NEUTRON SOUND

ORGONE ACCUMULATOR

DIY ASSEMBLY
MANUAL

version

0.23

BETA DOCUMENT

Contents
page numbers to be added on final document.

Introduction
Programming the teensy

Assembly

IC Sockets

Diodes

Resistors

Ceramic/Film Capacitors

Power header

SW8

Subsection A: Teensy

Cut the trace
Mounting the pins on the teensy
Mounting the Board Headers

Assembly - continued

Trimmers & voltage reference

Electrolytic capacitors

Rest of Switches

Potentiometers

Subsection B: Jacks

kobiconn & thonkiconn
“earthenvar”

ICs and Testing

Pre Testing

Install voltage driver/buffer IC

Test + and – 3v

Install opamps

Button caps and Knobs

Wiring your own panel

Controls
Power/Ground.

Introduction:

Thank you for your building a neutron-sound.com Orgone Accumulator digital oscillator.

This manual will show you how to build Orgone Accumulator in Eurorack or
how to wire it up for other formats.

Some of this build can be quite fiddly, and you do not want to be worrying about how you are soldering, If you have never build a board or kit I would suggest soldering practice, or building a more simple kit such as a mult, to gain experience first.

Please email me with any questions you may need answered. I can
be reached directly via email at NOA@neutron-sound.com

I hope the Orgone Accumulator becomes a worthwhile addition to your modular system.

Programming the Teensy.

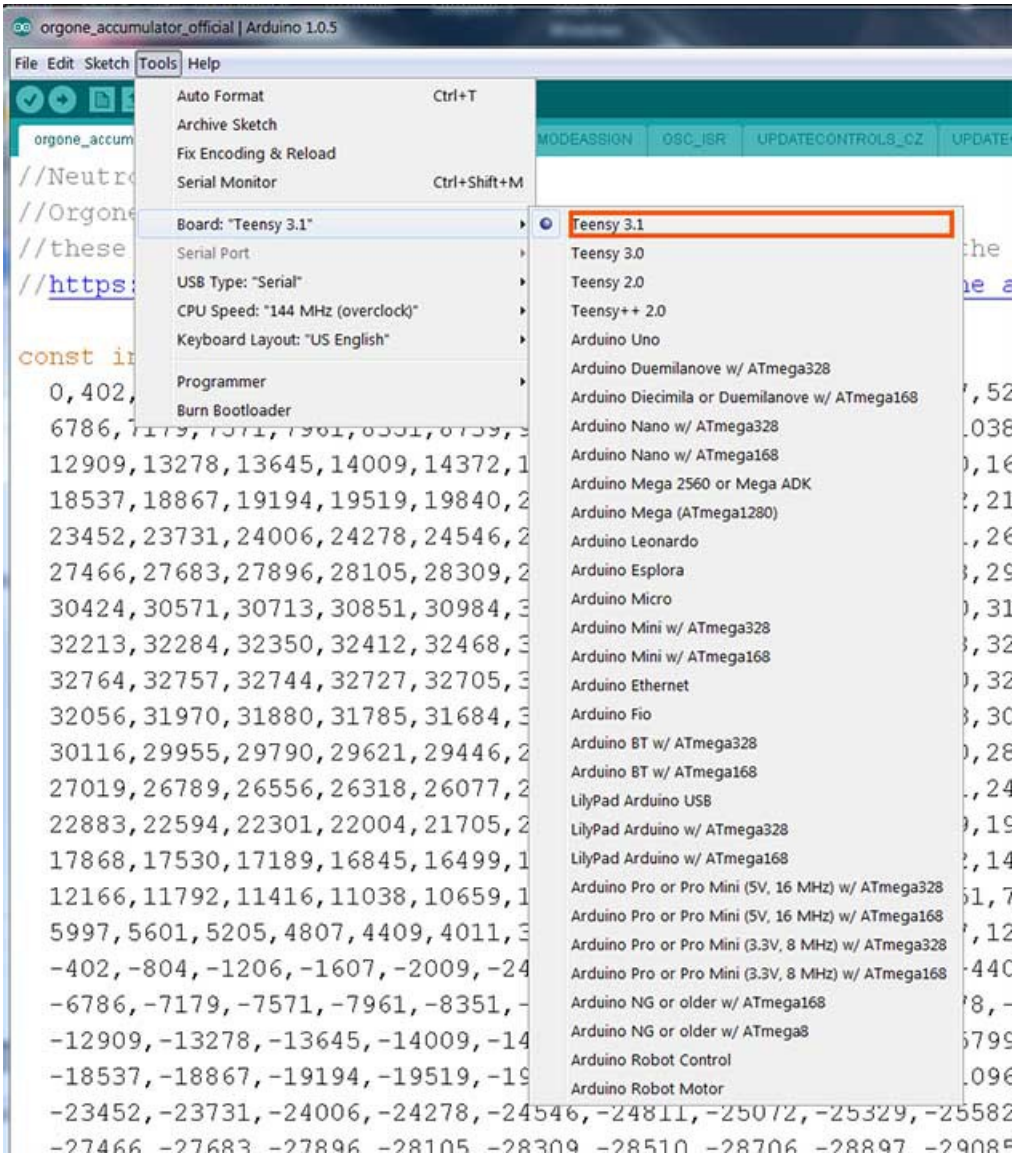
Before you start assembly, it is a good idea to program the Teensy to become familiar with the Arduino software and the Teensy loader.

First go to this site and follow the instructions to install Arduino 1.0.6 IDE and the Teensyduino software.
(previous to orgone firmware 1.1.0 and teensy 1.2 the firmware worked with arduino 1.0.5)

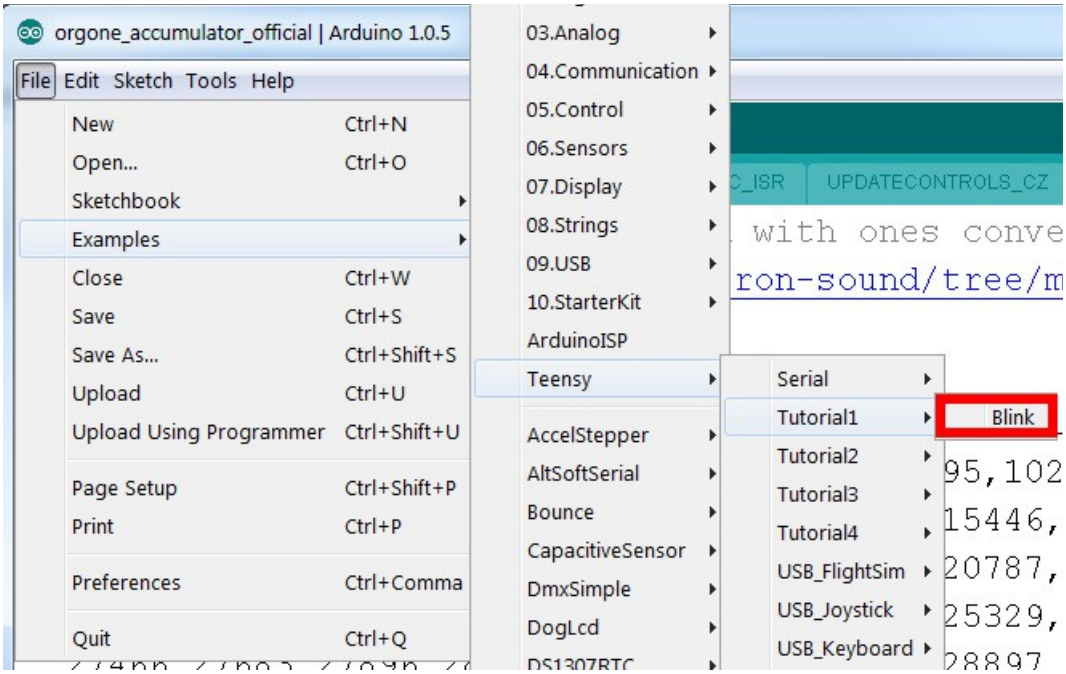
https://www.pjrc.com/teensy/td_download.html

Connect your teensy to the computer via USB A to Micro-B cable

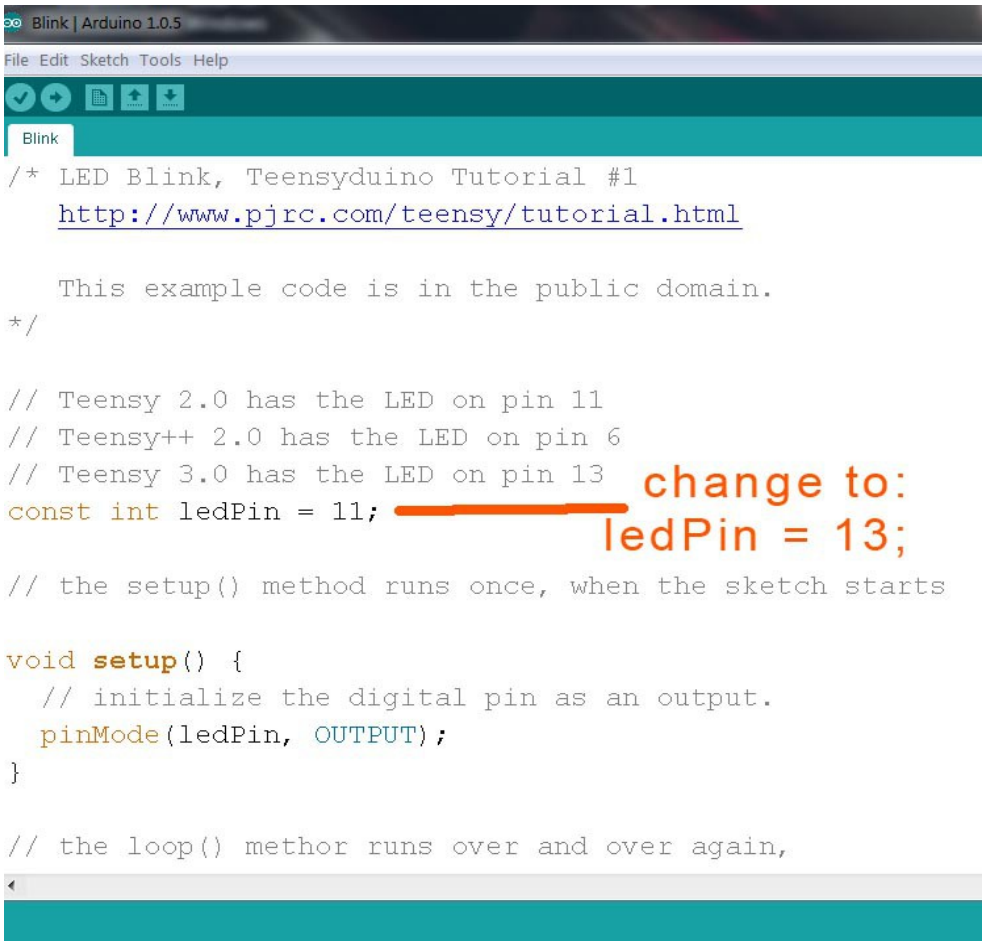
Open the arduino 1.0.6 IDE program and set the board to Teensy 3.1



open the “blink” sketch in the teensy examples



You will need to do a small edit for the blink program to work

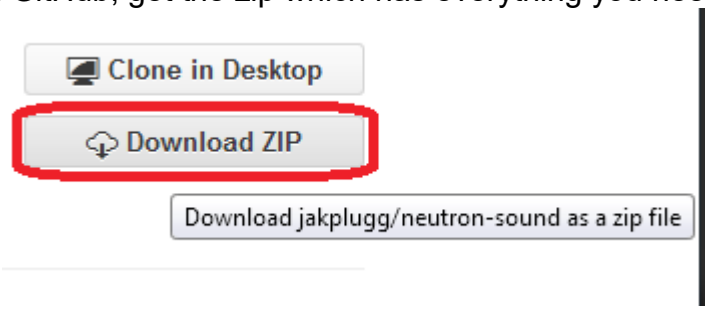


Now press the button that looks like a checkmark, you will see a bunch of text on the bottom of the window while the code is compiled. Then, if the teensy is new to your computer,The software will ask you to press the button on the Teensy, you should only have to do this the first time. Press it, then it should load. The LED on the teensy should blink. Your Teensy works!

