TTSH Version 4 Build Guide

Covering Synthcube's TTSH V4 release

Version 1.5 November 2019

By fuzzbass

Contents

About This Guide

This document presents one approach to building the TTSH. There are other approaches that are perfectly valid. This approach is designed to minimize the time for construction. These steps have been used to successfully build a large number of TTSH units. Typical build time for the author, using this method, is 32 hours.

The TTSH is not a beginner's project, and this guide assumes general knowledge of through hole electronics construction on the part of the builder. It does not describe how to make solder joints, or how to identify parts. It describes a certain order to install parts, and then test and calibrate the synth.

It cannot be overstated the importance of using a good quality, temperature-controlled soldering system to build this project. There are 3175 solder connections to be made. Use of a craft grade iron will exponentially increase the time to build, and result in countless connections of poor quality or non-connections, and possible component damage. If your soldering skills are not well developed, you will have gained proficiency by the end. If necessary, refer to videos on YouTube with explainers for through-hole soldering technique, and the various fault scenarios to watch out for.

During the main board construction, the soldering is split into two phases. The first phase includes immersible parts only. This is to facilitate a process using organic / water wash flux. This phase may also be built with no clean flux.

This document lays out an approach to soldering that is considered best practice. When through hole components are installed, the tailings for these parts are cut to length prior to soldering. The benefits of this approach are two-fold. First, cutting after soldering can shock or damage solder joints. Second, waste material removed and disposed of will not be contaminated with solder. This is of particular value if you use solder containing lead.

Cutting and then soldering necessitates immobilizing components, or moving them into place as solder is applied. Both methods are used here. The bulk of the solder joints will be formed using a backer board to press components flush with the PCB. Various components are installed in groups, based on their profile in relation to the PCB. For example, horizontally oriented resistors and diodes hug very close to the board, and are installed in the first pass. This process will be described in detail further down. Resistors and diodes that are vertically oriented, or located on the front side of the PCB (LED circuits) are set aside at first, and installed later.

The board is silkscreened with component values, and not component numbers (ex: 1K, not R1). A separate document has been produced showing the count of each component type and value (resistors, diodes, capacitors and transistors) for each section of the synth. This will speed the builder up locating footprints and installing the parts.

If you purchased the Synthcube kit, you received quantity 8 50mm M3 M-F nylon standoffs. These are intended as stilts to raise the main board from your bench so through hole components can be stuffed. These are used only during construction. If you purchased the BOM components yourself, it is recommended you obtain these as well. Brass or aluminum standoffs work even better if you want to spring for them.

Required non-kit items:

- Temperature controlled Soldering system.
- Flux Core Solder wire (lead free or leaded). Expect to consume about 75 feet of .031 solder wire.
- Nylon cable ties - small
- Isopropyl Alcohol and cotton swabs
- ¼ inch Heat shrink tubing (can be subbed with cable ties)
- Cutters
- #2 Phillips driver
- Nut drivers 5/16 and 1/2 inch. (Synthcube kit includes hex nuts for the 3.5mm jacks)
- 2.5M hex key
- Fine tip needle nosed pliers
- Voltmeter or Multimeter, with frequency counter capability
- Oscilloscope
  - A very basic kit scope such as the DSO138 is sufficient.
- ARP 2600 Service Manual

  https://drive.google.com/open?id=1TUjsTKBTVcuaf_m3qcXhh3aoX6WBkeX2

- Corrugated cardboard – continuous and unbent or creased, at least as big is the main PCB.

Useful non-kit items to have on hand:

- Liquid flux or flux pen
- Desoldering braid
- Dust-off (compressed air).
- Magnifiers
- Lead forming tool for ¼ watt resistors (Mouser part 5166-801). You will be sorry if you embark without one of these.

Transistor Coupling and Matching

Footprints showing transistors face-to-face indicate locations for thermally coupled transistor pairs. When such transistors are of the same type, matching them is highly recommended.

To thermally couple two transistors, place them, and add a small amount of thermal paste between them. Place a small (3/16) length of ¼ inch heat shrink tubing over them and apply heat. Do this before soldering. Alternatively, you can couple them using small nylon cable ties. When you are matching transistors, you can couple them off-board. But for locations with NPN-PNP pairs that are coupled, do this on-board, to avoid getting them reversed later during assembly.

Some builders have reported good results without hand matching the transistors. Doing so increases the risk of poor performance (especially in the filter) or tracking. At a minimum, buy name brand (ex: Fairchild/On) transistors from a reputable source, and use units from the same batch. Buying them on feed tape for automated assembly machines ensures the units come from the same batch. Matching will also go faster with tape feed transistors.

While some may forego matching, every location showing transistors face-to-face requires thermal coupling.

There are many ways to match transistors, and all of them require a moderately precise voltmeter (4.5 digits), and a process to minimize your body heat from affecting the results. The author uses the Ian Fritz transistor matching method with a home-built rig constructed on protoboard, and employing a Ziff Socket.

https://dragonflyalley.com/synth/images/TransistorMatching/ianFritz-transmat0011_144.pdf

The Synthcube kit contains the exact number of transistors needed to complete the build. It may or may not yield the required number of matched pairs. The Synthcube kit used in preparation of this document did yield all the required matches.

There are six locations that indicate a complementary pair (NPN + PNP): the 4027-1 sub boards, 4012, 4072 and VCA. These can be filled with randomly selected units, and no matching process has been identified to improve results.

Recommended matched pairs:

- 2N5087 – four pairs, 4072x4
- 2N3904 – eight pairs, RMx2, VCAx1, 4012x5
- 2N3906 – six pairs, RMx2, VCAx2, 4012x1, 4072x1

Part Substitution

If you are fulfilling the BOM yourself, here are some notes regarding part adjustments.

All resistors are ¼ watt through hole types and for the sake of consistency are stated as metal foil 1%. Synthcube may pack a few in the kits that are 5%, or 1/8 watt, and this is fine. It is assumed that most of the resistors used in the ARP 2600 were 5% carbon composition types, with probably a small number either higher precision, or hand selected. Generally you can get away with using carbon comp resistors, but there are no guarantees, and most carbon comp type resistors have a way of soaking up moisture and degrading slowly over time. The only resistors dissipating an appreciable amount of wattage are the four 10R units in the Amp section, and the 100R unit preceding the output to the reverb pan. Any others around the synth should be swappable with 1/8 watt types if you have those on hand.

All Electrolytic capacitors are polarized, and <u>must</u> have 5mm or less diameter and 2mm lead spacing. For longer life, they should be rated for 105c operation, although only the capacitors on the DC-DC converter sub-board will be exposed to any significant heat. The 47u units should be rated for 25V operation, and the rest may be 25V or 50V. You will find the best selection of caps with the correct dimensions will be rated 50V. The best choice of the 10u caps have 5x7mm dimensions, 105c and 50V rating, and 2mm lead spacing. If you get caps here that are max 7mm tall, they will fit better on the sub boards. If you have some caps that meet all these, except are 11mm tall, you can install them on the back sides of the VCO and VCF cores.

With one exception, all non-polarized caps have 5mm footprint, so try to get units with legs formed to that. The exception is the two .022u box film capacitors in the Noise and S&H sections, which have 7.5mm lead spacing. For the sake of consistency and cost, most of the non-polarized capacitors are listed as MLCC-COG types. There are a few .1u MLCCs that are X7R rated, including the 0805 SMD capacitors on the VCO cores. Excepting the SMD caps, any of the MLCCs can be replaced with box film types if you have those on hand. Any type listed as film should not be subbed with a ceramic cap of any rating.

Generally (and where available values overlap):

- A C0G MLCC is an improvement over X7R, X5R or disc capacitor;
- A box film capacitor is an improvement over a C0G MLCC;
- A polystyrene film capacitor is an improvement over a box film type.

If you are a confident builder, you may elect to omit the IC sockets. If you do, buy a couple of extra op amp chips, in case you mangle one removing it. If you are socketing, don't buy low cost sockets with leaf type contacts. Just DON'T. Use only machined contact sockets.

Standard Substitutions

Some capacitor locations on the TTSH are silkscreened to indicate a value that is not common and may be difficult to purchase retail. Substitutions have been made in the BOM/kit to the nearest common value, and in all cases the substitute value is acceptable.

PC Board Marking	BOM Equivalent Part
20p	Capacitor, MLCC, C0G, 22p
30p	Capacitor, MLCC, C0G, 33p
50p	Capacitor, MLCC, C0G, 47p
2n	Capacitor, MLCC, C0G, 2n2
20n	Capacitor, Film, 7.5mm LS, 22n
u1	Capacitor, Film, 5mm LS, 100n

Table 1 - Marked Capacitors and Their Equivalents

In the Voltage Processor, Lag Processor section, the LM301 op amp location is marked for an alternate TL071 op amp. This change is standard and is typically performed on ARP 2600s when they come in for service. The tracking accuracy of the TL071 is advantageous in this location, and the BOM/Kit reflect this substitution.

