Step 4: Voltage Regulators, and Trimpots

Next, insert and solder the two voltage regulators (7805) and (750L10). For the 7805, bend it down flush to the PCB as shown in the photo. The words must be facing upwards. Insert the 750L10, the orientation is crucial: the curved and flat side must match the outline drawn on the PCB, this component will be facing to the right. Finally, insert and solder the 10k trim pot.

- *7805 voltage regulator – Bend down flush to PCB x 1*
- *750L10 x 1*
- *10k trim pot x 1*



Above from left to right: MBR150 diodes, 750L10 regulator, and 7805 regulator



Above: 10k trimpot in the upper-left corner

Step 5: Male Header pins

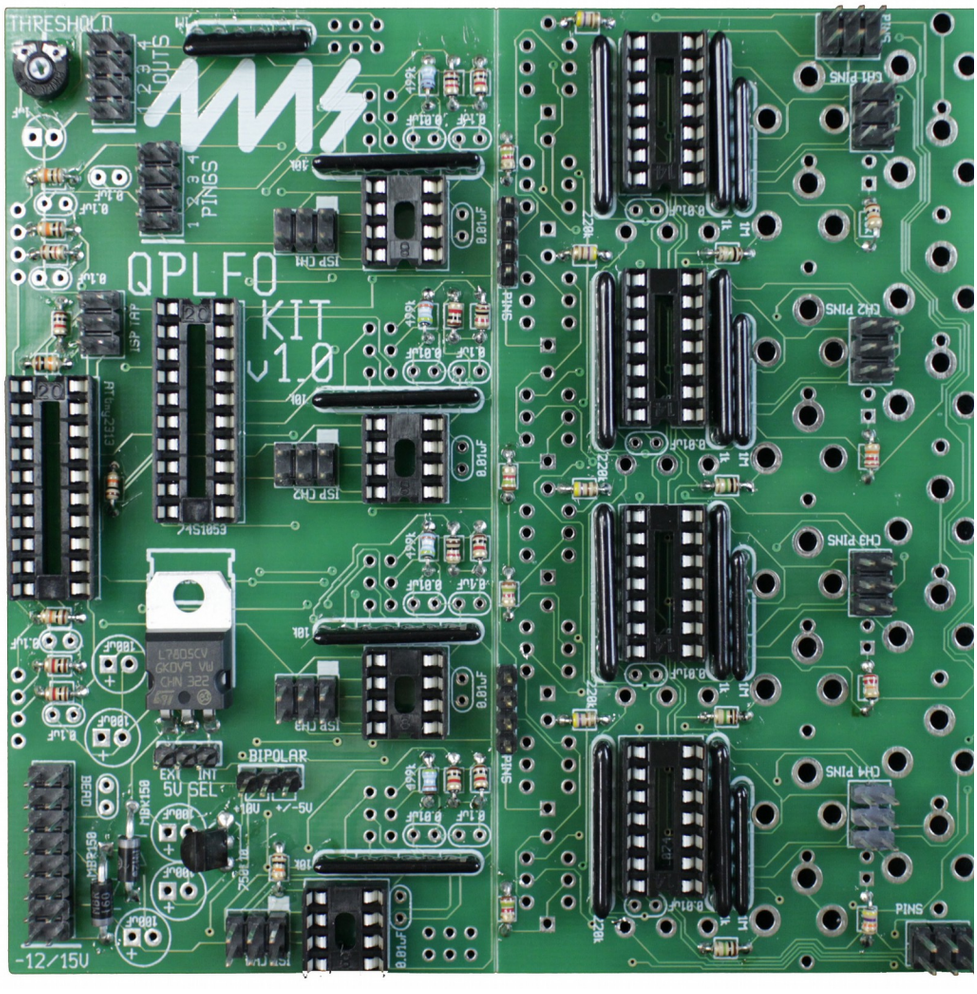
Insert and solder the male header pins. Since the headers fall out easily, it helps to put a piece of cardboard or a book over the PCB, then flip over the book and PCB together before soldering.