I have found sometimes you have to press the reset button more than once, but it could be because my computer has had several different Teensies connected.

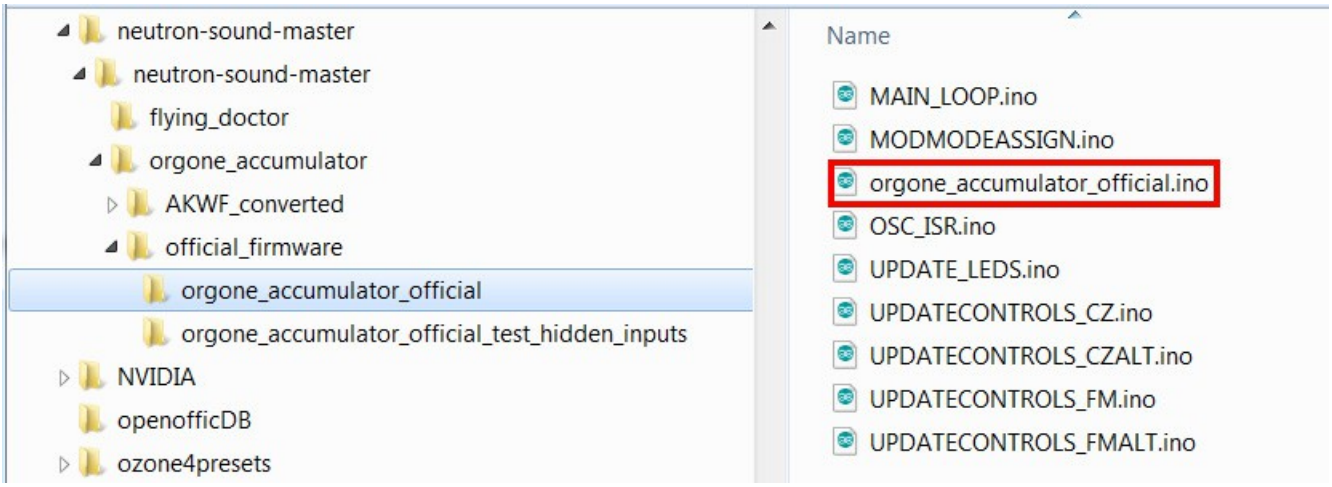
Next, go to
https://github.com/jakplugg/neutron-sound

and download the zip file. (bottom right corner of GitHub page). There are multiple files so don't try to download individual ones from GitHub; get the zip which has everything you need.

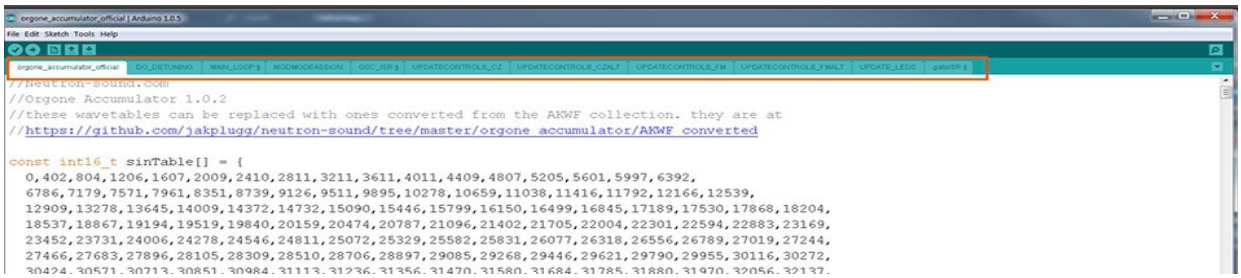


The zip contains the main firmware, as well as converted adventurekid waves for advanced users who wish to change the firmware waveforms, and a test program for troubleshooting the SMD pins on the headers.

Unzip the file you downloaded from GitHub, and remember the location where you put it.
select file>open in the Arduino IDE and navigate to this file you just unpacked.



You should see several tabs in the Arduino IDE if the file was opened properly.



Now you can go ahead and click the verify button to compile the code, then the upload button to put the firmware on your Teensy.

Overclocking option: see appendix B

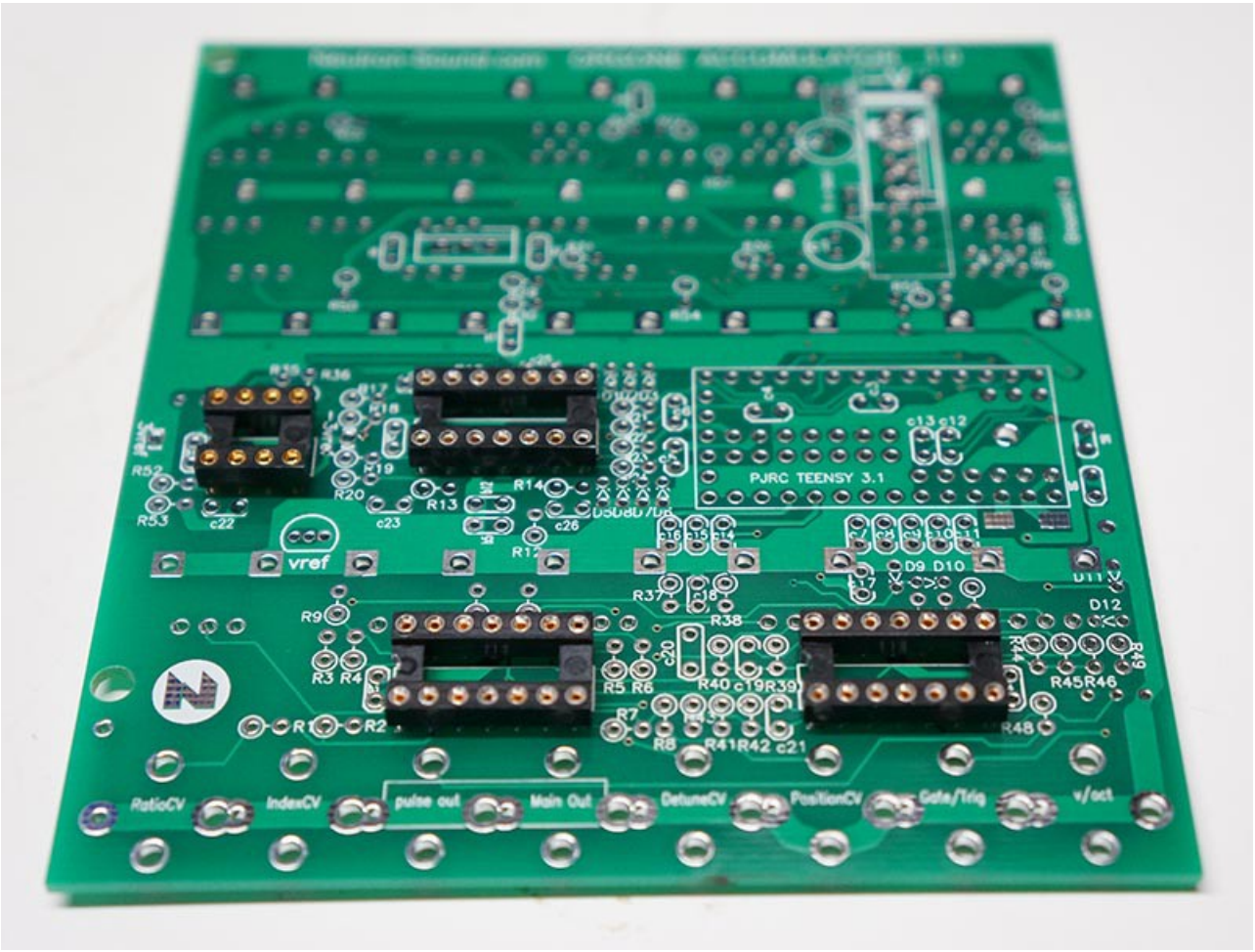
If all is good, you can put the Teensy aside in a safe place for now.

ASSEMBLY

IC Sockets:

Install the IC sockets, being careful to orient them correctly. There is a semicircular notch at one end, which must line up with the notch on the white outline on the PCB. You can put another PCB, book or something flat on them so you can turn the board over for soldering without them falling out.

Picture of board with IC sockets

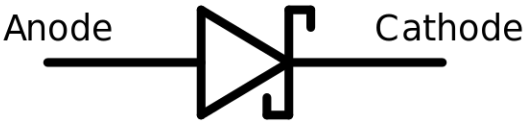


DIODES:

There are 12 BAT42 diodes. They are the only type of diode in the circuit.

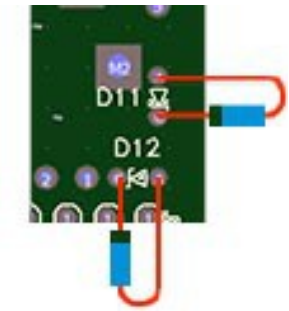
The orientation mark on the PCB is a V, it looks like this (but quite small)

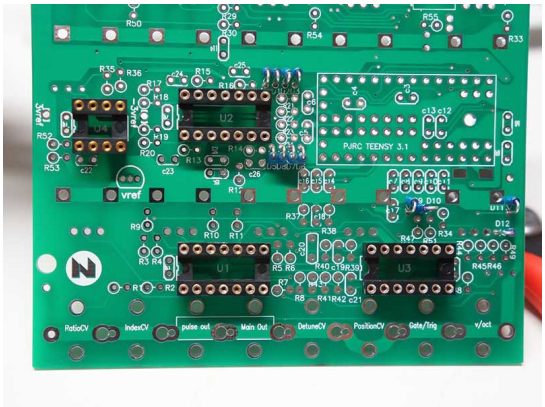
The diodes have a stripe at one end which corresponds to the cathode.



Be careful to get the diodes the right way around – check and double check!. They are difficult to desolder once you have soldered them, even with a heated desoldering gun.

Here is a representation of part of the board with diode leads shown in red (not to scale)





RESISTORS:

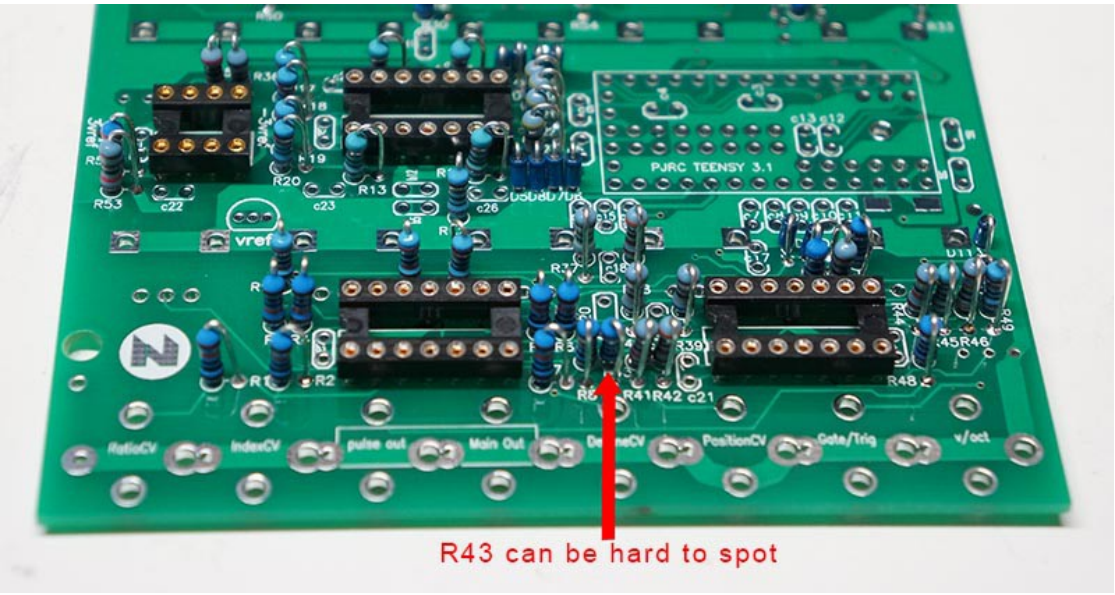
The resistors are all mounted standing up. This was done to conserve space and avoid the cost of an additional board. Resistors are unpolarized, it doesn't matter which way around they go.