New in TTSH V4

- All but one of the many bugs in the V3 boards have been resolved (see next section).
- The unused PDIP8 pad in LED section has been removed.
- A field change of one 220R resistor to 680R is formalized (clock rate indicator).
- Mounting points for the Fuzzbass Gate Booster PCB have been added to the EG section.

Known issues in TTSH V4

- TTSH V3 was revision 8 of the main board. This designation should have been updated for V4, but was overlooked. V4 main boards can be identified by <u>the lack of</u> a DIP8 footprint in the main LED section (front of the board, near the Ring Modulator inputs) <u>and</u> the word “REV8” silkscreened on the back of the main PCB, top right corner.
- The 2N3954 dual JFET footprint in the VCO2 section (still) has reversed pins 5 and 6. Instructions are included below to correct for this during construction.

If a subsequent run of V4 boards has the 2N3954 bug fixed, the revision number will be changed as well. It may not be worthwhile fixing these, however. Every time the board design is changed, that risks introducing new bugs. One bug on a board of this size is actually pretty good.

Quirks

Questions may arise due to some of the silkscreened information on the boards.

- As stated above, the main board is marked “REV8” whereas the 4V TTSH is a subsequent revision.
- In the speaker amp section, there are two capacitor footprints marked “u1”. These are .1u (100n) film capacitors.
- There are some resistors marked with an alternate value in parentheses. Ignore the alternate value.
- There is an oddly marked resistor location next to the pad for the reverb return RCA jack. It says “TOBU3 1K”. This is just a 1K resistor location.
- Some capacitor values are marked with non-standard values, and these are traditionally substituted with standard values. See the preceding section “Standard Substitutions”.

Main Board Power System

The power rails are distributed to all sections in a middle layer of the main board. This is hidden from view, and cannot be modified. This means you cannot drill holes in the main board without risk of permanent damage to the system. It also means you cannot selectively power individual sections.

There are two power distribution systems. Power rails are tapped on the DC-DC converter both pre and post the LDO regulators. On the 5 pin headers that connect the DC-DC converter to the main board, the pre-regulator rails are identified as +15.00V and -15.00V. When you measure them, you will find them to be more like +/-15.8V. This is intentional, and provides for the ~.5V drop incurred by the LDO regulators. The regulated outputs are V+ and V-. All of these are referenced to the "0" pin. Misleadingly, there are two test points on the DC-DC converter marked +15.00 and -15.00, when in reality they are V+ and V-, respectively. These test points are used to initially set the regulated output voltages for V+ and V-, via the 500R trimmers. +15.00 and -15.00 power the LED sections. V+ and V- power everything else.

Although the power and ground planes are internally routed in the board, the old power connector pads from TTSH V1 (where power was wired up with jumpers) are still in the design, and now function as vias from the internal power layer. These consist of twelve sets of pads for .1 pitch 2x3 pin headers. There is one in each of the major sections on the main board. Nothing needs to be installed in these locations, but they can be used to tap power and route it to optional boards. This is how the Gate Booster and VCO sync modifications are powered.

The location/footprint on the main board for the .156 five pin locking (Amphenol) header for power connection to the DC-DC converter board is silkscreened backwards. If the header is installed in the indicated orientation, the wires coming of the connector may create a problem with case fitting. Install this header backwards to avoid problems.

Planning for Modifications.

A selection of common TTSH mods are presented at the back of this guide. Glance through this and decide if any of them interest you. Almost every builder adds the Gate Booster. Some mods are more esoteric or involved. The Electronic Switch mod is easy, has zero cost, requires no trace cuts, sacrifices no functionality, and gets you a sub-oscillator! This one is highly recommended, and it remains a mystery why ARP didn't implement the Electronic Switch this way to begin with.

For mods requiring trace cuts, these cuts should be made before building commences.

Mods included in the guide:

- Extra power taps
- AC Coupling – VCF Audio Input Mixer
- Gate Booster
- Midi Implant
- Improved Calibration Accuracy
- External Clock capability – Electronic Switch
- Noise Improvements for Reverb

- Adjustment Range - LEDs

See these sections for installation steps and BOM info.

Construction – Getting Started

1. Whether you purchased a full kit, or assembled the BOM components yourself, refer to the Master BOM document and verify you have all the required parts on hand. There is generally no need to test the components. Nonetheless it is a good idea to count them and verify at least one of the components in each bag is marked with the correct value. If you have not memorized the resistor color codes, this will help you to do so. Alternatively, use your meter. You can embark on the build if there are some parts yet to be received, but doing so will increase the risk of an error or omission during the build. If you purchased the Synthcube kit, you will want to report any missing parts as soon as possible.
2. Visually inspect all PCBs and look for physical damage, scratches that cut through traces, illegible screen printing, cracks, etc.
3. Visually inspect the panel for damage, paint or silkscreen problems. There is a step where the panel is lowered down and fit over all the jacks and sliders, and then secured using screws. To avoid frustration at this step, make sure all the panel opening swill accept the parts that poke through and that the slider tangs do not rub or bind across their entire range of motion. It is possible that some of the paint has gone into the drilling and needs to be filed or removed. Using a 3.5mm Jack, the 6.3mm TRS jack, one of the sliders, and an M3 screw, check each hole and slider groove to make sure there is clearance for the parts. Check the 17 M3 screw hole punches to make sure there is adequate clearance for the screws. Where issues are found, file out the openings to resolve this. The evaluation panel sent to the author for review had a few paint problems with jack and slider openings near the left and right edges of the panel. You want to make sure you are not dealing with obstructed openings when you get to fitting the panel to the main board.
4. If you are planning to build with hand-matched transistor pairs, match them now. Recommended matched pairs:
    a. 2N5087 – four pairs, 4072x4
    b. 2N3904 – eight pairs, RMx2, VCAx1, 4012x5
    c. 2N3906 – six pairs, RMx2, VCAx2, 4012x1, 4072x1
5. Cut apart the 4027-1 boards and the VCF/PSU boards (which are scored). Carefully bend and break apart the boards. Use sandpaper or a Dremel tool to smooth the edges. There are six little riser boards intended for the slide switches, on the main PCB, next to one of the speaker cutouts. You can cut these away with nippers or the Dremel. Discard them – they won't be used in this build process. Lightly clean each PCB with isopropyl (not rubbing) alcohol and wipe dry.
6. If you plan to install the optional Gate Booster, or add AC coupling to the inputs of the audio mixer in the VCF, there are some trace cuts to be made for these mods. A Dremel tool with a small cutoff wheel is a good way to make these trace cuts. Be careful to make these cuts very shallow, to ensure no damage to the center layer of the main board.

a. The standard install for the Gate Booster requires that the trace running from the tip in of the Gate jack to pin 1 of the Gate Select switch be cut. The booster will be inserted in-line here. See the picture below for reference.

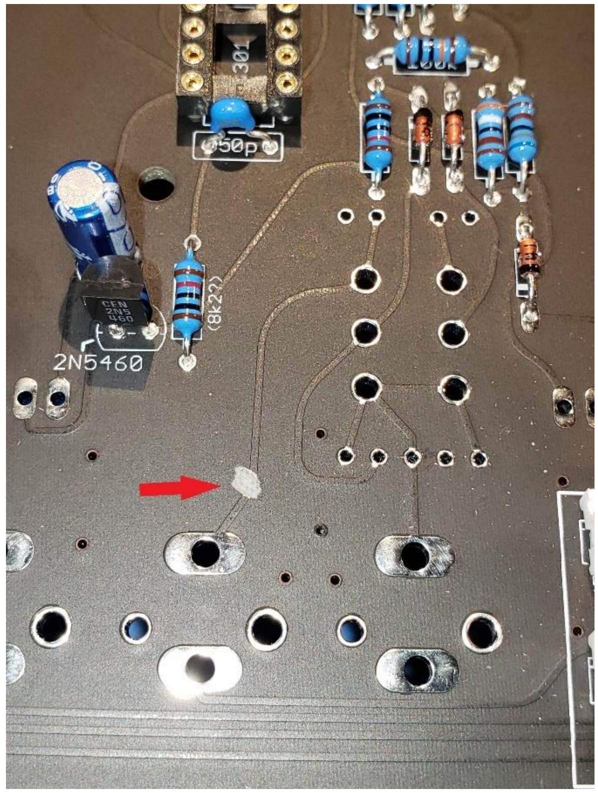

Figure 1 Trace Cut for Gate Booster

b. To implement AC coupling for the VCF audio mixer, the board is marked for these four trace cuts en route from the mixer's audio input jacks, at the footprints for the required capacitors. Four 1uF box film capacitors are recommend, and these are included in the Synthcube BOM and full parts kit.