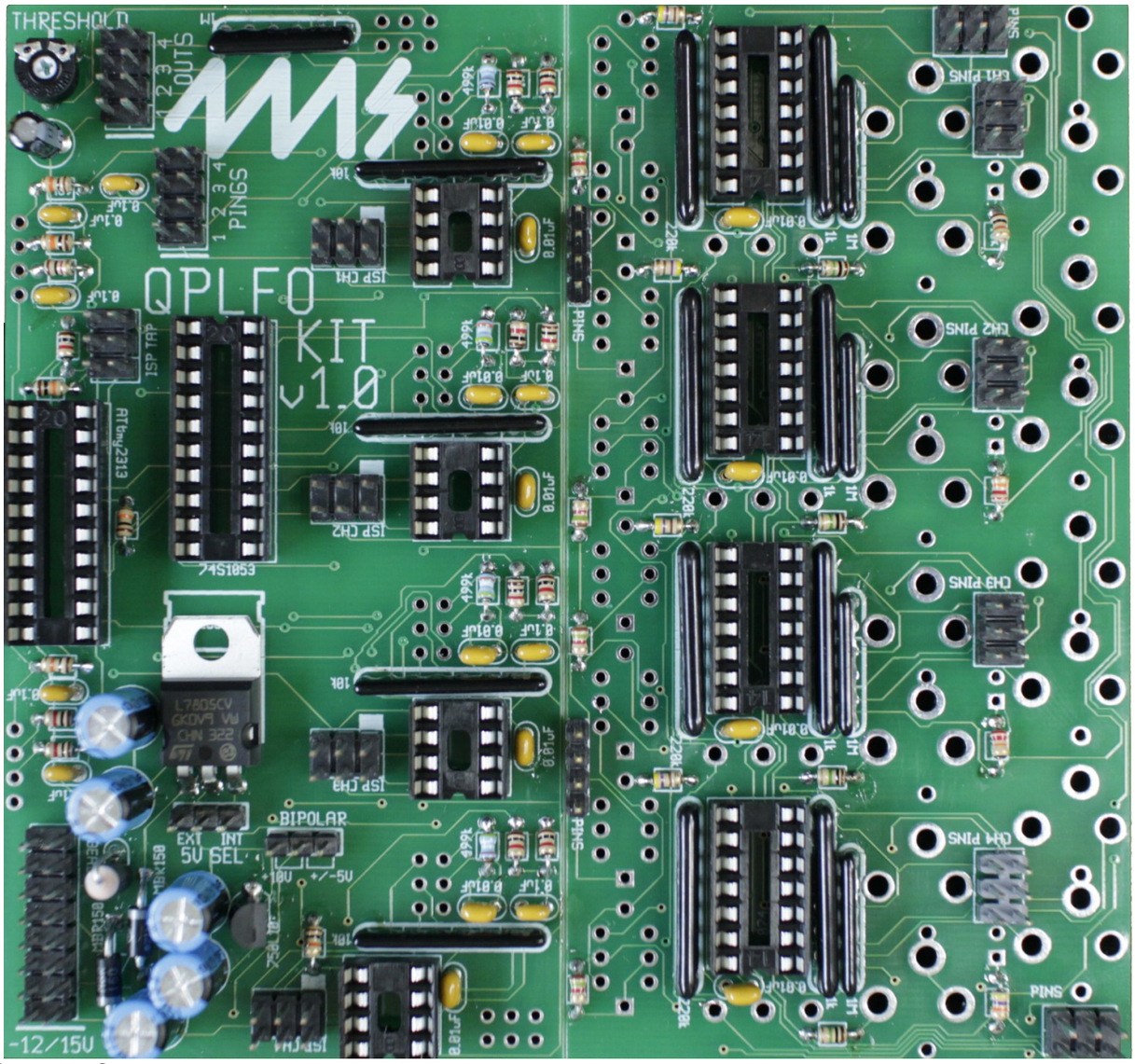
Peek ahead to Step 7: Don't accidentally put male headers where females go!

Make sure the header pins are flush and vertical-- your QPLFO won't fit together if the header pins are not perfectly vertical! Solder one or two pins per header, then flip the board back over and check to make sure they are lined up, flush to the PCB, and the pins are at a perfect right angle to the PCB. When you verify this, flip the board back over and solder the rest of the pins.

Right:

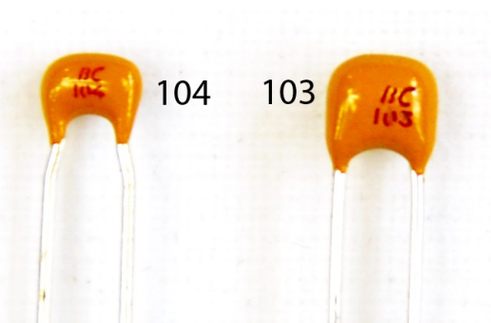
- *2x8 headers x 1*
- *2x3 headers x 11*
- *2x4 headers x 2*
- *1x4 headers x 2*
- *1x3 header x 2*



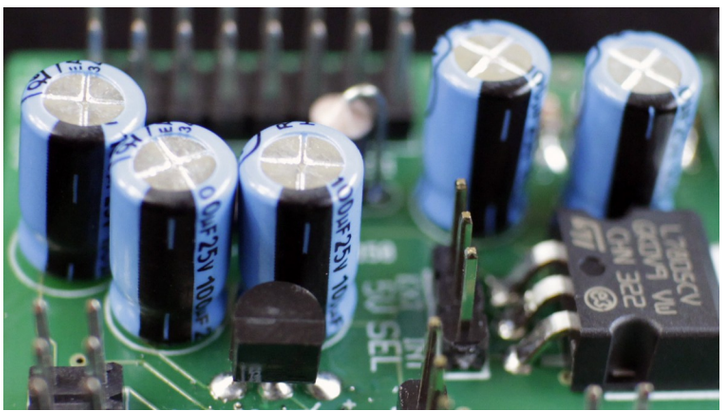


Step 6: Capacitors and Ferrite Bead

Insert and solder the 27 capacitors and the ferrite bead. The 0.1uF, and 0.01uF caps can go in either way, but the 1uF, and 100uF cylinder caps must go in with the long lead in the square hole. The 1uF, and all five 100uF caps are orientated the same way, with the striped side to the right. The ferrite bead can go in either way, but one lead must be bent 180° so that it stands up.



Above left: the 0.01 uF and 0.1uF caps are labeled "103" and "104" respectively on their faces. While they look very similar, the 0.1uF (on the left) is slightly smaller.