If you bought the resistors from the BOM, they will be packaged individually and have the value marked on the packaging.

Check the value against the resistor color code, or use a multimeter if you are unsure.

quantity	Designation on board	Value
8	R1-8	100K 1%
10	R9-12, R17-20, R49, R50	20k 1%
5	R13-16, 45	10k 1%
4	R21-24	47R
16	R25-34 ,43, 48, R54-57	100R
1	R35	2K
2	R36,53	2.2K
2	R37, R51	3.9K
2	R38, R42	3K
1	R39	6.8k
1	R40	1.8K
1	R41	1.2K
1	R44	18K
1	R46	30K
2	R47, R52	22R
2	un-numbered resistors near electrolytic capacitors	10R or bead

When installing the ferrite bead or resistor in the unmarked positions, leaving the longer leg facing down on the board will give you power test points.

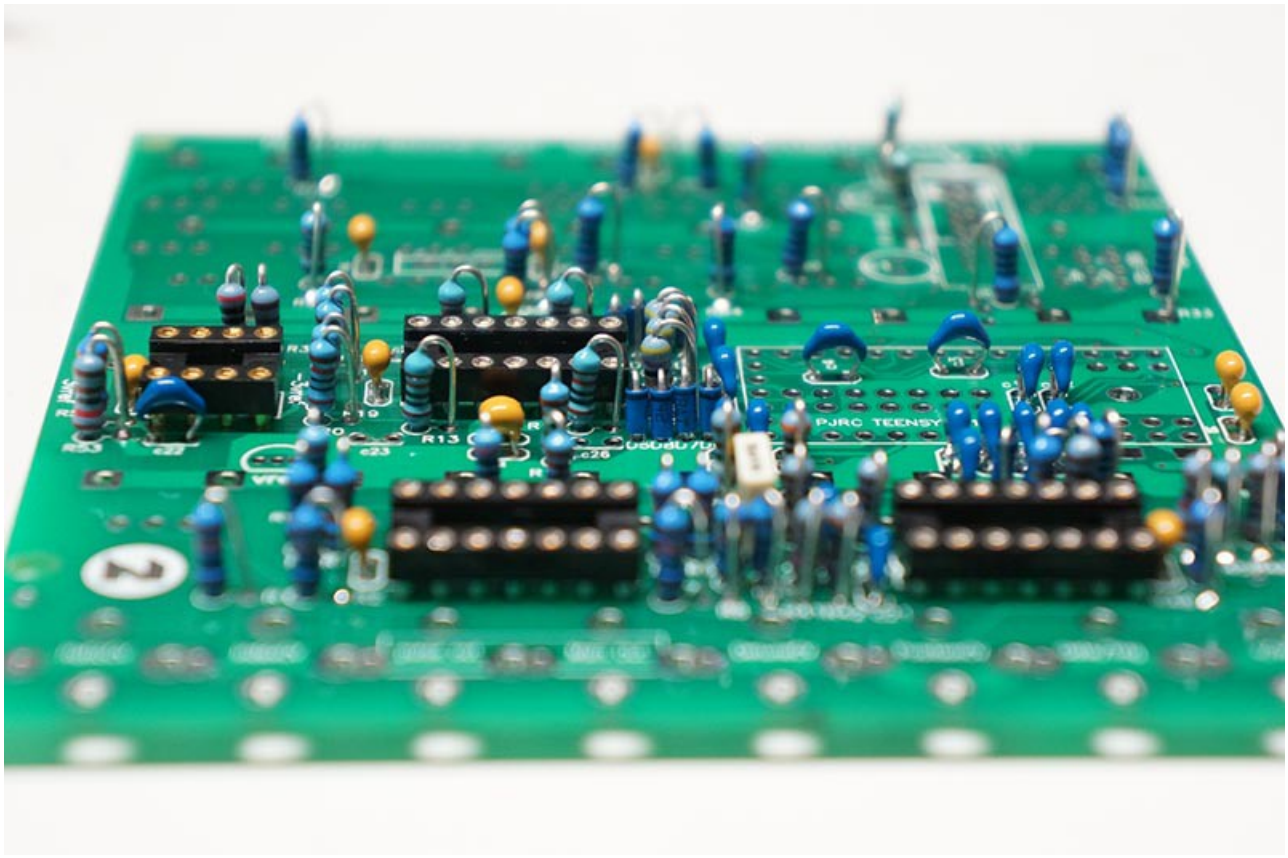


CERAMIC AND FILM CAPACITORS:

The ceramic and film capacitors are next. These capacitors are unpolarized, it doesn't matter which way round they go. The ones marked “B” are bypass capacitors and all $0.1\mu F$/100nF

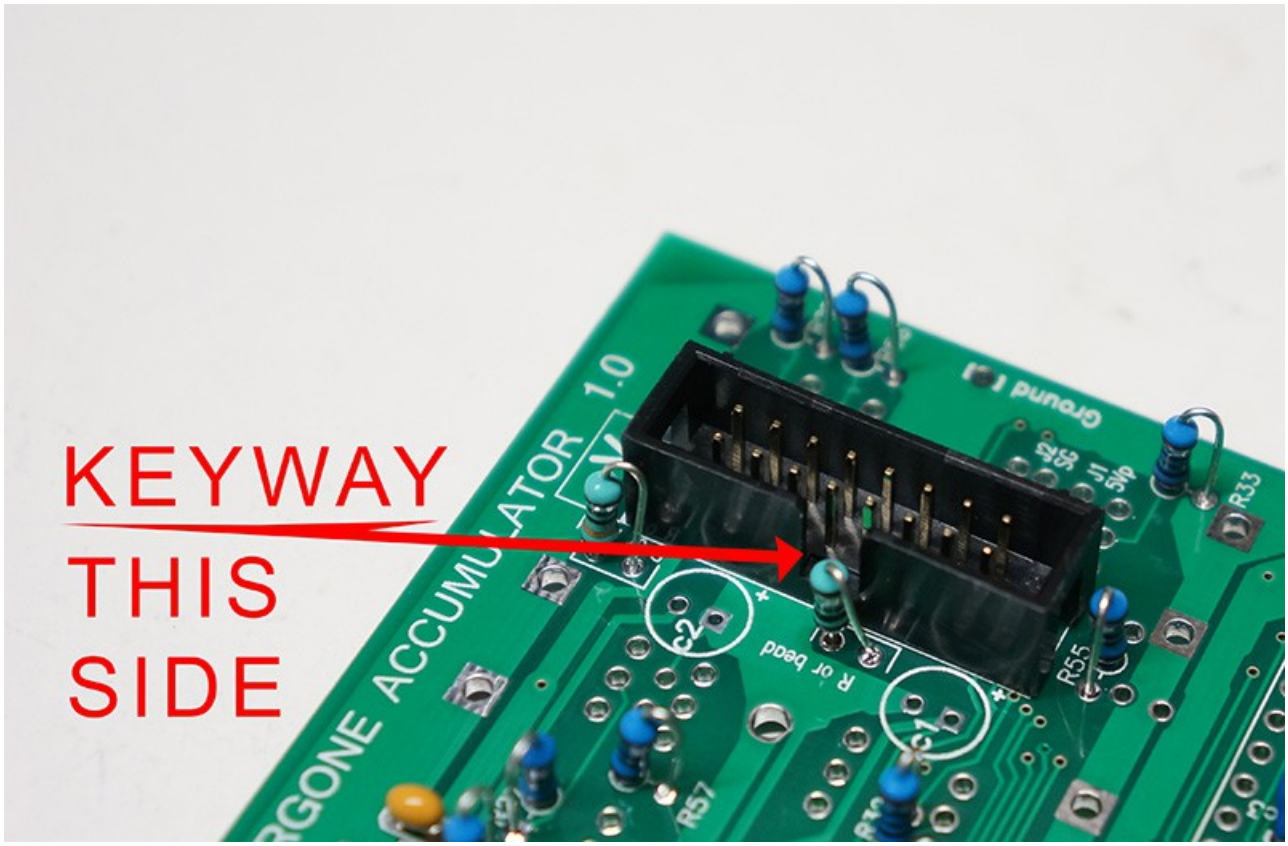
quantity	designation	value
20	**C3-17,C19,C21,22**	**1nF**
12	**B1-B12**	**100nF (0.1 µF) bypass**
4	**C23-26**	**47nF (any 0.1”/2.54mm lead spacing capacitor)**
1	**C18**	**3.3nF**
1	**C20**	**15nF film**

Photo of the C0G capacitors on board



POWER HEADER:

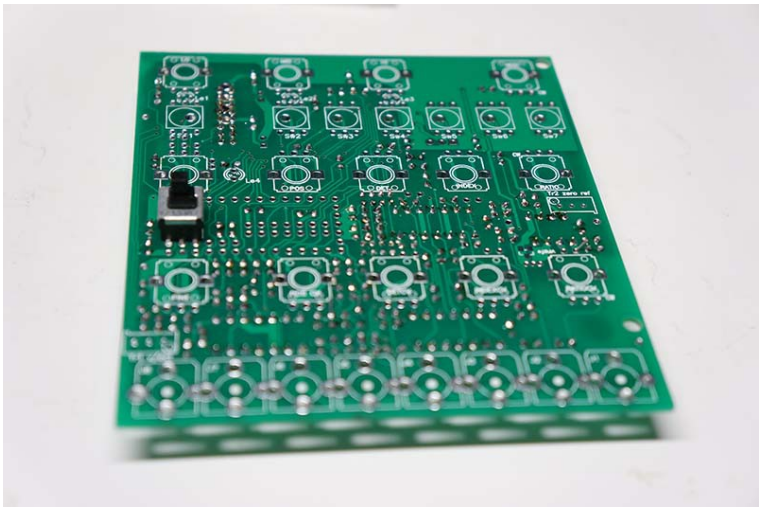
Next is the power header, it instals similar to the IC sockets, **IMPORTANT!** make sure you put the keyway in the header so it is facing the 2 round board markings for the electrolytic capacitors. This ensures that you plug the power cable the right way.

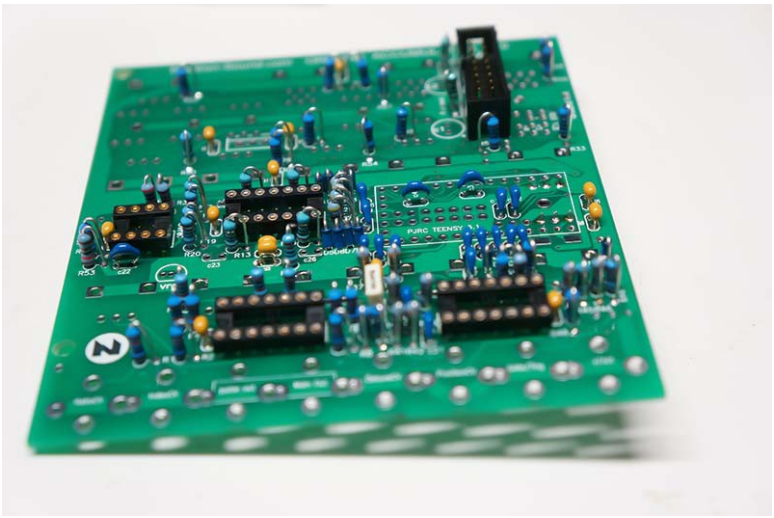


SW8:

SW8 is the latching button which mounts on the opposite side of the board to the Teensy. It has to be mounted now, otherwise it becomes very difficult to solder after the headers are soldered. What this means is that now you will no longer be able to place the board flat on the table for soldering. If you have a Panavise or other board vise, it is not a problem, but if you do not then you will need a little manual dexterity! **NOTE! If you have 1 momentary switch (synthcube kit) IT DOES NOT GO HERE**

Note the registration hole in the board for the little plastic bump on the bottom of the switch. Solder one pin of SW8 then look how flush the switch is to the board. If it is not, then re-melt the solder while pressing down on the switch. Once it is flush, then solder the other pins.