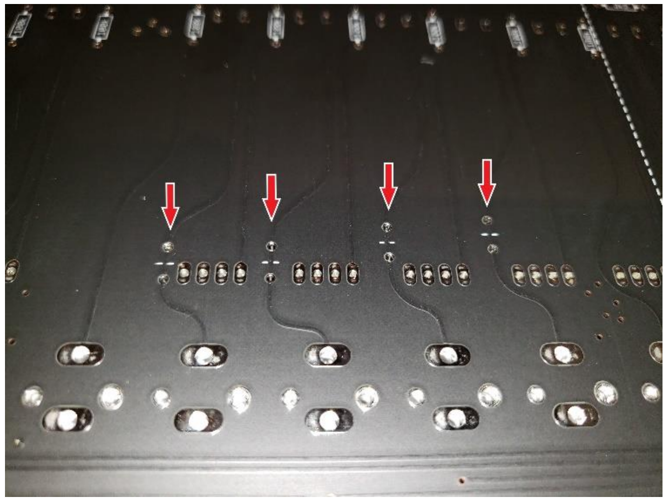

Figure 2 - Locations for trace cuts to implement AC Coupling for VCF Audio Mixer

7. Select a piece of cardboard larger than the main PCB, that is flat and without bends or creases. Mark the outline of the main PCB on this board and cut it out. This will be the backer board for installing the bulk of through hole components on the main PCB. You will also want a roll of masking or painter's tape on hand to use the backer board.
8. Build the 4027-1 VCO cores. Start with the 0805 SMD capacitors on the bottom side of the boards. This can be done with a standard soldering iron, but be sure to dial the temperature down to 600f/315c for the caps. SMD caps can be cracked or damaged from excessive heat – work fast. Apply liquid flux or flux pen (not included in kit) to the pads, place the part, and bring the iron to it with a small bead of solder on the tip. You will need to hold the capacitor in place using tweezers while doing this. When done, inspect under magnification. If you have a capacitance meter, you can now measure between the pads on the 3-position header location (this is the power inlet). You should see 100n between 0V (center) and the rails. Now, test for shorts between the pads on the power header footprint.

   The 2N3904 and 2N3906, and the 1k87 tempco are to be thermally coupled using paste.

   Do not install the pin headers at this stage.

4027-1			
Resistors		Sockets	
220r	3	DIP14 socket	3
1k	9		
1k37	3	Capacitors	
1k5	3	5p (C0G)	3
1k65	3	680p (Styrene)	3
1k87 (tempco)	3	10n (C0G)	6
3k9	3	100n (0805 X7R)	6
8k87	3		
12k	6	Capacitors (electrolytic)	
15k	3	10u	6

30k1	3		
33k2	3	Active Components	
45k3	3	CA3046	3
47k	3	2N3904	3
61k9	6	2N3906	3
68k	3	2N4125	3
100k	3	2N5459	6
121k	3		
475k	3		
3M3	3		

Table 2 - 4027-1 VCO Cores BOM

9. Build the PSU sub-board, starting with the SMD LDO Regulator chips. The heavy tab at the top of them needs to be soldered to the large pad footprint. First, solder one or two of the smaller pins. The apply a generous amount of flux to the tab, turn up your iron to 630f / 325c and heat the tab+pad for a bit before introducing solder. The time to do this depends on the heat of your iron, and the size of your tip. Larger tips will heat the work faster and produce better results. Maintain constant and firm pressure on the part with tweezers as you do this, and when you remove the iron, keep the pressure on until you are sure the solder has solidified. This pressure ensures that your iron heats the tab, and the tab heats the pad. If the pad does not heat up, it won't accept solder. When the work cools, make sure the chip is bonded to the big pad by pulling up on it. Under magnification, inspect the smaller pins to make sure they are all soldered and there aren't gobs of solder bridging the pins. Then, install the remaining PSU parts. Install two wire jumpers as shown. Trimmed rectifier diode (ex: 1N4001) leads are a good choice for these jumpers, due to their increased thickness.

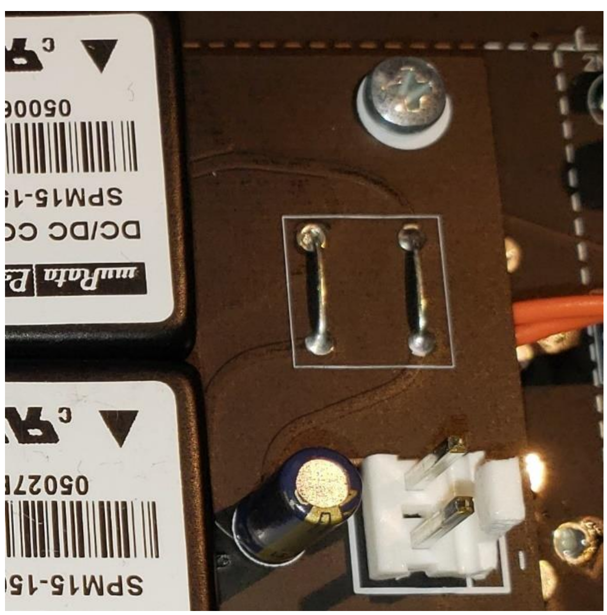

Figure 3 - Jumpers on PSU Board

PSU			

Resistors		Inductors	
240r	2	100uh	2
2k4	2		
27k	2	Active Components	
		580-SPM15-150-Q12P-C	2
Trimmers		LM2991	1
500r	2	TL1963A	1
Capacitors (ceramic/film)		Locking Headers	
4n7	4	.156 x 2	1
220n	2	.156 x 5	1
Capacitors (electrolytic)			
10u	2		
47u	3		

Table 3 – PSU BOM

10. Build the 4012 VCF Core. There are six matched transistor pairs (5x 2N3904 and 1x 2N3906). Additionally, there is one thermally coupled NPN-PNP pair. If possible, try to thermally couple the 1k87 tempco resistor to the NPN-PNP pair. Doing this will mean the tempco's leads will traverse some other resistors. Consider adding some heat shrink to the tempco's leads, or just let it hug the board. Author's preference is to bring the NPN and PNP into contact with some paste between them, and then shroud them in heat shrink. Later, bring the tempco over and with a dab of paste, lean it into the side of the pair. It's probably better that the two transistors are in direct contact, rather than having the tempco in between them.

    <u>Do not</u> install the pin headers at the stage.

4012			
Resistors		Sockets	
220r	1	DIP8 socket	1
470r	6		
820r	1	Capacitors	
1k	1	10p	1
1k8	1	47p	3
1k87*	1	10n	4
2k2	1	100n	2
3k32	4		
10k	1	Active Components	
15k	3	1N4148	4
23k2	1	BC558	1
30k1	4	LM301	1
56k	1	2N3904	14
100k	3	2N3906	4
196k	1	2N3958	1

220k	1		

Table 4 - 4012 VCF Core BOM

11. Build the 4072 VCF Core. There are five matched transistor pairs (4x 2N5087 and 1x 2N3906). Additionally, there is one thermally coupled NPN-PNP pair. Thermally couple the 1k87 tempco resistor to the NPN-PNP pair.

    Do not install the pin headers at the stage.

4072			
Resistors		Sockets	
100r	1	DIP14 socket	1
220r	9	DIP8 socket	1
1k	1		
1k3	1	Capacitors	
1k87*	1	4n7 (Film)	4
2k2	5	10n C0G	1
4k7	1		
10k	3	Capacitors (electrolytic)	
12k1	7	10u	4
13K	3		
39k	1	Active Components	
56k	1	1N4148	1
100k	4	LM1458	1
196k	1	LM3900N	1
220k	1	2N3904	1
270k	1	2N3906	6
22M	4	2N5087	8
Trimmers			
100k (Piher)	1		

Table 5 - 4072 VCF Core BOM

12. If you plan to install a Gate Booster, build the PCB now. The Fuzzbass Gate Booster is silkscreened with reference designators for parts (ex: R1). The parts list below assumes you are performing the standard install, and does not include optional parts for LED gate indication (R17: 1k, and Q3: 2N3904).

    Ferrites may be replaced with wire jumpers. 10u electrolytic capacitors may be replaced with 1U 50V types. The DIP socket is optional. The point of headers and wire housings is to allow disconnect for servicing. If you are using a header and wire housing on the main board for the power connection, you can directly solder these wires to the power header location on the Booster (you only need one disconnect).

Fuzzbass Gate Booster		
Resistors		
1k	2	R7, R15

15k	1	R11
18k	5	R3, R4, R5, R12, R13
47k	3	R6, R8, R14
82k	1	R2
100k	3	R1, R9, R10
1m	1	R16
Ferrites	2	L1, L2
DIP8 Socket	1	IC1
Capacitors		
330p (film)	1	C1
100n (x7r)	2	C4, C5
10u (electrolytic)	2	C2, C3
Actives		
1N4148	1	D1
2N3904	2	Q1, Q2
TL072	1	IC1
Wire Housing .1 x2 IDC	3	
Wire Housing .1 x3 IDC	1	
Friction Lock headers .1 x 2	3	
Friction Lock headers .1 x 3	1	

Table 6 - Fuzzbass Gate Booster BOM

Construction – Main Board Phase 1

This phase will affect a bulk installation of parts that can be immersed in a water bath. If you are using organic flux solder, phase 1 can be completed with it before washing the board and proceeding to completion.