Top:

- *0.01uF (103) x 12*
- *0.1uF (104) x 9*
- *100uF x 5*
- *1uF x 1*

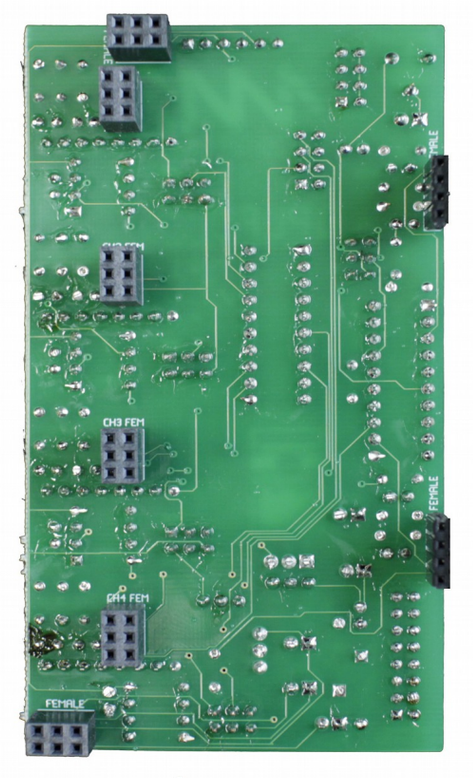
Above right:

- *100uF x 5 (note the stripe orientation)*
- *Ferrite bead x 1 (standing vertical, middle background, behind 100uFs)*

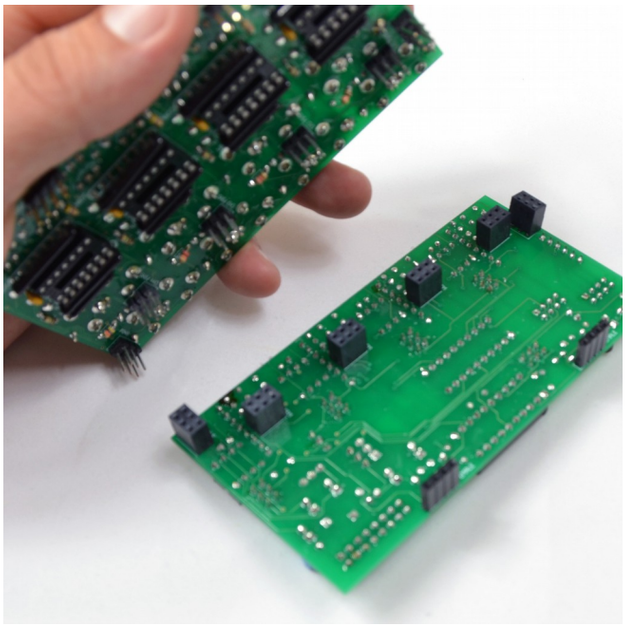
Step 7: Female headers

A. First, snap your boards apart. They are scored down the length, and you can just snap them apart with your hands.

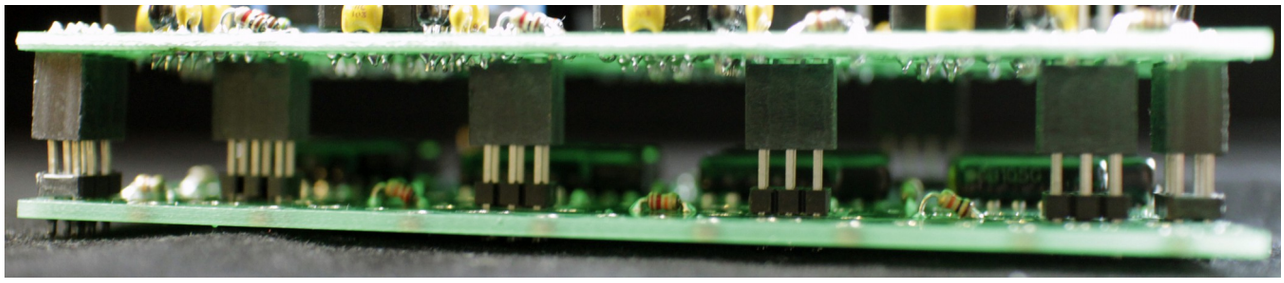
B. Flip the smaller board over so the resistors, sockets, etc are facing down. Insert the 8 female headers as shown. **Do not solder yet!**



C. Lay the larger board on top of the smaller board so that the 8 header pins line up with the female headers.



D. Gently press the boards together. Verify that everything is straight and none of the headers are at an angle, and no pins are sticking out.



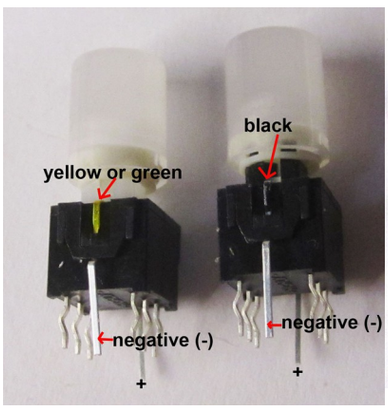
F. Now, squeezing the boards together so the female headers don't fall out, flip the boards over and solder the female headers

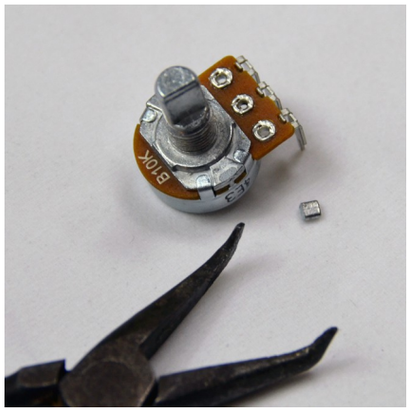
G. Take a minute to carefully inspect the bottom (control) board. Make sure every component is soldered, and that you didn't miss a joint. It's especially easy to miss a joint in the rows of header pins. **After the next step, locating and fixing a problem will be harder, so take the time now to check everything over.**