If you don't have a vise, it might help to temporarily put a couple of standoffs in the mounting holes, if you have some spare.

DO NOT PUT THE BUTTON CAP ON YET!
If you need to reflow any solder joints later, the button caps are easily melted by your soldering iron.

SUBSECTION A: Teensy Hardware.

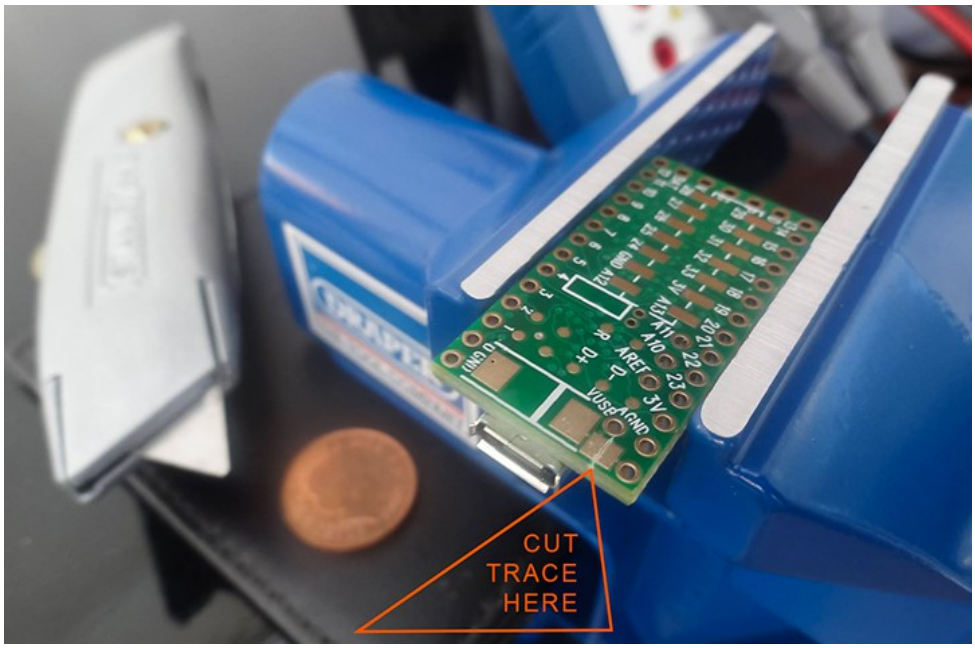
Cut the trace:

You should have programmed the Teensy by now with at least the blink program to make sure you know how to operate the software, and that the Teensy is working. Now you need to separate the 5V VIN from the power supplied by USB. Make the cut between the two rectangular pads. If you have a multimeter, check that the two pads no longer make an electrical connection.

Cutting this trace still allows the Teensy to be programmed via USB while it is connected to the synthesizer power, however, the Teensy will no longer work on USB power alone.

If the trace is not cut, your synthesizers 5v and the computers USB power will be connected together, which could break something.

*advanced option in appendix 1

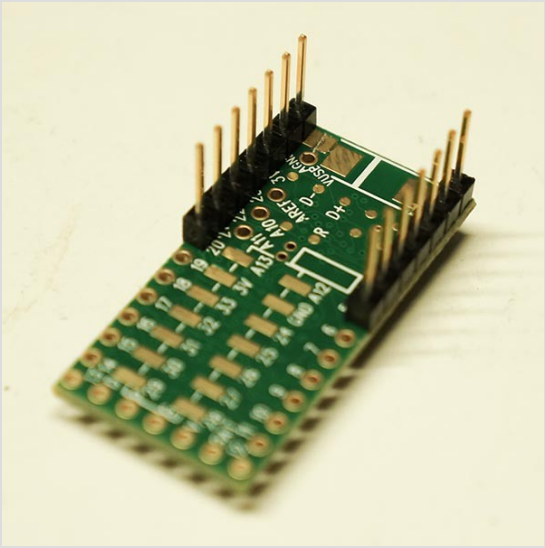


Mounting the pins on the teensy:

The Teensy needs to have pins along both long sides, along the short side, a group of three near one edge, and a special set of 14 pins in the middle which use a SMD connector. Lining these all up correctly can be tricky, so proceed as follows.

Warning! be careful not to get any solder on any of the pins long ends, even the tiniest blob of solder makes it impossible to insert the pins in to the headers, and it is hard to clean off without a vacuum desoldering tool.

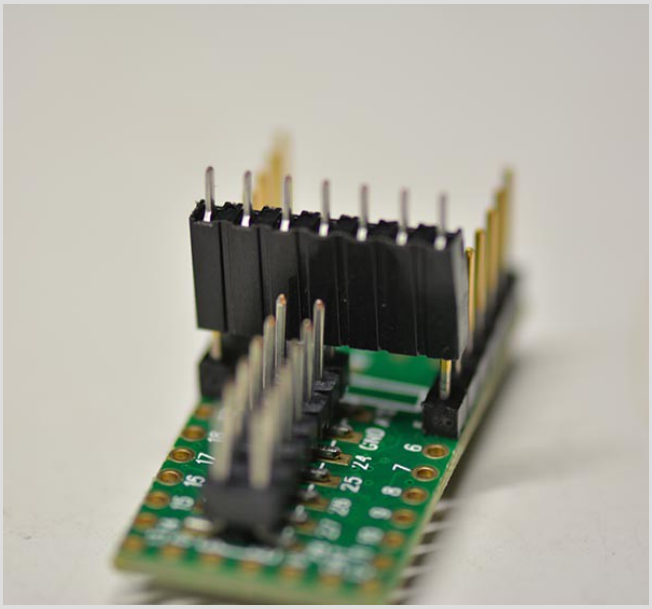
Instead of the 14 rows you might expect for the two long sides, break off 2 rows of seven from the breakaway pin strip. These will cover half of each long side, nearest the USB connector. This makes getting at the SMD pins easier, while also giving you an alignment point.



Get the 7 pin female board header, we are going to use that as an alignment tool, we will call it “alignment tool” for now.

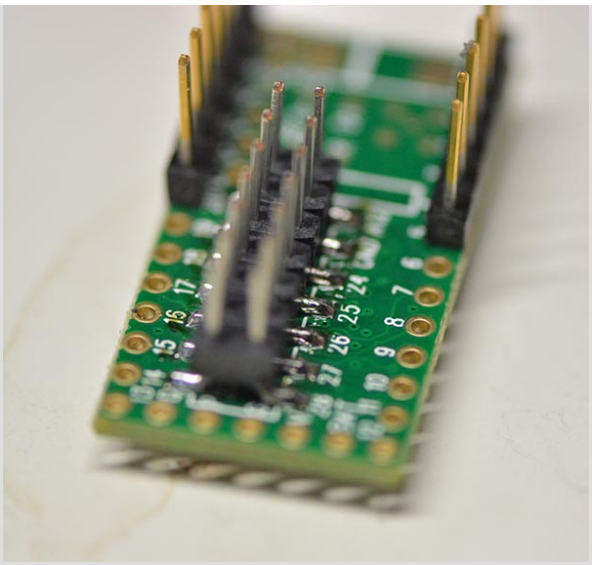
Solder in 2 of the 7 pin rows, at the USB end of the Teensy. Just solder one pin for now. Make sure they are vertical and flush to the teensy (you can use “alignment tool”)

Now place the alignment tool over the rows you soldered, and also the SMD header,



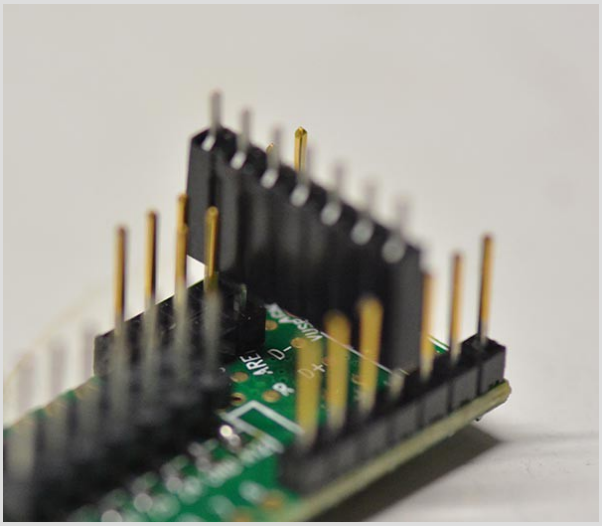
Solder a couple of the SMD pins, and then remove the “alignment tool”

Now you can solder the rest of the SMD pins fairly easily.



Now break off 3 pins from the breakaway header and use the alignment tool to hold them in the 3 holes marked AREF A10, and A11 on the Teensy.

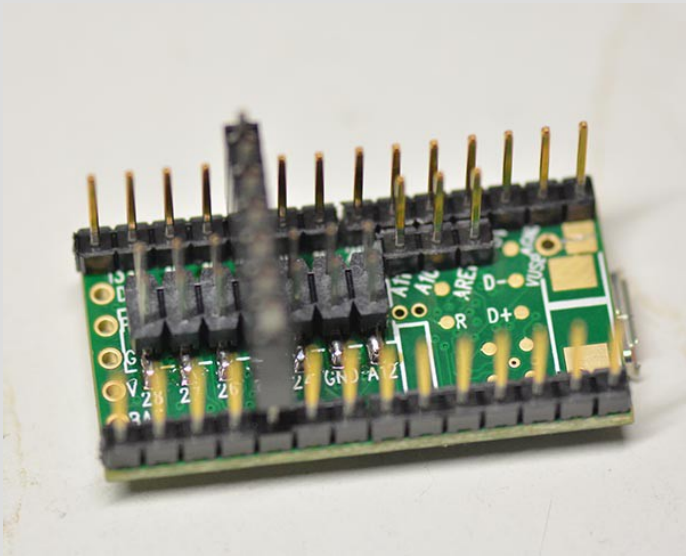
And solder one pin, to hold them in place.



Now you can break off 2 more strips of 7 pins and place them along the remaining side holes. The alignment tool can be used again

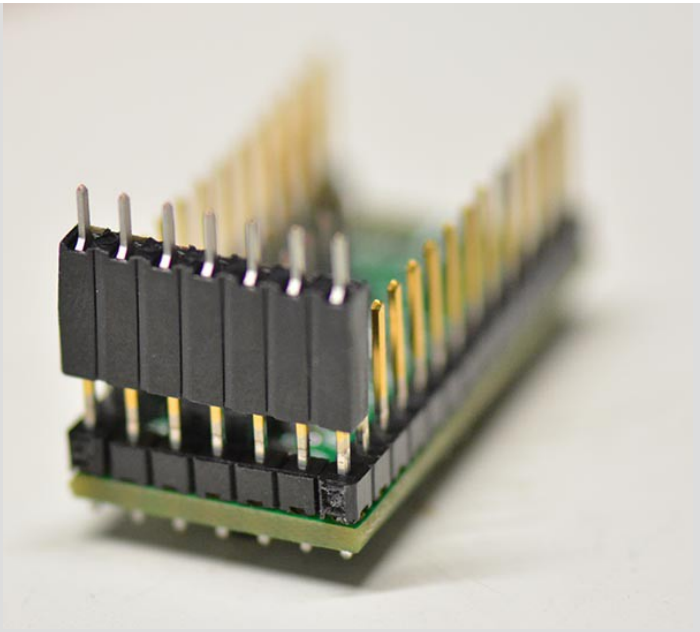
You may need to file or sand the ends of the strips you cut, if they interfere with the strips already in place.