1. From the hardware parts, identify 4x M3 screws and 4x M3 x 50mm nylon hex standoffs. Install the four standoffs to raise the board from your work surface, rear side of the board facing up.
2. Lay out parts from the table "Main Phase 1", all resistors and 1N4148 diodes. Omitted from this list are parts from the LED sections (front side of the board), sub boards, and vertically oriented resistors. Stuff all parts as shown in the table. When done, you should have almost all of the resistors and all diodes in place on the back of the main board. Take a minute to double-check the orientation of all diodes.

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
10r								2				4	6
56r											2		2
100r	1							1					2
120r						1							1
220r	2						2					2	6
330r											1		1
470r							1	1					2
680r	2							1					3
820r								2					2
1k	4	3	4	2		3	4	3	1		3	1	28
1k2								2					2
1k5						1			1				2
1k8											2		2
2k2			3			1							4
2k7	1												1
3k3							1						1
3k9		1	1	1		1			1		1		6
4k7	2					1	3				2		8
5k36										2			2
6k8		1				1							2
8k2		1											1
9k1						1							1
10k	2		1			4		1			4	1	13
12k						1							1
15k			3			1				2	1		7
18k	4						1						5
22k			1			1		1			6		9
27k	2												2
30k1			3	1							2		6
33k		2	2			1	1	2			1		9
39k	1		1			1							3
40k2						1							1
45k3				1									1
47k	1						1	1	1		4		8
54k9							1						1
56k							2						2
61k9											2		2
68k			1			2		2			1		6
68k1						1							1
82k	2										2		4
84k5		1	1	1									3
100k	7	1	1	1	8	2	4	9	1	9	5	2	50
120k	1		1			1							3
121k			1										1
150k		1	1	1	2		1				3		9

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
180k	1		1			2	1	2					7
182k							1						1
196k			1										1
220k		3	3	3			1	5				2	17
270k	1					1			1				3
470k		1	1	1		1		3			2		9
680k							1						1
1M	2			1		2				1			6
1M5	3						1						4
2M2								2					2
3M3		1	1	1	1						2		6
4M7									2				2
5M6									2				2
10M	1												1
22M											4		4
1N4148	8	1	1	2		13	10	2			6	4	47

Table 7 Main Phase 1 (resistors and diodes)

3. Place your cardboard backer board over the work, and using tape, secure the edges. While supporting the cardboard in the middle with one hand, carefully flip the work over and place it down flat on your work surface, pressing down across it to eliminate any gaps between resistor bodies and the main board.
4. Working from one side and moving across the board, clip all the component tails so they are ~1-2mm proud of the board. Remove the clipped tailings, and solder everything. Inspect carefully under good illumination to find any missed solder joints. When I do this, I move section by section, but you may also just move across the entire board in one pass.
5. Flip the board back over and remove the cardboard. Check by feel for any loose components. Hold the board up to a light source. Any holes that permit light yet appear to be obstructed indicate a missed solder joint.
6. Remove the four M3 screws holding the stilts on, and replace them with 50mm stilts (front side).
7. Referring to the LED Section table, install all parts (resistors, diodes and transistors) for the LED sections, front side of the board. Reach under and splay the legs out to keep the parts in place (don't use the backing board here). Don't miss the two resistors and one diode that are vertically oriented, located next to the clock rate slider footprint. Double check your diode orientation.

Slider LED sections, Front Side	
Value	Qty
220r	6
680r	1
3k3	1
10k	1
12k	1

47k	8
1N4148	2
2N3906	1
BC337	8

Table 8 LED Sections

8. Flip the board face down, cut and solder the LED section parts.
9. Remove the four stilts on the front side, replace with M3 screws.
10. If you elect to omit the DIP8 sockets, install the op amps now. Otherwise, proceed to the next section. Install the backer board, tape, and flip the work. Solder all pins at 600f/315c. In the following step you will be installing capacitors only.
11. Place the DIP8 sockets. Then, place all of the ceramic/C0G capacitors. Sockets and ceramic caps have roughly the same board profile so can be soldered at the same time. If you purchased the Synthcube full kit, the 10uF electrolytic capacitors included have a lower profile than other polarized caps. They can be worked in with this batch.

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
Socket	6	1	4	1		2	1	7	3	3	3	2	33
3p3		1		1									2
5p	1												1
10p	1						1	5	2			4	13
12p	1												1
20p	3							2	1	4	3	2	15
33p	2		3			1	1	4	1	3	2		17
47p	5				1	1		5	1				13
100p	1		2				2	2			2		9
220p	1						1						2
330p	1					1		1					3
1n	1					1					4		6
2n2	1								1		1		3
3n3									1				1
10n	1					1		2	1		6		11
15n		1	1	1									3
100n	2							1		1			4
10u	6					1		2					9

Table 9 Main Phase 1 Ceramic Capacitors (10u are electrolytic)

12. Install the backer board, tape, and flip the work. Find and clip the leads to all the capacitors. You can find the count of caps in each section looking at the table. Solder everything, inspect, flip the board back over and remove the cardboard.
13. Using the same process as above, place all the small signal transistors (singles, duals types and matched pairs). Set aside the MJE172 and MJE182 transistors for now – as they stand much higher from the board.

    <u>Note that in the first batch of V4 (Synthcube) boards, the footprint of the 2N3954 in the VCO2 section has pins 5 and 6 reversed. If you are building this main board, be sure to</u>

form those pins on the 2N3954 to fix that bug. This dual JFET is used to shape the triangle wave into a sine wave.

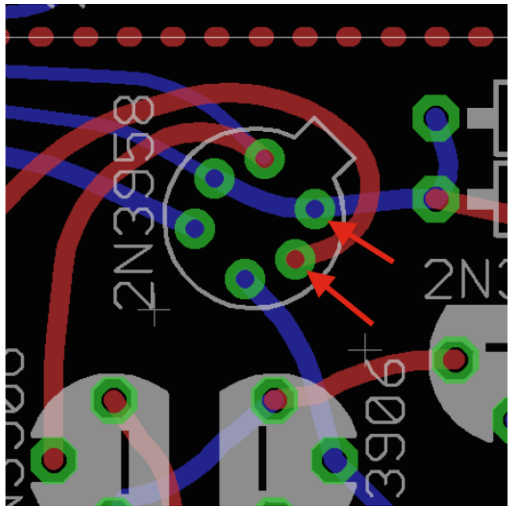

Figure 4 - Reversed Pins 2N3954 in VCO2. Note the image shows a different transistor, but the reversed pins are as indicated.

I start with the PNP/NPN complementary pairs (VCA and VCO2) adding thermal coupling as I install them. Then I install the two CAN-6 JFETs (VCO2 and S&H). Next, the NPN/NPN and PNP/PNP matched pairs. Lastly, all the singles.

When all transistors (singles and pairs) are in place, place the film caps in the Amp section. Put on the cardboard, flip it over, clip the leads, and solder. Solder the capacitors first, then proceed to the transistors. When soldering these parts, keep your iron temperature at or below 600f/315c, and work quickly. Going section by section, solder the first lead of each transistor, then go back and do the second leg of each, then the third. This will reduce heat stress on the transistors, and this is especially important for matched pairs and JFETs.

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
BC558							3	1			4		8
2N3904	6					1	4						11
2N3906	6		1			3	5						15
2N3954			1										1
2N3958											1		1
2N4392						1					3		4
2N4870											1		1
2N5172			1			3	1	2	1		7	2	17
2N5460						2							2
100n(film)												2	2

Table 10 Main Phase 1 - small signal transistors (100n are film capacitors)

- 2N3904 Matched pairs: RMx2, VCAx1.
- 2N3906 Matched pairs: RMx2, VCAx2

- There are complementary pairs (2N3904+2N3906):VCAx1, (2N5172+2N3906): VCO2, that must be thermally coupled before soldering.

14. Repeat the same process to install all remaining electrolytic/polarized capacitors on the main board. Pay attention to orientation when placing these parts. When done, take a minute to double-check the orientation, to avoid smoke and fire later. After this stage, you are done with the backer board, and it can be discarded. This assumes you have previously installed the 10u capacitors, since they stand 7mm tall, whereas all of the below are 11mm tall. If you have 11mm 10u capacitors (see table 8), add them to this lot.

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
100n	1												1
1u	3	2	2	2	2	5	2	2	5		2	4	31
1u5											1		1

Table 11 Main Phase 1 - Electrolytic Capacitors

15. Install reserved parts: vertically oriented resistors, the MJE172 and MJE182 transistors and the 22n film caps with 7.5 LS in the Noise and S&H sections.

Val	RM	VCO1	VCO2	VCO3	VCF	EG	VCA	MIX	Noise	VP	S&H	Amp	Total
100r											1		1
470r	1												1
1k5											1		1
4k7											1		1
10k											1		1
15k											1		1
22k						1							1
1M2						1							1
22n									1		1		2
MJE172												2	2
MJE182												2	2

Table 12 Main Phase 1 - Oversized parts

16. In you intend to install the four optional film capacitors to AC couple four of the inputs of the audio mixer in the VCF, install these caps now. Remember that for them to function, traces running under them must be cut.