Step 8: Prep mounting

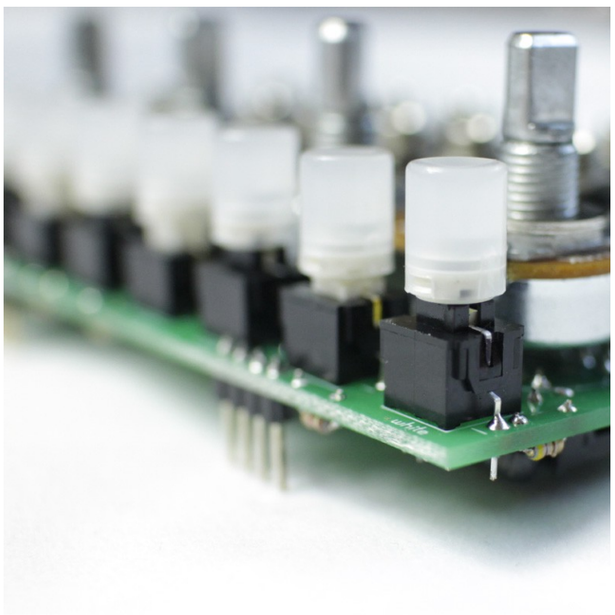
A. Put a hex nut on each of the 16 jacks. Finger tighten the nut (don't use a wrench). Insert the 16 jacks into the PCB. Do not solder yet.

B. Bend the tabs off of the pots (see photo below) with your needlenose pliers. Place two washers on each potentiometer (see below right photo), this will ensure proper spacing, and tightening during the mounting process. Insert the 4 pots into the PCB. Do not solder yet.





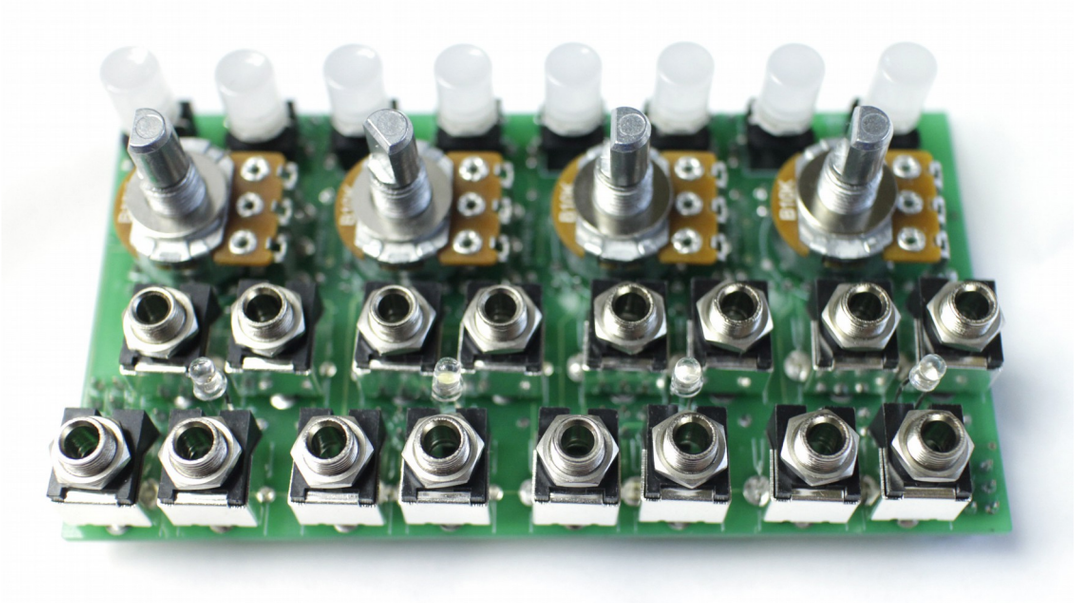
C. Look closely at the 8 buttons. Four are marked with a yellow mark and four are marked with black. See the photo on left. The side with the black or yellow mark is the negative side. On the PCB, this side is marked with a minus sign (-) as well as the name of the color that should go in that hole**.** Insert the buttons into the PCB, being careful that they go in the right way and that no pins are bent.



D. Verify your LED buttons are in correctly. It's very difficult to remove a button that's backwards or is the wrong type. See photos. Check carefully.

E. Notice the LEDs come color coded on a piece of paper. The one with milky off-white head is white, and the other three have clear heads. If these three become separated from the colored sheet they will be nearly impossible to discern from one another, so make sure to remove one LED at a time. Insert the LEDs into the PCB where marked. The longer lead goes in the square hole.

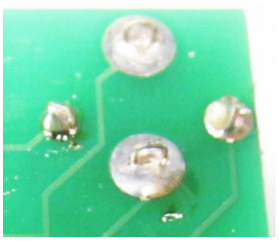
F. After inserting all 4 LEDs, verify the long lead is in the square hole. This is easy to mix up, and hard to fix later.



Step 9: Mounting part 1

A. Push the LEDs down close to the PCB. Take the panel and gently lower it down on the PCB. Wiggle it slightly so that it fits over the pot shafts, then the buttons, then the jacks.