Just solder 1 pin on each for now.



Now utilize the alignment tool yet again for the remaining 5 pins at the short end of the Teensy. You may now retire the alignment tool and call it a board header again.

Again solder only 1 pin.



Mounting the Board Headers

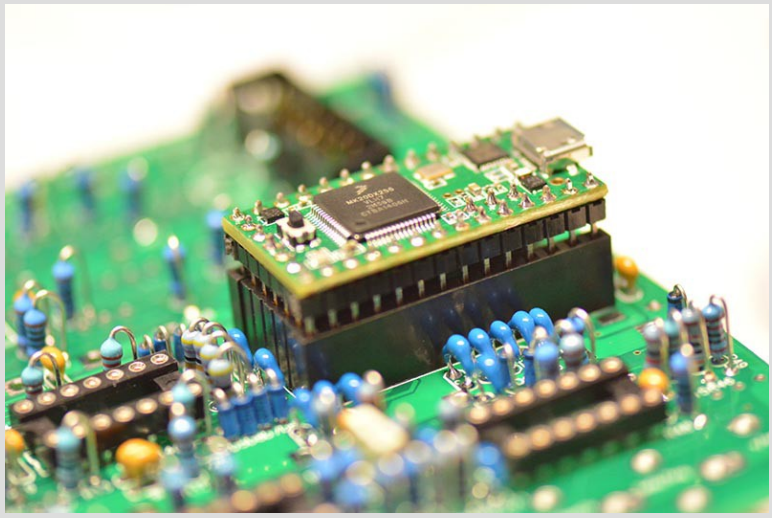
You can put all the headers on their respective pins, to align them all. You will not be able to push them all the way down though; this is fine.

Press the 7 pin one on the middle row of the SMD pins first and use that as a guide for how far to press the others.

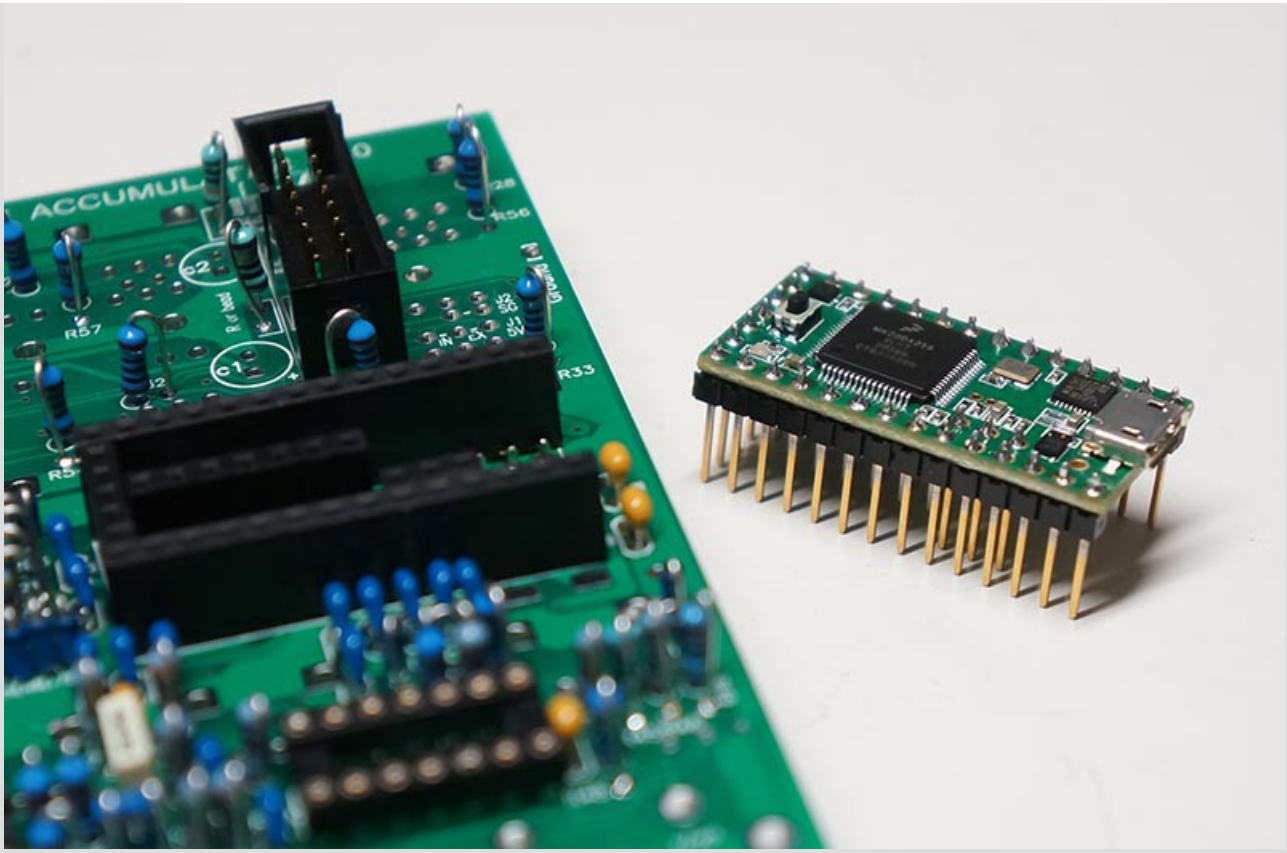
Note that only the “middle” row is used on the SMD header. It is the row that goes down the middle of the Teensy. The other row is just to add stability to the SMD contacts if you have to remove the Teensy.

It can be a bit tricky to get all the header pins in to the Orgone Accumulator board, and you may need to move the pins/header a bit to make if work nicely. This is why you only soldered one pin on each (Idea thanks to hexinverter).

Solder all the board headers to the PCB, then all the Teensy pins to the Teensy. Be careful not to get any solder on any Teensy components.



At this point, If you are not using a Panavise or other PCB clamp, remove the Teensy. You should be able to pry it out quite easily by going around it bit by bit with a screwdriver. Don't rush it and bend pins.



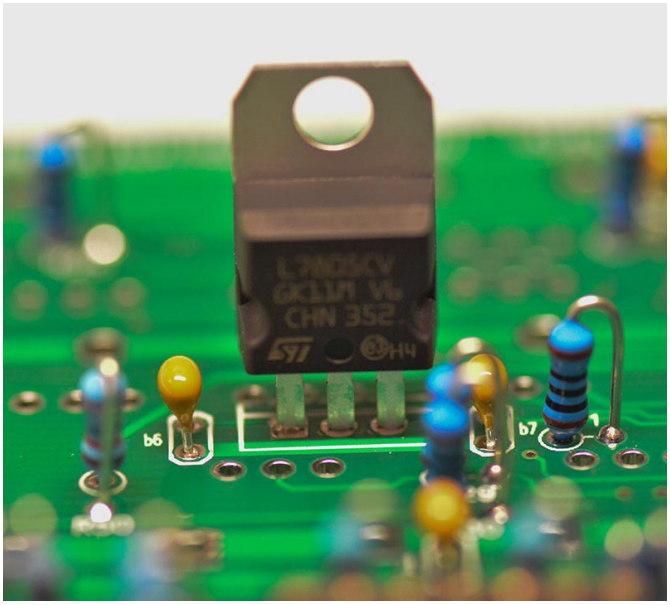
That's it for the Teensy for the time being.

Regulator, Trimmers, jumpers & voltage reference

There are 2 power options: you can use your systems 5 volt power if it is equipped, or you can use an on board voltage regulator to generate 5V from the 12V rail.

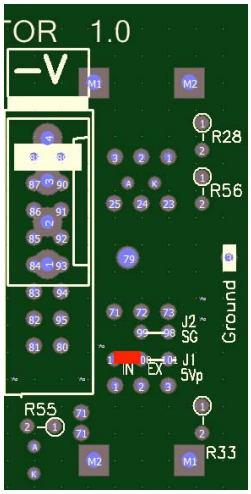
Regulator

If you wish to use the regulator, then you can solder it now. Make sure to put it in the right way around. The marking for it shows a line on the PCB, on the side where the metal tab goes.

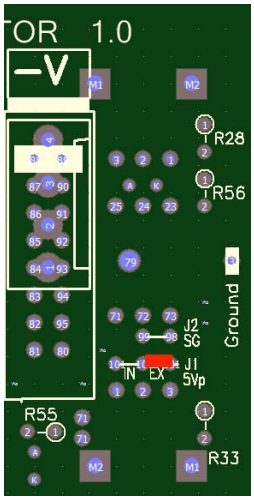


Jumpers

If you are using the regulator, Then solder a jumper across J1 where it says “IN”

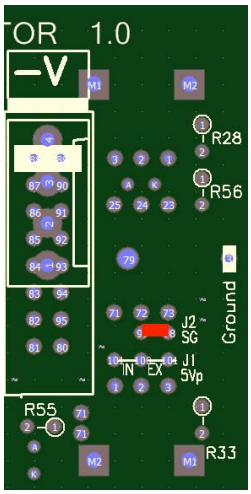


If you are using system 5v power, instead solder a jumper across J1 where it says “EX”



There is a jumper called “SG” which stands for system gate. If you have a modular with internal gate and CV signal this will attach the system gate to the Orgone Accumulator. The oscillators will always reset when a gate is received.

I prefer to leave it off, because with and without resetting are quite different in some cases. (see video 2) so it is better to patch a gate signal via the front panel when you want it.

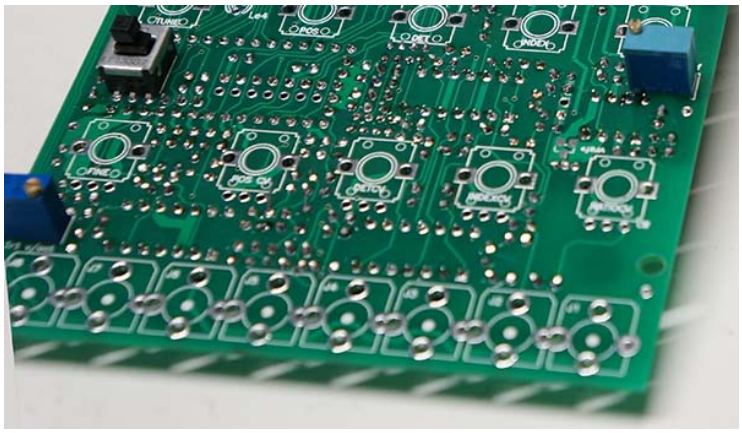


Trimmers

The trimmers go on the other side of the board where the pots are. (you can put them on the back if you

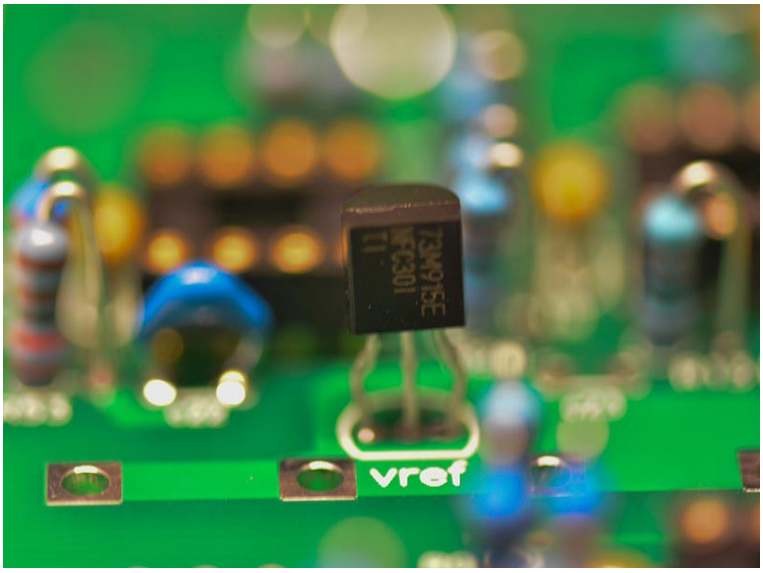
prefer, but then you will not be able to adjust then via holes in the front panel). If front mounting them, just solder the pin farthest away from the adjuster screw for now. This makes it easier to line up the adjustment screw with the hole in the front panel. Try and get their bases as flush as possible to the PCB.