This concludes Phase 1 Main Board. If you are using organic flux-based solder, now is the time to remove it and change to no-clean solder. Use compressed air to blow water out of the nooks and crannies, and allow the board to dry completely.

Construction – Main Board Phase 2

1. Start by installing the stilts, both sides.
2. Install the Amphenol headers (1x5 .156 (qty 1), 1x2 .1 (qty 5) and 1x3 .1 (qty 1). The large five pin header for the power input should be installed reversed from the orientation indicated in the silkscreening. This will avoid problems fitting the synth into its case. The 1x3 .1 header goes

onto the 2x3 header pad in the EG section. This will be used to power the Gate Booster. It helps to put the headers and wire housing together before soldering them in, to avoid burning your fingers while making final positioning adjustments.

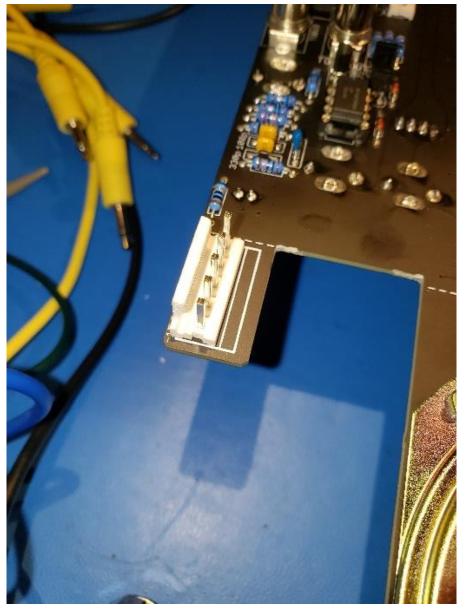

Figure 5 - Install Power Header as shown, not as indicated in silkscreen

3. Install the .1 pin headers (males and females). Start by cutting the bulk pin header material from the 40 pin sticks provided. You will need sections of (6) 2-pin, (3) 3-pin, and (6) 6-pin. Fit these together with the pre-cut female headers provided. Place the headers such that the female section is soldered to the main board, and the male section is soldered to the sub board. Double check this before soldering – a mistake here can lead to pain later. Without going into detail, the idea is to set the sub board on the headers, and immobilize it (cable ties for the 4027-1s, and 12mm spacers with screws for the VCF cores). Once you have them positioned and held in place, solder the headers – both sides. Don't forget that there are two VCF cores that need pin headers soldered in.
4. Install the reverb jacks. The red one goes in the location marked "to out".
5. Install the trimmers.

Type	VCO1	VCO2	VCO3	VCF	VCA	P / RM / EF	S&H / Clock / SW	LED	Total
250R				1					1
10K		2		1	2			1	6
25K	1	1	1						3
50K						2			2
100K	2	3	1	2	2		2		12
250K						1			1

Table 13 - Trimmers on main board

6. Install the 100k audio rotary potentiometer in the Preamp section.

7. Install the sliders. Working in small groups or quadrants of the board, place the sliders and then turn the board up 90 degrees to access the back side. While holding each slider down to the board, and using fine tipped plyers, bend one pin at each end of the slider outward. This works best if the pins are diagonally opposed. Lower the board down and check that each slider is flush to the main board, it's tang is upright, and it won't wiggle or move when touched. Adjust the bending of the pins until all sliders are in place and immobilized. Turn the board over and solder the sliders.

Type	VCO1	VCO2	VCO3	VCF	EG	VCA	P / RM / EF	Mixer / Rev	Noise	Voltage	S&H / Clock / SW	Amp	Total
100ka	3	3	3	5		2		2	1			2	21
100kb	2	4	3	5	1	3	3	3	1	3	2		30
1ma					5					1			6

Table 14 - Sliders

8. Install 12mm standoffs to support the VCF Core (2), Gate Booster (2) and DC-DC converter (3). Fasten on back side of the board using hex nuts.
9. Mount the speakers, using 12mm spacers on the front of the PCB, and fasten with nuts in the back. Tighten these nuts well as they will take some vibration.
10. Wire up the speakers and headphone jack as shown:

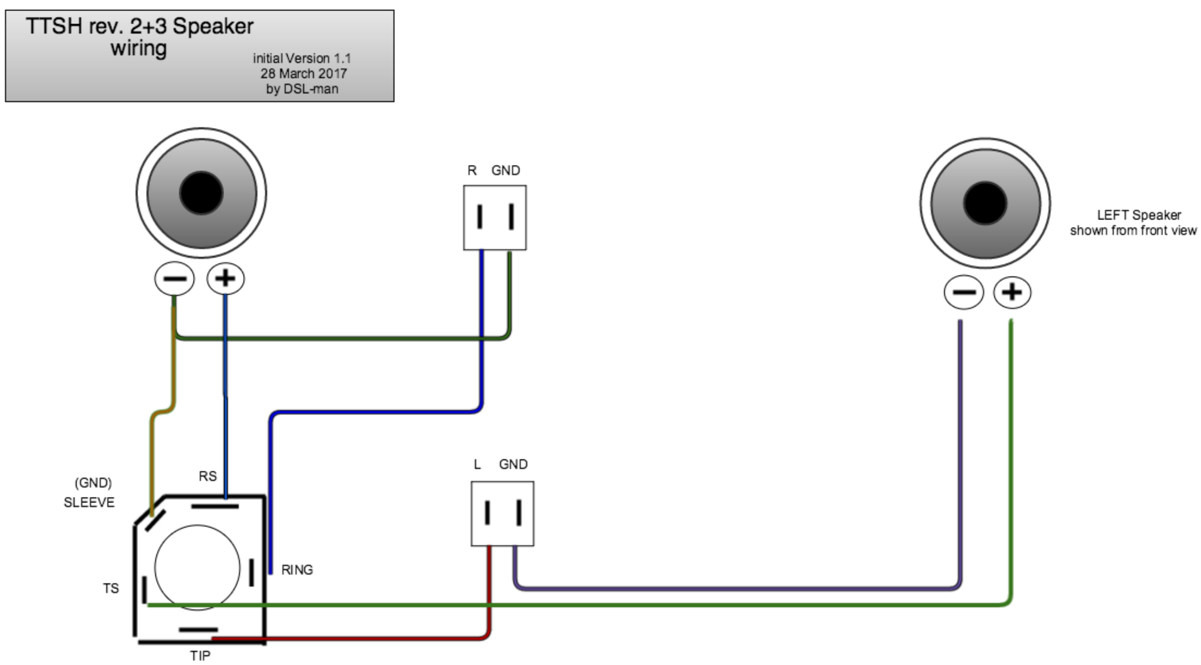

Figure 6 - Courtesy of DSL-Man

11. Replace the stilts on the front side of the board with 12mm hex spacers (4) such that the female side of the spacer faces up.

12. You will need to file or Dremel off some material from the corner of one of the 3.5mm jacks (the VCF output jack). This is to avoid a conflict with the fastener for one of the 12mm standoffs supporting the VCF core sub-board, which is too close. See the picture.

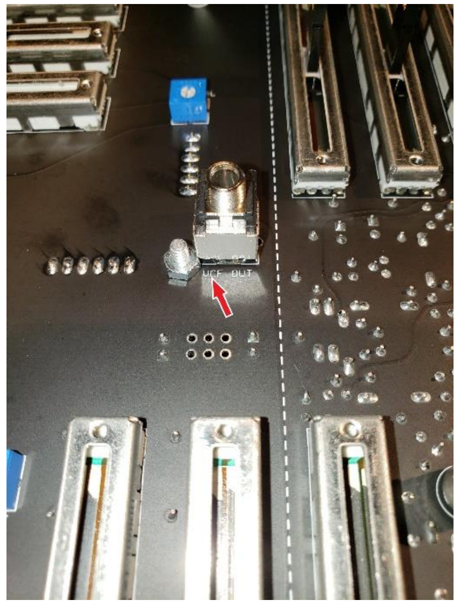

Figure 7 - To seat the VCF Output jack, you will need to remove some material from the corner of the jack to relieve conflict with the mounting hardware for the VCF Core sub-board.

13. Once you have the VCF Output jack fit, place all remaining 3.5mm jacks, the momentary push switch in the EG section, and all six slide switches.
14. Place the power LED, paying attention to polarity.
15. Carefully, gently, patiently, drop the panel down onto the main board, making sure all the sliders, jacks and switches come through. The first time you do this, it will take some time and jostling about. Work from left to right, and install a few nuts to the jacks to prevent them dropping back out. Be patient. Once the panel is fully seated, install the four previously removed stilts through the panel and into the 12mm spacers below. Install ~12 of the nuts to various 3.5mm jacks, tightening them <u>LIGHTLY</u>. Make sure you hit all the corners and the middle sections. The objective here is to get all the jacks seated properly, and set a consistent 12mm distance between the panel and main board, all the way across.
16. Flip the work over and place it down on the stilts. Press down across the board to eliminate any gaps.
17. Start with the slide switches. Make sure each of them falls down so that its bezel is contacting the back side of the panel. Apply solder to <u>only</u> one pin. While the iron is still on the work, reach under and push up on the switch to bring the solder pin back up to flush with the PCB surface (not protruding). Remove the iron and allow the solder on the one pin to harden. Test the switch to make sure it can be moved with your finger. Once you are satisfied with the switch placement and elevation, solder the remaining pins, being sure to fill in the through holes. Repeat for the remaining switches. This might seem convoluted, but it is in fact much easier and less error prone than using the little riser boards that we discarded earlier.