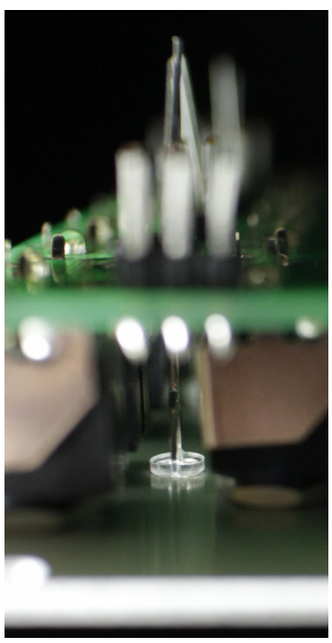
B. Once the panel is on, hold everything together with one hand while putting a nut on the corner jacks, and potentiometers with the other hand. This is tricky and you may drop the nut a few times: don't worry, this is the hardest part. Tighten the nuts down **slightly (1/8 turn)** with a 5/16" (jacks) and 3/8” (pots) socket or pliers (be careful not the scratch the panel!). See photo on next page.

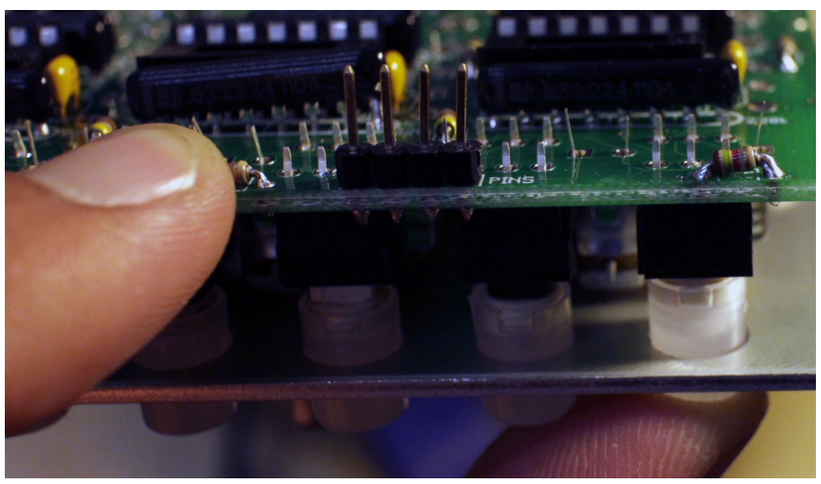


C. Flip it over and make sure there are no gaps between the metal “jacket” around the jack and the PCB. Also, make sure the PCB is as close to parallel with the panel as possible. Now solder the corner jacks (see photo to the right) and one tab on each corner potentiometer (see photo in 10b).

Above: Step 9B, notice the "corner" nuts on two of the jacks, to hold the panel in place

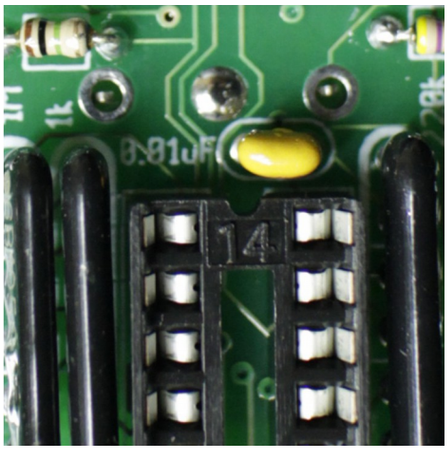
Step 10: Mounting part 2





A. Hold the unit so the panel is facing the floor. Gently guide the LEDs by their leads so their heads fit flush into the holes in the panel (see photo on left). Once positioned, you may want to slightly splay their leads so they stay in place.

Push up on the heads of the 8 buttons to verify they are firmly pressed to the PCB (see photo above right). The buttons have a tendency to slip or fall. If they are not 100% lined up, the cap will rub on the panel. This is very difficult to fix later, so press each button firmly now.



B. Now solder one joint on each component:

- One lead of each LED (either one)
- The center pin of each pot (see photo on left)
- One of the middle-ish pins of each button

C. Flip the unit over and install the final two potentiometer nuts, tighten them down with your 3/8” driver or pliers. Now install the 4 knobs on the pot shafts. Notice how the pot shafts have a D shape, as well as the inside of the knob.

D. Move each knob so that it's centered within the panel artwork(see photo on right). If the center detent of the pot does not match the center point on the panel silkscreen (in this case the triangle shape, see photo), you may have to loosen the pot nut and reposition it. Try holding or pushing the shaft of the pot in the direction you need it tweaked, while tightening down, using your needlenose pliers, with the other hand. If you have trouble pulling the knobs up by hand to loosen the pot nuts, you may have to pry them up with a tool. If this is the case make sure you don't scratch the panel, or damage the knobs. A small screwdriver acting as a lever, with a soft cloth underneath to prevent scratching of the panel, should suffice.

E. Verify that each button can be pressed without rubbing on the panel, that each LED is visible through it's hole in the panel. If necessary, re-position a control by heating up the one pin you soldered in step 10B.