The 100k trimmer mounts above the 1v/oct input, and the 500R one mounts below the freq pot.



Voltage reference.

The LM4040-3 3v reference goes in as shown on the board. This may become obsolete in this format soon, so there is provision on the back of the board for an SOT23 version.



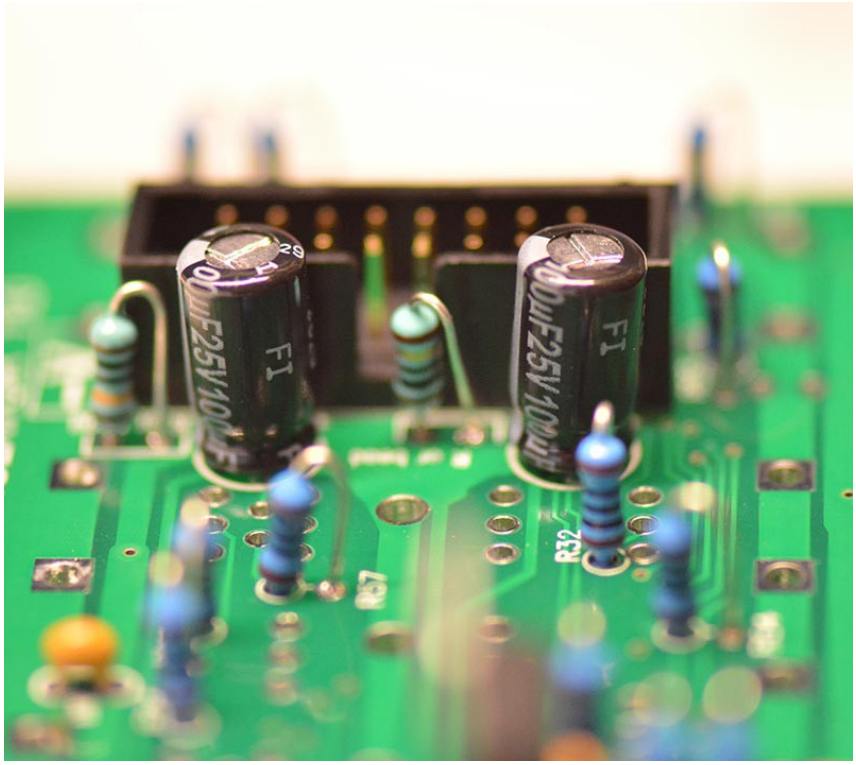
You will have to bend the legs to make them fit through the holes, then bend them away from each other on the reverse side of the board before you solder them. Be careful soldering, because of the close spacing, and do not let it overheat.

ELECTROLYTIC CAPACITORS:

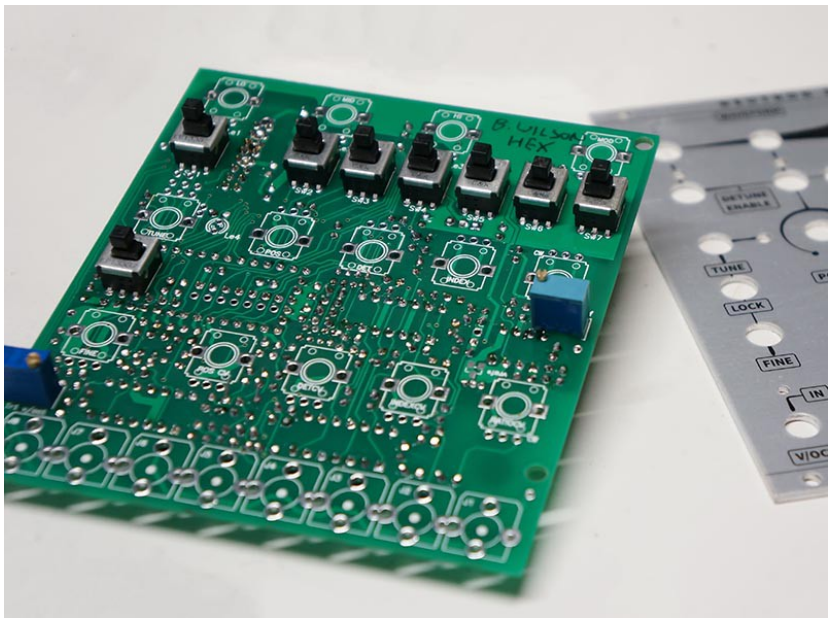
There are 2 of them, electrolytics are polarized and can explode if powered the wrong way so they must be installed the correct way around. You can identify the stripe on the capacitor with the – sign on it and insert the negative terminal in the hole with OUT the + sign (the negative hole has a round pad)

Alternatively you can look for the longer lead on the capacitors and insert this positive terminal in the positive hole (the positive hole has a square pad).

Check that they look like this, before soldering:



The rest of the switches (sw1-sw7)



The base of the switches needs to be flush with the board, solder one pin, and then you can inspect the switch and, if necessary get it flush by pushing the switch against the board while re-flowing the one pin you soldered.

Once you are happy with them, solder the rest of the pins.

NOTE: If you have one momentary switch (such as with synthcube kits) it is SW3! The effect selector.

Potentiometers & LEDs

The pots have to be lined up with the holes in the front panel, as well as the trimmer screw heads.

Take off the nuts on the pots (if they came with them threaded on).
The small tabs on the pots do not have to be removed.
Place the 4 corner pots in their holes, without soldering. Remember the “FREQCV” pot is B100k, and the other three are B10k

Put the panel over the pots and the brass screw heads of the trimmers, so they line up.

Thread the nut on the pots on to hold everything in place.

Now carefully turn the entire assembly over while holding the board and panel so that no pots come loose. Solder one pin on each of the pots.

Inspect to make sure the metal standoffs on the pots are touching the board, if they are not then re-flow and press where necessary. Once all the pots are lined up nicely, go ahead and solder them, and solder the remaining pins on the trimmers.

Note: *you may have to increase your soldering iron heat for the potentiometer mounting tabs, as they are relatively large, as well as being attached to a large ground trace.*

Now do the rest of the pots the same way. Making sure to put the B100k ones in the “POSCV, DETCV, and INDEXCV”.

LEDs

You can put the LEDs in now, observing the polarity. Some LEDs have a flat side, which is marked on the board, and LEDs have a longer leg. This is the positive lead.

Remove the panel, and place the LEDs loosely in their positions, then you can put the panel back, holding in place with a couple of potentiometer nuts.

use the LEDs leads to position them so they sit in their holes up to the flange, then you can solder them in place.

Subsection B: Jacks
“Erthenvar”
(no photos yet)

These jacks are higher than the pots, so there are a few options to deal with it.

- If you have a drill press, an easy method is to re-drill the jack holes in the panel to 5/16th inch or 8mm. This reduces the height from the PCB to the panel, but you will need washers under the jack nuts.

- If you are lazy, you can just put nuts on the jacks, and the top row of pots. This may be less durable in the long run.

- You can put no washer on the top row, one washer on the middle row, and 2 washers on the bottom row of pots which gives a slight slant to the PCB..

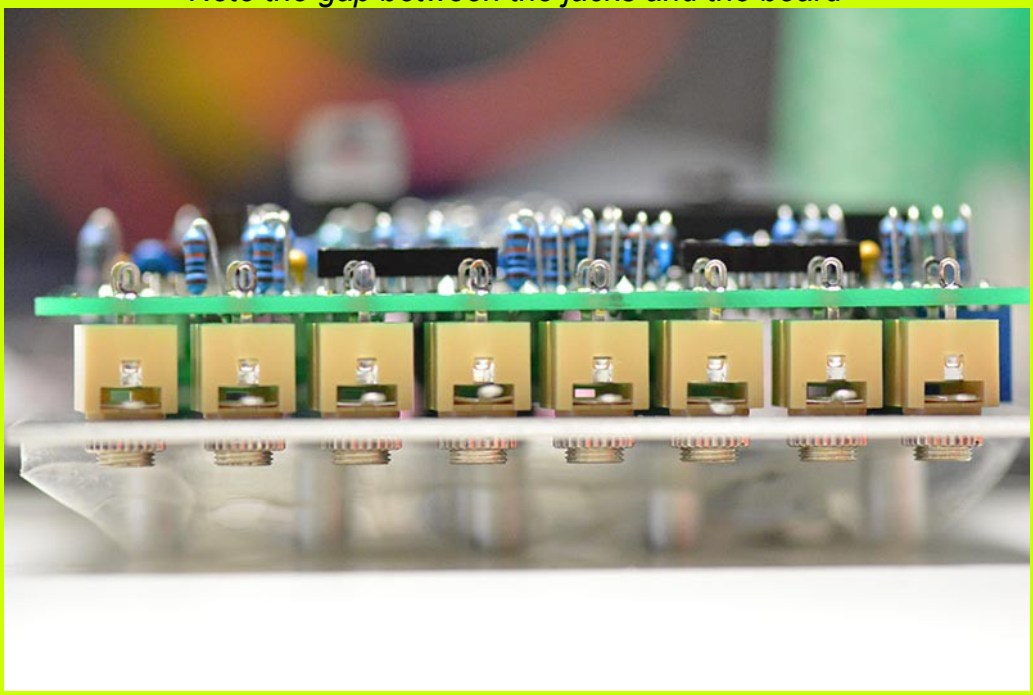
- You can cut the jack standoff tabs by 1mm, this worked for me, but it broke a pair of snips, and I am not sure if the jacks will stand the test of time, because those tabs also hold the plastic in to the metal.

Kobiconn & Thonkiconn

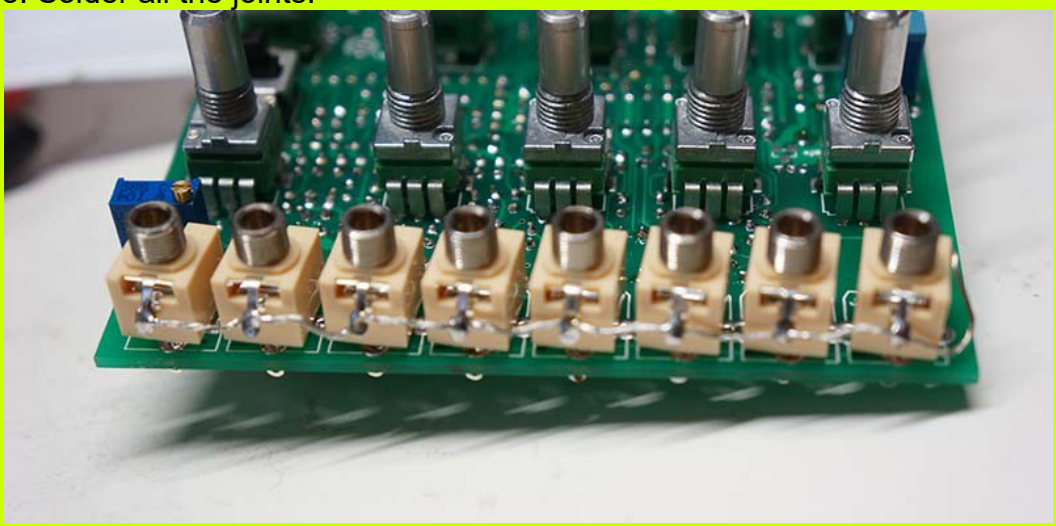
These jacks are a little lower than the pots. To make up the difference, they are “floated” so they are flush to the panel rather then the board.