18. Repeat this process for the momentary gate push switch.
19. Make sure the power LED is dropped down into position on the panel. Bend the leads out, cut and solder. If your kit includes a flat top LED, drop it just down to flush with the panel front, for improved appearance.
20. Solder all pins, all 3.5mm jacks. Make certain the panel and PCB are pushed tightly together to 12mm clearance, all around. Periodically check this, especially near the edges of the panel, where the stilts are not holding it up.
21. Socket all op amps, main board. Start with the TL071 in the Lag Processor. All the rest are LM301.

At this point the main board solder assembly is complete. Remove the top side stilts, the jack nuts and separate the panel. The panel does not go back on until all checkout and calibration is complete. It will be much easier to install the next time. Now put the stilts back on the front, into the 12mm spacers.

Construction – Main Board Finishing Up

1. Build wire jumper assemblies:
    a. 11 inches / 30 mm five-way, .156 wire housings, for the DC-DC converter to main board power connections. These terminations flip; position 1 on one housing equals position 5 on the other.
    b. 12 inches / 32 mm two way, .156 wire housing to barrel connector, for the external AC adaptor to the DC-DC converter. This jumper will be partially disassembled later, to put the AC power switch in line. For now, just wire it straight to the connector without the switch.

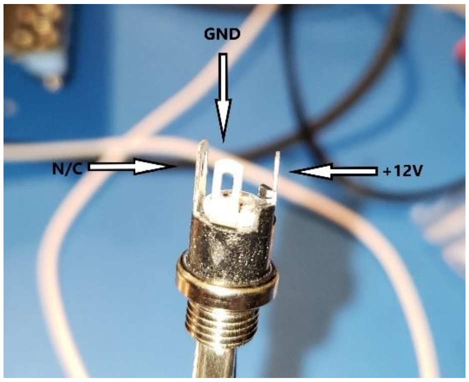

Figure 8 - Barrel connector for Power Entry

    c. If you are adding a gate booster, build a power jumper for it, using .1 x 3 wire housings. Whether this connector is straight through or flips depends on how you installed the locking header in the EG section. Refer to the section on installing the Gate Booster for more information.
    d. If you intend to custom wire your reverb jumper using RG174 coax and the basic plugs, do that now. You may also use a standard duplex RCA audio cable. RG174 offers better

noise shielding than the standard cable, and allows for the shortest possible run from the return plug on the reverb unit. The return line carries a very weak signal and thus is the primary source of noise in the reverb section. Minimizing the length and optimized placement can dramatically improve the noise rejection. Information on terminating the coax to the plugs is included in back of this manual.

2. Install sub boards (4027-1, VCF core – start with the 4012, and DC-DC converter. Only fasten down the DC-DC converter. If you are installing a gate booster, complete that installation now. Details are included in the back of this guide.
3. Connect the DC-DC converter board input to the barrel connector assembly, but do not install the five-way jumper connecting its output to the main board.
4. Power-up checks:
    a. Before applying power, and before connecting the DC-DC converter board to the main board, using the DMM, in resistance mode, check the main board for shorts across the power rails. Test at the five-way .156 locking header. Place one probe on the middle GND pin and measure resistance to all the other four pins. Then, check between all pin combinations. You should not see less than a few hundred ohms and some will have much more. If you see any reading of a few ohms or less, you have a short that must be identified and corrected before proceeding. The most likely cause will be a solder bridge somewhere. If this is the case, you should be able to find it through inspection.
    b. Repeat the resistance checks on the DC-DC converter board outputs.
    c. Set the DMM in voltage mode, connect the negative probe to the center GND output pin, and the positive probe to the pin (not the test point) marked +15.00. Connect the AC power supply to power up the DC-DC converter. You should measure ~+15.75V. Move the probe to the -15.00V pin (not the test point). You should measure ~-15.75V.
    d. Move the positive probe to the V+ pin. Adjust the top trimmer to achieve +15.00V. Move the positive probe to the V- pin. Adjust the bottom trimmer to achieve -15.00V.
    e. Disconnect the external AC adaptor.
    f. Install the five-way jumper from the DC-DC converter output to the main board connection.
    g. Reconnect the external AC adaptor.
    h. Repeat step D. Once the full load is on the PSU, it is usually necessary to adjust the V+ and V- voltage trims again. A good 0V reference point for these measurements is the "gnd" pin on the CV header. Connect a grabber probe there, and use a pointed probe to measure the voltages present on the various IDC connectors in the power jumper.
    i. Swap VCF cores to the 4072 and make sure it does not introduce a power issue. Then swap the 4012 back in.
    j. Peek underneath and make sure all slider LEDs and the power LED are lit. Then disconnect power.
5. Take a Sharpie and sign the main board builder's label. You have reached the point of no return!
6. Proceed to Check Out and Calibration. It is important to run through all checks and calibration before installing the panel. There is possibility you will encounter problems that need to be

fixed with the panel off. There are 81 nuts and 13 screws involved in removing the panel. Be patient. Don't tie down the 4027-1 VCO core modules until you have completed calibration of all three VCOs.

Check out and Calibration.

Follow the checks and calibration steps as documented on the ARP 2600 service manual. You will need an oscilloscope and a frequency counter. You may follow the VCO calibration presented in the ARP manual, or use the alternate calibration method presented below. The one contained here allows for getting the instrument into concert pitch quickly. This does not change the range of the instrument, but does more carefully define the VCO pitch when the base frequency slider is all the way left.

The first thing to check when you power up is whether you have a sawtooth wave on each of the VCO outputs (10V P-P, +5V DC offset) , and whether the Freq Cal trimmer can get the VCO to 32.7hz (0V or nothing on KBD CV, Coarse Freq: left, Fine Freq: center, switch: KBD ON, all modulation inputs down). Perform this check using the oscilloscope and frequency counter. If the shape looks good but you cannot get the frequency into correct range, there are suggestions in the Tips section, below.

Its best to calibrate the internal clock and VCO triangle and sine shapes first, since these trimmers are on the back of the panel. The test points for these are clearly marked. Connect your scope's ground reference to the GND pin of the TRIG header in the EG section. For VCO2 shapes, first set offset and symmetry while observing the triangle output. Then, set purity and gain while observing the sine output. You will not get a pure sine wave, but you should be something close and it should be 10V P-P, with no offset.

Don't forget to check out the 4072 VCF core and ensure it calibrates. Start with it's on-board (Piher) trimmer centered. If you have difficulty getting init frequency and 1V/Oct trims for at least three octave range, work with the Piher trimmer.

Only work with the reverb wiring while power is off. The return section has such high gain that just touching the shield connector with your finger will destroy the LM301 in the return amp, if it is running.

The relevant checkout sections to run through in the ARP 2600 Service Manual…

https://drive.google.com/open?id=1TUjsTKBTVcuaf_m3qcXhh3aoX6WBkeX2

… are:

2.3.3 Preamplifier

2.4.3 Envelope Follower

2.5.3 Ring Modulator

2.6.3 VCO 1 (bypass the Freq Cal and Volts per Octave steps if using the Fuzzbass calibration)

2.7.3 VCO 2 (bypass the Freq Cal and Volts per Octave steps if using the Fuzzbass calibration)

2.8.3 VCO 3 (bypass the Freq Cal and Volts per Octave steps if using the Fuzzbass calibration)

2.9.3 Voltage Controlled Filter

Note1: If AC coupling has been added to the four inputs on the VCF audio mixer, the observation at step 6.1 will a range of wave shapes moving around the 0V line; there will be 0V, not 5V, DC offset.

Note2: Bypass the Freq Cal and Volts per Octave steps if using the Fuzzbass calibration.

2.10.3 ADSR Envelope Generator

2.11.3 AR Envelope Generator

2.12.3 Voltage Controlled Amplifier

2.13.3 Mixer and Pan

2.14.3 Reverberator

2.15.3 Noise Generator

2.16.3 Electronic Switch

2.17.3 Sample and Hold

2.18.3 Internal Clock

2.21.3 Voltage Processors

2.22.3 Power Amplifiers

Note: Whether or not you hear distortion during this check depends on which speakers you installed. Eight ohm speakers can be expected to distort.

Common Build Issues and Tips

If you run into a problem calibrating one of your VCOs, swap the cores around to see if the problem follows. Such problems can come from the core itself, or from the CV summing mixer in the VCOs, which are on the main board. The cores generate a sawtooth wave, whose frequency is set by the CV summing mixer, which is on the main board. Other waveform outputs are created from the sawtooth wave, and by wave shapers, which are also on the main board. All the inputs, controls and trimmers on each VCO are part of the CV summing section, except the PW control, which is part of the pulse wave shaper.