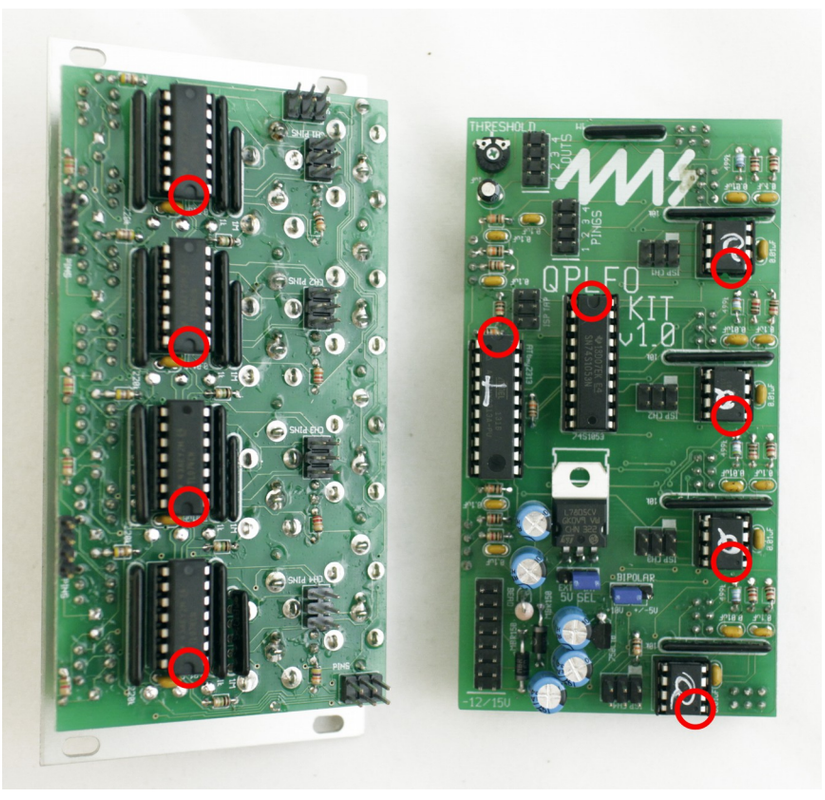
F. When you're sure all the pots, jacks, buttons and LEDs are all placed perfectly, install the rest of the jack nuts, and go around and tighten all of them with a 5/16" socket driver or pliers (careful not to scratch the panel!)

G. Check all the pots, buttons, and LEDs a third and final time, and then flip the unit over and solder the rest of the PCB (all jacks, pots, buttons, and LEDs). Snip the LED leads short.

Step 11: Insert ICs and jumpers

Each IC has an orientation, the dot or notch should be pointed towards the notch in the IC socket. Verify you didn't put the IC socket in backwards by checking that the IC notch/dot lines up with the notch drawn in white on the PCB. See photo (the red circles indicate the notch/dot).

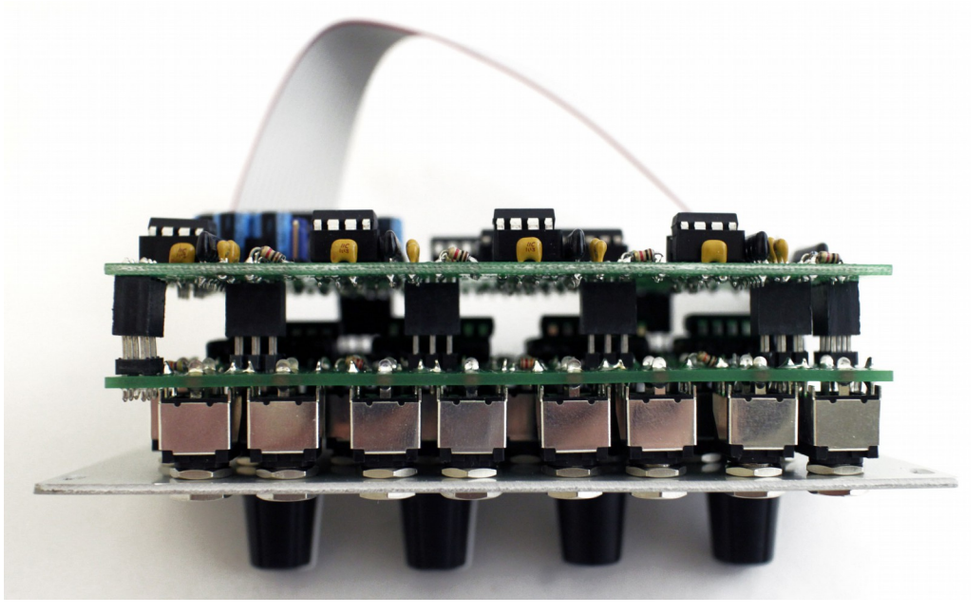
- *TL074 (14-pin) x 4: All on the bottom board (all notches facing down)*
- *ATTNY2313 (28-pin) x 1: Both on the small board (notch facing up)*
- *74S1053 (20-pin) x 1: On the small board (notch facing up)*
- *ATTINY85 (8-pin) x 4: All on the top board. (all notches face down)*



Install one of the blue jumpers on the 3-pin header “5V SEL”, on the right side ("INT"). Install the other jumper on the “BIPOLAR” 3-pin header, on the left side (“+10V”).

Step 12: Finishing touches

Put the two PCBs together: just like you did before in step 7. Push firmly so the pins go all way into the headers and no metal is exposed on the pin. Install the 16-pin ribbon cable with the red stripe at the bottom (-12/15V).