To float them, place them in their holes with the ground leads facing down, then put the panel on, tightening a couple of pot nuts. Then tighten all the Kobiconn nuts which will pull them to the correct height then solder them. They will turn a bit, but they will be lined up to the panel which is whats important.

Note the gap between the jacks and the board



Then you have to wire the Kobiconn flying ground pins. Thread a wire through all of them and then bend them down so they will not interfere with your modular rails. Then put one end of the wire in the unused plated hole next to the “N” logo. Solder all the joints.



Pre IC Testing

There are a few tests which can be performed before inserting the ICs. Just in case there is a short or open circuit in the power rails, which is the most likely thing which will destroy op-amps

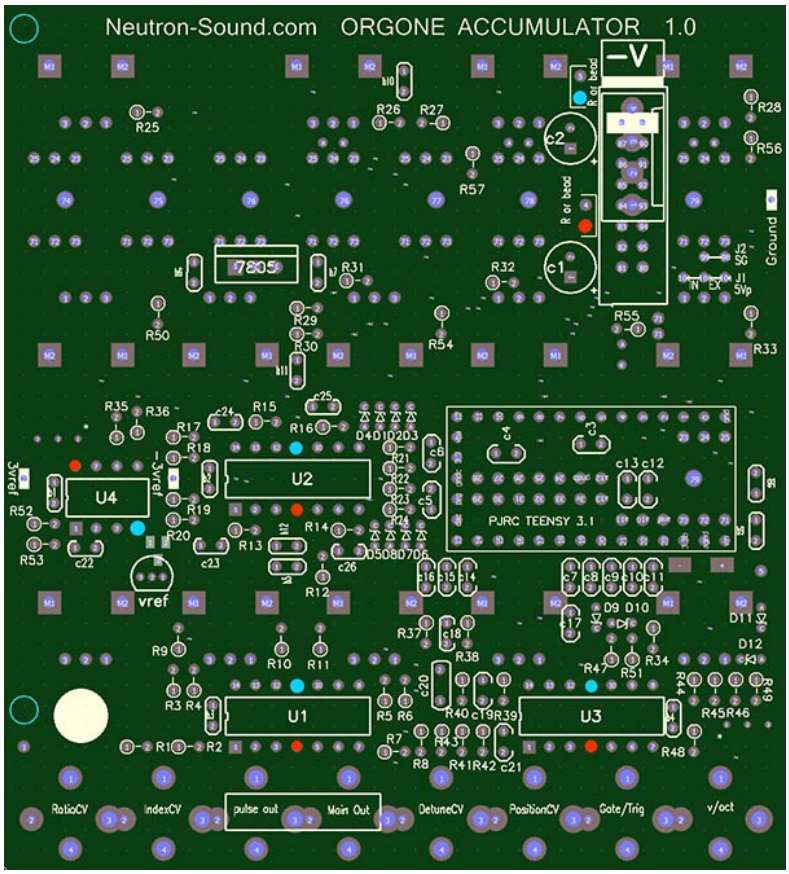
Test the power is not shorted, you can check for continuity between each of the 10 ohm resistor legs and a ground pad (there is one marked ground on the right side near the power header)

you might get a breif beep from your continuity tester as capacitors charge.

Test that the IC sockets are all connected to the power rails correctly.

The pins marked red in the image are positive +12v and the light blue are negative -12v

You can use the leads on the 10 ohm or beads near the power header, or power it up and measure the voltages.



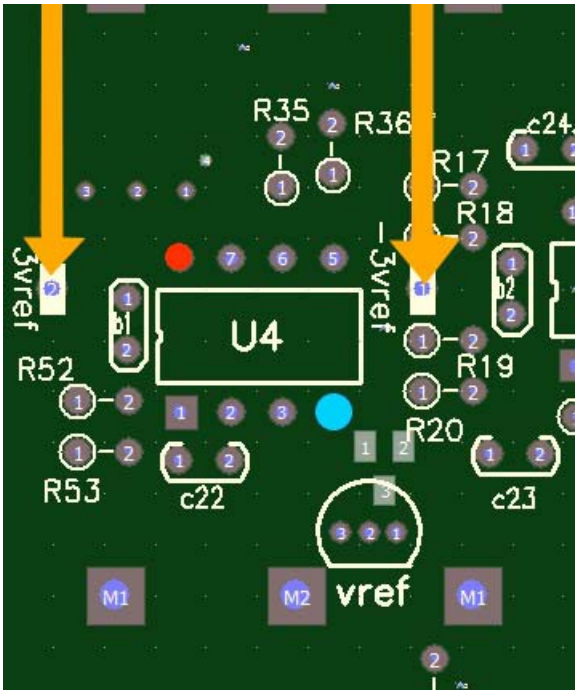
Install reference buffer IC

The one 8 pin Op amp is the reference buffer/driver. Pins on new DIP ICs are splayed out a little, straighten them with an IC pin straightener or by carefully pushing them flat on a bench or table so the pins are at a right angle to the plastic body. Make sure to insert the IC the correct way around, with the dot or cut out facing the same way as the cut out in the IC socket.

Test + and – 3v

There are 2 test points near the 8 pin IC. One is 3 volts and the other -3. The -3 is adjusted by the trimmer under the “freq” knob. Close is good enough, it is fine tuned later.

The +3V is used to protect the teensy from over voltage on the inputs, so it is important to test this before putting the teensy back in.



Install op-amps and teensy

Now install the 3 14 pin quad op-amps, straightening the pins, carefully observing the correct orientation, and that the pins are all going in the socket. If you opted for the higher precision option, the LME49740 goes in socket U3 (bottom right of board).

Carefully reinstall the teensy if you had removed it, checking all the pins are aligning eith their respective headers.

(photo of board with op amps)

Button caps and Knobs

Button caps:
Leave the button caps off until you are absolutely sure everything is working properly. You can depress the buttons with any object such as a pencil, which will fit. This is because removing the button caps without a special tool can destroy the switches, and the button caps melt quite easily from an errant soldering iron which is rather unsightly on your new module.

The button caps are very tight, they usually click when they are in position. The button should protrude about 1mm beyond the panel, when depressed.

Knobs:
Since some of the pots are up side down on the board, round shaft pots were used. This means you have to be a bit careful they are pointing the right way when you tighten the set screw. Turn the pot all the way to the left, then make sure the line on the knob points to the right place for the start position on the panel.

Testing the controls

(note to come about CV test program and control test program)

Checking the potentiometers, switches, LEDs

0.5: Connect a volt/octave signal to the input and play a note you can hear. Without any input, it defaults to a low frequency that might be hard to hear. Connect the main output to something you can listen to (duh!)

1: Set all the buttons in their off position, turn “index” down to 0.

2: Turn the POSITION pot back and forth. The LEDs should light under the corresponding wave selector pots

3: One by one light the low, medium and high wave pots and turn them to verify they select waves, the last wave on low should be silence.

4: Press the buttons below each wave pot and turn up the detune knob. All positions should be detuned when the button is pressed.

4.5: Press the prime button, the detuning should sound different (no beating).

5: Turn up the INDEX and change the MODULATION wave, you should hear different modulations

6: Turn the FREQ knob and the frequency of the modulation should change

6.5: Press the fix button. the frequency of the modulator will no longer be following the pitch, but instead be a wide range fixed oscillator/LFO

7: Turn the tune and fine knobs to make sure they work. Press the tune lock, the tuning knobs should stop working and the LED should come on

8: Get a source for control voltage (envelope, sequencer, LFO etc) and plug it in to the POS CV input. all the attenuverters are normalled to this, check the function of the attenuverters. remember, if something is turned all the way down, turning the attenuverter in the negative direction with a unipolar signal (envelope) will not do anything.

9: Press the FM button. all the controls do more or less the same thing, but the waves are different and some knobs have different ranges.

10 Press the X button. X + AM = detune enable buttons are now “index enable”. X + FM = hi position becomes a pitch bender. (nice for percussion)

-3v reference calibration.

This ensures that the “zero” on the attenuverters is actually close enough to zero.

1 make sure the mid wave is not noise (the last 2 positions are noise with the un-modified program), also sine (fully to the left) might be hard to hear if your module is not connected to a CV source. Unplug any CV sources.

2 Turn the index knob all the way down to the left

3 Turn the lo and mid wave detune enable buttons off

4 Turn the tune knob to something you can easily hear

5 Put the lo wave in to the silent position (fully clockwise) then turn the position knob to lo (anti clockwise) make sure there is nothing plugged in to the cv inputs.

6 Adjust the trimmer below the FREQ clockwise until there is just no sound. if there is already no sound then adjust anti-clockwise it until you just hear something and then back it off. You will have to turn the volume way up (and hear a lot of noise) to do this because the lowest level is 1/2048th of full volume (-66db)

DON'T FORGET TO TURN THE VOLUME BACK DOWN! I WOKE MY NEIGHBORS

Troubleshooting (update from muffs build thread)

one pot or switch does not work
The switches are only pulling a teensy pin to ground, and they are pulled up when the switch is opened by a resistor which is internal to the teensy. That means there are only a few things that can be bad. The connection between the switch and the teensy

(photo of switch-teensy connections to probe)

The pots are connected to the teensy via a resistor. The pot wiper must be connected to the resistor, and the other side of the resistor must be connected to the teensy. There is also a capacitor across the teensy input associated with that pot.

(photo of pot-teensy connections to probe)

can not move from low position, but everything else works
the position attenuverter or level shifter is pinned at maximum. Usually a bad solder joint on r8, r9, r12 r14

wild noisy output even with everything turned off and down
Did you connect the 3 pin header for A10, A11, and VREF? This was missed by one user before these instructions were available.

no output
you might just be in the silent waveform! If the LED for low position is on and the low waveform potentiometer is turned all the way up, silence will fall.

teensy LED does not light
is there 5v power? You can test it on the jumper

one LED does not light
the test program included in the firmware it flashes all the LEDs as well as testing the CV inputs.
Using the arduino software, Locate the file called “orgone_accumulator_official_test_hidden_inputs.ino” and upload it the same way you installed the firmware. All the 4 LEDs should flash on and off.

If one does not flash, then there is a bad solder joint on the LED or its associated resistor (see schematic) or the LED was soldered in backwards.

One CV input does not work
please see above for uploading the test program, but after it is loaded, open the serial monitor on the arduino software. (magnifying glass icon near top right) you should see 4 numbers.

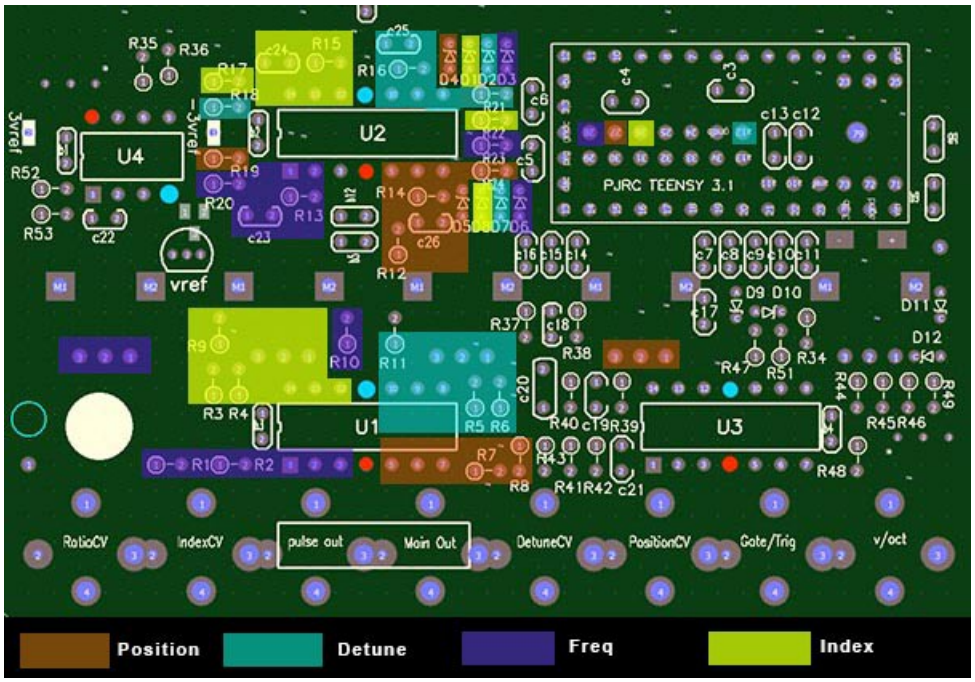
with nothing plugged in to the CV inputs, they should all be hovering around or above 4000

try plugging something like an LFO in to the position CV input and turn all the CV attenuverters up. You should see all the numbers move together up to just over 8000 to around 0.

there are a few things that might look wrong:

stuck at 0 (or so)	backwards soldered diode bad solder joint in analog section
stuck at 8191 (or so)	backwards soldered diode bad solder joint in analog section
floating about kind of random or following slowly	no connection between analog section and teensy SMD header pads
stuck at 4096(or so)	no connection between attenuverter and level shifter attenuverters are associated with U1 and the level shifter is associated with U2.