If you have difficulty getting the VCO into the correct base frequency, it may be the CV summing section (which is on the main board) or the complementary transistor pair (on the 4027-1). You can narrow it down by moving the 4027-1 to a different VCO to see if the problem moves with it. If it does, start by changing out the transistor pair (2N3904/2N3906) on the 4027-1. I have not yet found a way to match them outside of the VCO – it's pretty much the luck of the draw. Using randomly selected transistors, in most cases you get a good match. But every so often you don't.

If you have no output, it may be the VCO core. If you cannot trim the correct wave shapes, it may be the shaper sections, or the VCO core. The sawtooth wave output is taken directly from the VCO core, so if that looks wrong on the scope, there is probably a problem in the 4027-1 core.

When testing the Sample and Hold circuit, use it's CV out to drive the pitch of a VCO, and set the clock to a very low speed. This will allow you to hear whether the circuit is really holding, or drifting. Drifting or sagging is a common build problem with the S&H circuit. If you hear it drifting, the problem will most likely be one of the JFETs in the circuit being bad or damaged, and not pinching off completely. The memory element is the 22n capacitor. On the schematic you will see there are JFETs on both sides of it (a single and a dual JFET). If either of these is leaky between clock cycles, the capacitor will discharge and the CV will drift. The most likely one to be defective in a drift situation is the 2N4392.

Flip the board face up to perform all the rest of the checks and calibrations. <u>Be sure to only connect the reverb unit while power is off</u>. One little mistake here will destroy the LM301 in the return amp.

If your EG outputs don't rise to the full +10V, and you did not install a Gate Booster, and that is why.

Fuzzbass TTSH VCO Pitch Calibration Routine

Regardless of what pitch and tracking calibration routine you use, you are not aiming for perfect tracking. You want *acceptable* tracking. If everything tracked absolutely perfectly, this would not be an analog synth.

The point of this alternate calibration routine is to allow the player to set the VCO controls visually, play a concert C, and have a pitch reasonably close to concert C come out of the VCO or VCF. It does not preclude tuning up, but it gets you close to pitch quickly.

The ARP 2600 service manual describes a usable calibration routine. Below is an alternative method. This only varies from the standard routing in the values selected for base frequency, CVs used, and range of tracking.

<u>In all cases, ignore the frequency markings on the VCOs and VCF Coarse Frequency controls printed on the panel.</u> If you follow the standard frequency calibration routine for the VCOs, as stated in the ARP Service Manual, and then set the Course Frequency control under "100Hz" and apply 0.00V to the keyboard CV bus, the VCO output is nowhere near 100Hz. If you adjust the calibration to achieve 100Hz at this position and with 0V, and then move the slider up to "1kHz", the output is not 1kHz.

- *Initial frequency calibration* sets the operating point for the VCOs and VCF
- *1V per Octave tracking calibration* adjusts how the VCO and VCF pitch tracks the Keyboard CV across the scale.

You may find it necessary to adjust the frequency calibration while the 1V per octave tracking needs no adjustment. However, the reverse is never the case. Each adjustment to the 1V per octave calibration requires frequency calibration to be repeated. You will typically move through four or five repetitions of these adjustments, each time homing in on *acceptable* tracking.

You can perform these adjustments yourself if you have adequate measurement devices. It is also possible to make these adjustments by ear, if you have some type of minimal standard to work with, such as a tuning fork, a reference standard oscillator, or good ears. Modern synthesizers employing digitally controlled VCOs can be used as a reference, but be careful to set up your reference patch so there is no pitch, amplitude or pwm modulation, or any FX present. You want your reference tone to have no discernible beating of any type. Purer tones such as sine or triangle work best for adjustments by ear.

Frequency Calibration

Note: Frequency calibration and 1V/Octave calibration should always be performed at the same time. Each of these adjustments affects the other. You therefore have to alternate through both procedures, with multiple iterations of each, to approach your final result.

Frequency calibration adjusts the set point of the manual frequency control for the VCOs and VCF. The VCOs have a very wide frequency range from 0Hz all the way up to 90kHz(!). 1V/Octave calibration adjusts how each VCO and the VCF respond to pitch modulation CV via the KBD CV bus.

> **Note**: If you don't like the frequency set point used here, you can create your own set point.

The frequency calibration method from the ARP Service Manual is effective. But it results in added work every time the synthesizer must be tuned in pitch with other instruments. The standard frequency calibration method also limits the LFO functionality at the low end of their range. If you intend to play the instrument in a setting with other instruments or musicians, you will require a quick and easy way to get into concert pitch. The objective with this alternative frequency calibration routine is to place the VCOs and VCF into a pitch range that is useful, versatile and easy to work with. The procedure is adjusted to result in an instrument that is much easier to bring into standard tuning – should you want tunefulness.

Standards Used

Proper tuning involves standards and measurement. The two standards used were:

a. Note C1 (MIDI note 24) = 32.70Hz
b. 0.00VDC = note C1.

These standards are commonly used, <u>but they are not universally used</u>. You may alternatively use a tuning fork for a standard, or pick your own note value/frequency for 0.00VDC.

This procedure is written for VCO 1, and should be repeated for all VCOs and the VCF.

DC Offset

Frequency is measured at the pulse/square wave output of each VCO taken at the VCF output jack. There is +5V DC offset present on the VCO direct outputs, and this can confuse some frequency counters. You can remove the DC offset by running the signal through the VCF, assuming the VCF mixer is modified for AC coupling (commonly done), with the resonance zeroed and the cutoff frequency opened all the way. In this case take the measurement from the VCF output. You may alternately run the signal through a 1uF film capacitor, if you have one.

Step by step is provided below, but the basic idea is this:

Frequency calibration: with all modulation removed, and 0V applied to the keyboard CV, the note C1 is produced from the VCO.

1V per Octave calibration: with frequency calibrated, and 4V applied to the keyboard CV, the note C5 is produced.

You will need an accurate method of measuring the input CV, and the output audio frequency:

- A digital multi meter (DMM) accurate to at least 10mVDC
- A frequency counter. Many DMMs have this function. All modern oscilloscopes can count and report frequency. There are also dedicated devices for this.
- Miniature flat blade screwdriver
- MIDI keyboard

Procedure

1. Warm up all equipment (synthesizer, measurement devices and CV source) for 30 minutes.
2. If you're using a MIDI keyboard, configure your MIDI to CV device for 0.00V at MIDI note 24. For example, with the Kenton Pro Solo:
    a. Power up the Kenton in analyzer mode NT (note). Set the octave range on your keyboard so the lowest C key results in "24" on the Kenton display. Then reboot the Kenton into normal operating mode.
    b. Connect the pitch CV output of the Kenton to your DMM and measure the voltage when you press the lowest C key. Adjust the voltage on the Kenton if necessary. You should be able to get to the range -0.99V to 0.00V on the DMM. For better accuracy, make the voltage measurement while the pitch CV is also connected to the TTSH.
3. Remove any other patch cables
4. If VCO sync is installed, make sure it is not engaged.
5. Zero all CV input attenuators on VCO1.
6. Set VCO1 Coarse Frequency slider to minimum / left and Fine Frequency slider as closely as possible to its mid point
7. On VCF, set VCO1 input to maximum, all other inputs to minimum, coarse frequency to maximum, fine frequency to middle, and resonance to minimum
8. Connect VCF output to frequency counter.
9. Ensure pitch bend and modulation wheel are centered/zeroed
10. Calibrate VCO1 frequency using the trimmer via the front panel. Adjust the FREQ CAL trimmer to obtain a value in the range.
11. Repeat steps 18-19 for VCO2 and VC03

VCF

As above, however:

1. Remove all audio and CV inputs
2. Set all audio and CV attenuators to zero
3. Increase resonance to full
4. Measure frequency at VCF output jack
5. Adjust the FREQ CAL trimmer to obtain a value in the range 32.65-32.75Hz @0VDC

The 32.7Hz base set point for the VCF will place the sweet spot of the filter (with no other modulation applied) somewhere to the right of center on the coarse frequency control. This will bias the filter such that more headroom will be afforded to frequency modulation via the KBD CV bus, or the two modulation inputs. If this is not to your liking, you can instead use 130.8Hz as your base set point. This will shift the sweet spot on the coarse frequency control to the left.

1V per Octave Calibration

Changing the 1V per octave trimmer will change the frequency calibration set point. Every time you make an adjustment to the 1V per octave trimmer, you return and adjust the base frequency at the base note/voltage of C1/0.00v (i.e. Frequency Calibration). It is typical to run through multiple iterations of this until your ears tell you it is good enough. If you have a very fine frequency counter, it will never look perfect on there. You are trying to get the resulting frequency into an acceptable range. Let your ears be the final judge.