Step 13: Take a break

That's right, walk away and do something else. This is a critical step **especially if you are an advanced kit builder or electronics person** (beginners tend to check their work with more skepticism!). There are many things you can do wrong in building a QPLFO that causes it to smoke and destroy components. So don't rush, have a clear head, and check your work. Come back refreshed. Look over everything:

- Check all the solder joints, it's easy to miss one.
- Verify the ICs and IC sockets are not in backwards.
- Verify the diodes have the band pointing to the line on the PCB
- Verify the 100uF caps are not in backwards (stripe to the right).
- Verify the 1M bussed resistor array has the dot oriented with the square hole on the PCB legend.
- Verify the header pins are not bent.
- Verify no components are sticking up and potentially able to short out to something.
- Verify you installed the two blue jumpers in the right places.

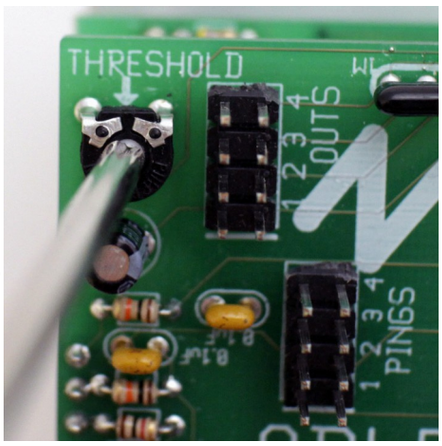
Step 14: Power up and calibration

Power it up! Some lights should come on. If not, unplug immediately and check around for errors, especially near the power connector (missing a blue jumper on the 1x3 header??). If there's a problem, skip ahead to the Troubleshooting section.

Trigger Threshold Trimpot:

There is only one feature that needs to be calibrated on the QPLFO, the trigger threshold:

1. Locate the THRESHOLD trimpot in the upper righthand corner of the PCB
2. Turn back and forth with screwdriver, until you locate a general center point (50%), leave it there.
3. Now try triggering the QPLFO's Ping and Reset inputs with another module, with the channel button set to ON, you should start seeing an output indicated on that channel's LED.



When the trimpot is set to minimum, a very low voltage will trigger the Ping and Reset jacks. When the trimpot is set to maximum, it takes about 5V to trigger the Ping and Reset jacks. The factory setting is at about 50%, or 2.5V. This is compatible with nearly every module. However, if you have special requirements (such as needing the QPLFO to be triggered by 1V triggers from an LZX video system), then you may wish to set to trimpot to somewhere other than 50%. *Note: it's not recommended to set the trimpot to 0% because the triggering module will have to swing a tiny bit negative to indicate the "end" of a trigger. Since very few modules are compatible with this, the minimum recommend setting is about 10%.*

Test Procedure

Start with the first channel (red).

1. **Ping button:** Tap the Ping button twice
    - It should continue to flash at the tempo you tapped.
    - Hold the Ping button down for three seconds: it should stop flashing. (Make sure to tap it twice again to set a tempo for the remaining tests)

2. **“On” button:** Press the On button a few times.
    - The orange light should turn on and off.

3. **OUT jack:** Patch the OUT jack into another module (such as the FM input of a VCO).
    - Listen to the LFO modulate the other module (e.g. listen to the pitch of the VCO go up and down).
    - The modulation should start and stop when you press the “On” button, and the LED next to the OUT jack should flash to the tempo when the “On” button is lit.

4. **Skew knob:**
    - Turn the Skew knob to 0%: you should hear a short trigger modulating the VCO
    - Turn the Skew knob to 10%: you should hear a falling sawtooth wave modulating the VCO
    - Turn the Skew knob to 50%: you should hear a triangle wave modulating the VCO
    - Turn the Skew knob to 100%: you should hear a rising sawtooth wave modulating the VCO

5. **Skew CV:** Patch an LFO (or any CV source) into the Skew CV jack (adjacent to the Skew pot).
    - The CV source should control the Skew. The Skew knob provides offset, so for a positive-only CV source, turn the Skew knob down.

6. **PING jack:** Patch a clock source (QCD, or PEG EOR/F out, or RCD, or SCM) into the Ping Jack. You can actually use anything for a clock, an LFO works great too.
    - The tempo of the LFO output should match the tempo of the clock
    - Unplug the cable from the Ping jack. The LFO should stop (the Ping button should stop flashing, too)

7. **Reset jack:** Plug the cable into the Reset jack. Make sure nothing is plugged into the Ping jack.
    - The LFO should fire once every time a trigger occurs. Try stopping and re-starting the clock to verify this.

8. **Reset One-shot mode:** Slow down the clock (or fire a single manual trigger).
    - The QPLFO channel should do nothing until it receives a trigger on the Reset jack and then it should fire exactly once and stop (this is One-shot mode, or triggered envelope mode).

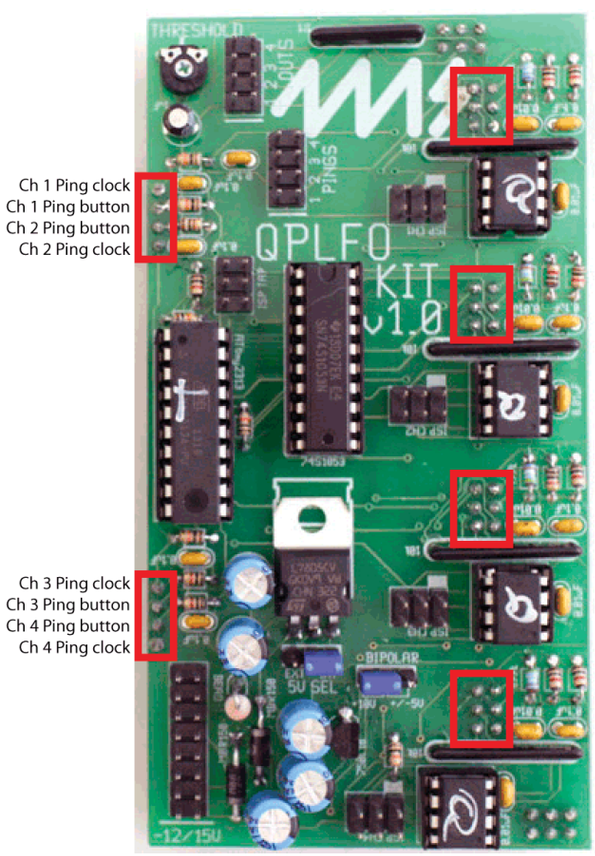
Repeat steps 1-8 for the remaining three channels.