Here is a diagram showing which areas of the analog section are associated with which CV input.



Appendix

1: Building in other formats:

If you want to build the Orgone Accumulator in other formats such as 5U with jacks or 4u with banana, here are some tips.

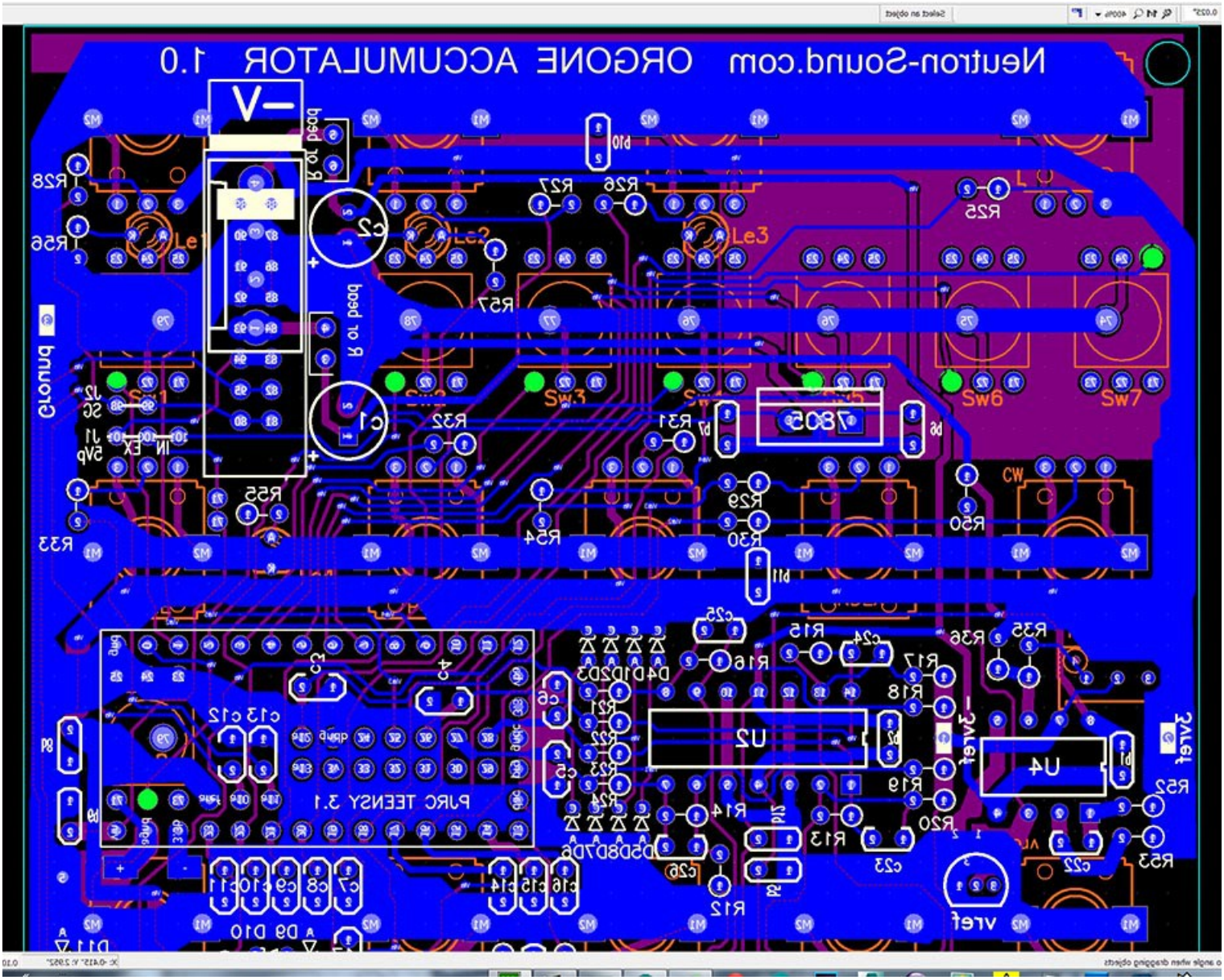
Digital control Pots x10:
To save on wiring, you can use a common ground (CCW) and 3v (CW) for all of the 10 digital control pots (that is all except the attenuverters) just be very careful the 3v will never short to anything. That line is the 3v reference for the entire system, including the teensy.

You can omit capacitors B10 and B11 from the Orgone board, and put them across 0-3v along the pots about halfway and at the end of the “string”

CV attenuverters x4:
use 3 wires per pot as on the board.

Jacks x8:
you can common the grounds on the jacks, but keep a separate ground line for the inputs, CVs and outputs. (n/a banana)

Switches x8:
This is an image showing where to wire the switches. In all cases, the “other” wire can be a common ground (can be shared with pots) you can use your favorite SPST toggle switches.



With 1.01 boards, which will be sent to anyone who purchases “board only” assuming you are building a different format, the correct switch pins are marked with circles on the front silk screen.

Appendix B: overclocking

The sound of the orgone accumulator can be improved by overclocking the teensy to 120 Mhz the rate at which the control voltages are read, and internal values are calculated is almost doubled, the improvement is mostly because the osxillator ISR is not taking up so much time each cycle, so the rest of the code has a greater share of the time.

I have not had any problems with overheating, but I can not guarantee this will shorten the life of the teensy. Personally I think it is a worthwhile upgrade to the sound, especially if you use the gate input for percussive sounds.

To achieve this, you will have to edit the file “boards.txt”

it is in:

```
[where you installed]\arduino\hardware\teensy
```

in the file you will see a line like this:

```
#teensy31.menu.speed.120.name=120 MHz (overclock)
```

all you have to do is remove the “#”

and save.

Next time you open the arduino IDE there will be an additional item in the
Tools>CPU speed for “120MHz (overclock)

Select it before you upload the orgone accumulator sketch to the teensy.

So far I have not found any of the code needs to be edited, the program just executes faster,

note: this was previously set to 144 mhz, with improvements in the code, that speed is no longer required.

Appendix C: murphy boot failure!

Some experienced a problem with the teensy not booting on first power up. It seems that it can be fixed by putting a 10u capacitor on the output of the regulator.

On a new build I recommend replacing the bypass capacitor b9 with a 10u electrolytic (positive facing the bottom of the board where the jacks are)

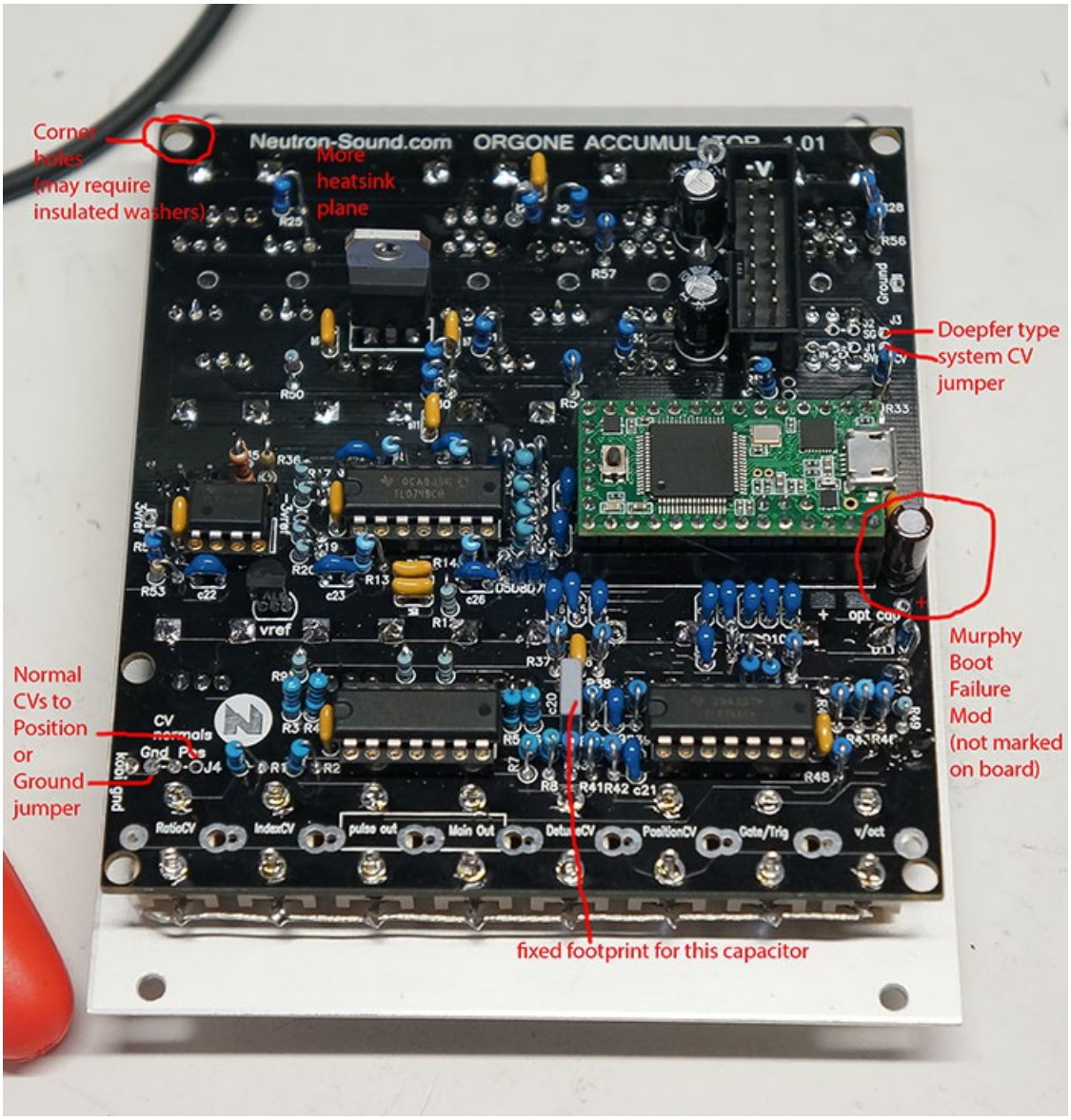
to retrofit an existing board, there are 2 unused holes/pads right next to r55, either of which can be used for the positive lead, and the negative lead can be soldered to the nearby potentiometer tab.

Appendix D: 1.01 board differences

the new board has a few small changes to take note. Most are for non eurorack builders, but please take note of the normals jumper.

Some people did not like having to zero the other control attenuverters while they are normaled to the position control input.

The jumper gives the option of normaling to position, or ground. Normaling to position is probably more useful for smaller systems without many modulators.



Like the gate, The volts per octave no longer defaults to the system power bus (doepfer style) a jumper must be installed to use that.