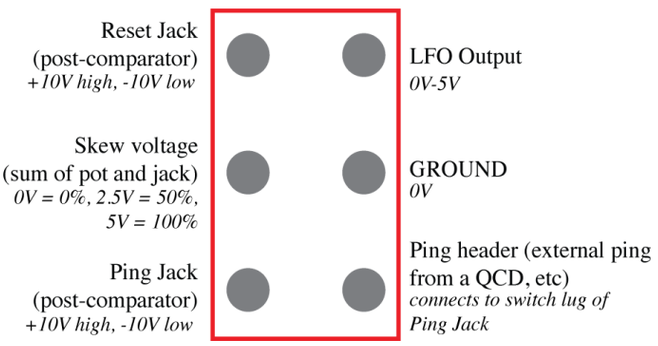
Troubleshooting

If something in the testing section isn't working, continue the entire test procedure and take notes what doesn't work. For instance, if one Ping button doesn't work, are all the Ping buttons not working? The answer tells you if the problem is around the button, or a component that's common to all the buttons.

1. **Verify everything in Step 13 is correct.** ***The two blue jumpers are MANDATORY!!!***
2. **Check for bad joints.** The number one cause of problems is bad solder joints. Even experienced kit builders miss a pin, or short two pins together. Take 5-10 minutes right now to inspect each and every joint. Remove the faceplate if necessary.
3. **Check for component errors**. The number two cause of problems is wrong components in the wrong place, or backwards components. Backwards chip? Backwards button? Check again, it's easy to do! IC socket pins have a tendency to bend under rather than go into the hole, so check all the pins.
4. *Note:* While an easy scapegoat, it's extremely rare for the cause of problem to be a bad component. However, ATTINY chips can get zapped. If one of the channels doesn't work, try swapping its ATTINY85 chip with a working channel.
5. **Measure voltages** on the chips:
    - Each ATTINY85 (four chips): Pin 8: +5V. Pin 4: 0V (ground).
    - SN74S1053: Pins 1 and 20: +5V. Pins 10 and 11: 0V (ground)
    - ATTINY2313: Pins 1 and 20: +5V. Pin 10: 0V (ground)
    - Measure the voltage on the three pins of the 7805. You should read 11.6V, 0V, and 5V.
    - Measure the voltage on the three pins of the 750L10. You should get 11.6V, 0V, and 9.9V.

6. **If you have a problem with the Ping buttons** not doing anything, see diagram below, left.
   - *Ch 1/2/3/4 Ping button pins:* The middle two pins of both of the 1x4 headers should be 5V normally, and drop to 0V while the respective Ping button is depressed. These header pins, from top to bottom, correspond to the channels 1-4 from top to bottom. These points also connect to pins 1, 19, 9, and 11 of the ATTINY2313, respectively for channels 1-4.
   - *Ch 1/2/3/4 Ping clock pins:* If you see the correct 5V/0V behavior on the ATTINY2313 pins in the previous step, then check the remaining pins of the 1x4 headers. You should see a steady square wave going from 0V to 5V, at the tapped tempo. (Hint: if you don't have a scope, tap a tempo that's slow enough that your multimeter can keep up). An identical signal should also be present at pins 3, 18, 8, and 12 of the ATTINY2313, respectively for channels 1-4
7. **If you have a problem with the Ping jack (but not the buttons):** A square-wave version of the signal on the Ping jack should be present on pins 6, 17, 7, and 13 of the ATTINY2313, respectively for channels 1-4. This should match the signal on the "Ping Jack (post-comparator)" pin of the 2x3 header for the channel (see Troubleshooting step 8, and diagram below, right). If you see nothing on the 2x3 header pin, then the problem is likely on the control surface PCB.
8. **If you have a problem with the Reset jack:** You should see a square-wave version of the Reset signal on the 2x3 header (see diagram below, right).
9. **If you have a problem with Skew:** measure the voltage on pin 2 of the channel's ATTINY85. The voltage should be 2.5V with the Skew knob at 50%, 0V at 0%, and 5V at 100%. It also should reflect the CV signal on the Skew CV jack. If not, check the voltage on the 2x3 header pin. If that's not correct, the problem is likely on the control surface PCB.
10. **If the problem is unique to only one of the channels** and seems to involve one of the jacks or knobs, check the 2x3 header for that channel (see diagram below, right). If you don't see the proper signal pins, that can help you identify which PCB the problem is on. Also, if the channel is totally dead, try swapping the ATTINY85 chip with a working channel.





Above: Each channel has a 2x3 header.

Right: The two 1x4 headers connect the Ping buttons and Ping button LEDs to the ATTINY2313 chip. Also, the four 2x3 channel headers are shown.

Don't give up!!