Procedure

1. Perform Frequency calibration, steps 1-10 (VCO1)
2. Play C5. This will send 4.00V to the KBD CV IN jack.
3. Adjust the 1V OCT trimmer to obtain 522-524hZ (ideal is 523.2hZ)
4. Play C1. This will send 0.00 Volts to the KBD CV OUT jack.
5. Adjust the FREQ CAL trimmer to obtain 32.70Hz
6. Repeat steps 1-5 as needed to set the tracking for VCO1.

    TIP: The 1V/OCT and Frequency Calibrate trimmers interact. When you change the 1V/OCT trimmer, a compensating adjustment will be required on the Frequency Calibrate trimmer. As you adjust the 1V/OCT trimmer for C5, the tendency will be to overshoot the Frequency Calibration point needed for C1. While adjusting the 1V/OCT trimmer at C5, <u>approach the value you need, but don’t close the gap completely</u>. Keep it a wee bit back. In this way, you can more quickly reach the optimal adjustment for both trimmers, and reduce the number of repetitions of adjustment.

7. Repeat the procedure for VCOs 2 and. Don’t forget to change which VCO is brought up on the filter. When you calibrate VCO3, use a patch cable to connect is pulse output to the VCF.

8. On the VCF, perform adjustments with the resonance control fully up, the Coarse Frequency control all the way down, and the Fine Frequency control centered, and all inputs to the VCF minimized. When performing 1V/octave adjustment here, you may not get more than three octaves of acceptable tracking. So aim for 1046.4Hz @ 3V CV. Your mileage may vary. WRT to the 4072 filter, the 1V/Oct trimmer is on the sub board, and accessible via the back. Generally you won't get tracking on the 4072 as accurate as on the 4012. Set points for these VCF cores on the Frequency trimmer are different, for the same result.

Listening Test

1. Baseline the VCO tunings (Coarse Frequency=minimum, Fine Frequency=center)
2. On the VCF, reduce resonance to 0, and bring the Coarse Frequency control all the way up.
3. Bring VCO1 up to full in the mixer of the VCF.
4. Raise the input of VCO2 on the VCF to 80%.
5. Play C1
6. Adjust the VCO2 Fine Frequency control to tune with VCO1, eliminate any beating.
7. Reduce the input of VCO2 on the VCF to 0, and increase the level of VCO3 to 80%
8. Adjust the VCO3 Fine Frequency control to tune with VCO1, eliminate any beating.
9. Raise the input of VCO2 on the VCF back up to 80%. Now you should hear all three VCO together, in tune, at C1.

Play up the keyboard. You should not hear any objectionable amount of dissonance for at least four octaves. You will hear beating as you go up, but it should not become fast, dissonant beating. If you do, go back and refine the 1V per octave tracking adjustments. If you can make this work across six octaves, you have done very well – this is difficult and / or impossible on some TTSH units.

Putting It All Together

<u>Only when you are completely sure all functions are working, should you install the panel.</u>

1. Make sure all the sub boards have their retaining screws installed. For the 4027-1 cores, use small nylon cable ties to secure them (not included in kit or BOM). Make sure the ratchets are on the back, not the front of the board (as shown). Later, you can remove these, but you cannot replace them without separating the panel from the main board.

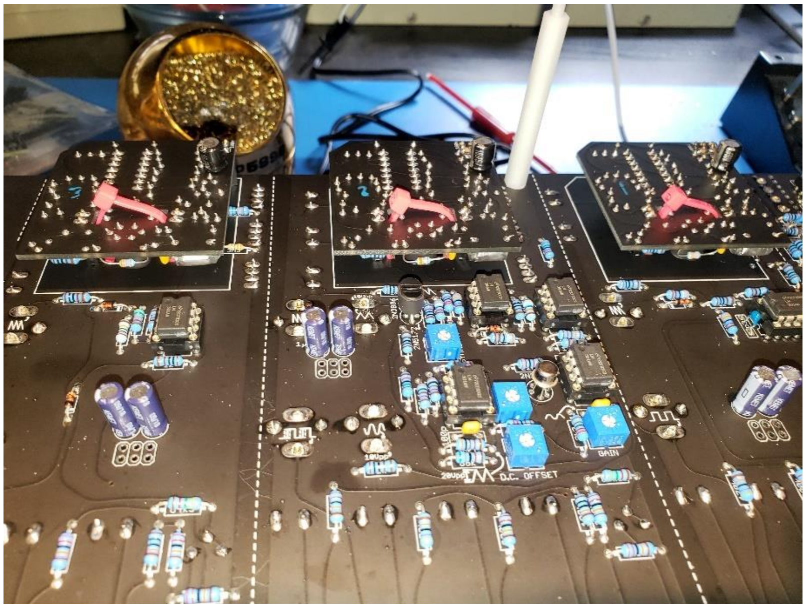

Figure 9 - 4027-1 Cores Secured with Cable Ties

2. Press the rocker power switch into the panel.
3. Fit the panel. Ensure that the power LED is in position. Install the nylon stilts on top of the panel.
4. Install the headphone jack to the panel and tighten with ½ inch nut driver.
5. Install the knob to the manual gate jack
6. Install the knob to the Preamp attenuator.
7. Install all 81 of the hex nuts.

   Tip: when tightening the hex nuts on the 3.5mm jacks (qty: 81), use a 5/16 inch nut driver. Cut a small square of lightweight cotton fabric (like a handkerchief or bandana) and place it between the nut and the driver. This will prevent the driver from marring or scratching the panel. <u>DO NOT OVERTIGHTEN!</u>

8. Flip the work over, face down, so it rests on the stilts.
9. Remove the four stilts from the back, and replace with hex nuts.
10. Desolder the +12V wire from the barrel connector and solder it to the center pin of the power switch.
11. Connect the lower pin of the power switch to the +12V pin on the barrel connector.
12. Turn the assembly over, face up. Remove the four stilts and install the M3 hex key nuts (13 total). Be careful not to overtighten these.
13. Install the barrel connector to the case and tighten.
14. Install the reverb unit in the case and connect the wires to it. Red = return. Position this jack on the reverb unit nearer the reverb jacks on the main board.
15. Connect the reverb unit to the main PCB
16. Drop the work into the case and install the four corner hex screws to secure.

Modification: Add Power Taps for Future Mods

Consider adding a few locking headers to the various 2x3 pin header pads located around the synth. These can be used down the road to power other modifications, such a VCO sync, MIDI Devices, of the Shapers option.

Modification: AC Coupling on the VCF Audio Mixer Inputs

This modification is commonly applied by most builders, and has been applied to large numbers of ARP 2600 units in the field. Four of the audio inputs for the VCF audio mixer have film capacitors placed in-line with their inputs, to block DC offset in the signals coming from the VCOs. The Sawtooth and Pulse wave outputs have a +5V DC offset. This can lead to some choppy or thumpy behavior in the VCA, so many builders apply this fix. The first audio input (normally the Ring Modulator) is left unmodified. The change does result in some waveform distortion that is apparent on an oscilloscope, but does not have a dramatic effect on the perceived sound.

Parts to perform this change are included in the Synthcube kit and BOM; 4x 1uf film capacitors. The locations for these are marked in silkscreening on the board, and traces must be cut underneath the capacitors footprints before the capacitors are installed. The location for these trace cuts are also silkscreened.

Modification: Gate Booster Installation

The purpose of the Gate Booster is twofold; 1. Drive the gate inputs on both the AR and ADSR; 2. Generate the trigger impulse required by the ADSR.

The 2600/TTSH EG's gate inputs are not gate detectors, they are gain stages. If you don't drive them with a full +10V gate signal, the EGs will never rise to a full +10V on their outputs, leading to reduced signal gain from the VCA outward. Many devices in the modern studio will supply +5V gate signals, which will not fully drive the EGs. The Gate Booster uses a comparator on its input to detect gate signals of varying amplitude (threshold is +2.7V), and then generates a consistent +10V gate on its output. It then uses its own generated gate to produce a +10V trigger impulse for the ADSR. The trigger represents a momentary impulse at the leading edge of a detected gate.

> Note 1: if you drive your TTSH using MIDI-to-CV converter, program it for multiple trigger mode. This function will allow you to overlap notes on a MIDI keyboard and the ADSR will reset and make a new note event for each new overlapping note. This mimics the behavior of an ARP 3600 series keyboard. You do not need to connect a trigger CV from the converter, just gate. The Booster takes care of the rest.

> Note 2: ARP user manuals refer to the Gate and Trigger jacks as outputs. This installation converts the function of the Gate jack to *an input*. The Trigger jack still functions as an output for the Booster's generated trigger signal. There is no external output for the Booster's generated gate signal.

For installation, make sure you have cut the trace running from the tip of the Gate jack to pin 1 of the Gate Switch (see Figure 1). Then, refer to the photograph. The Booster is installed in the span between the tip of the Gate jack and pin 1 of the Gate switch, where the trace was previously cut (the cut is obscured in Figure 10). In the photo, the polarity of the power connector is indicated, and power jumper connectors cross over (here Red = +15V, Green = 0V and Black = -15V). If you install the locking header on the main PCB the other way around, your power jumper should not cross over.

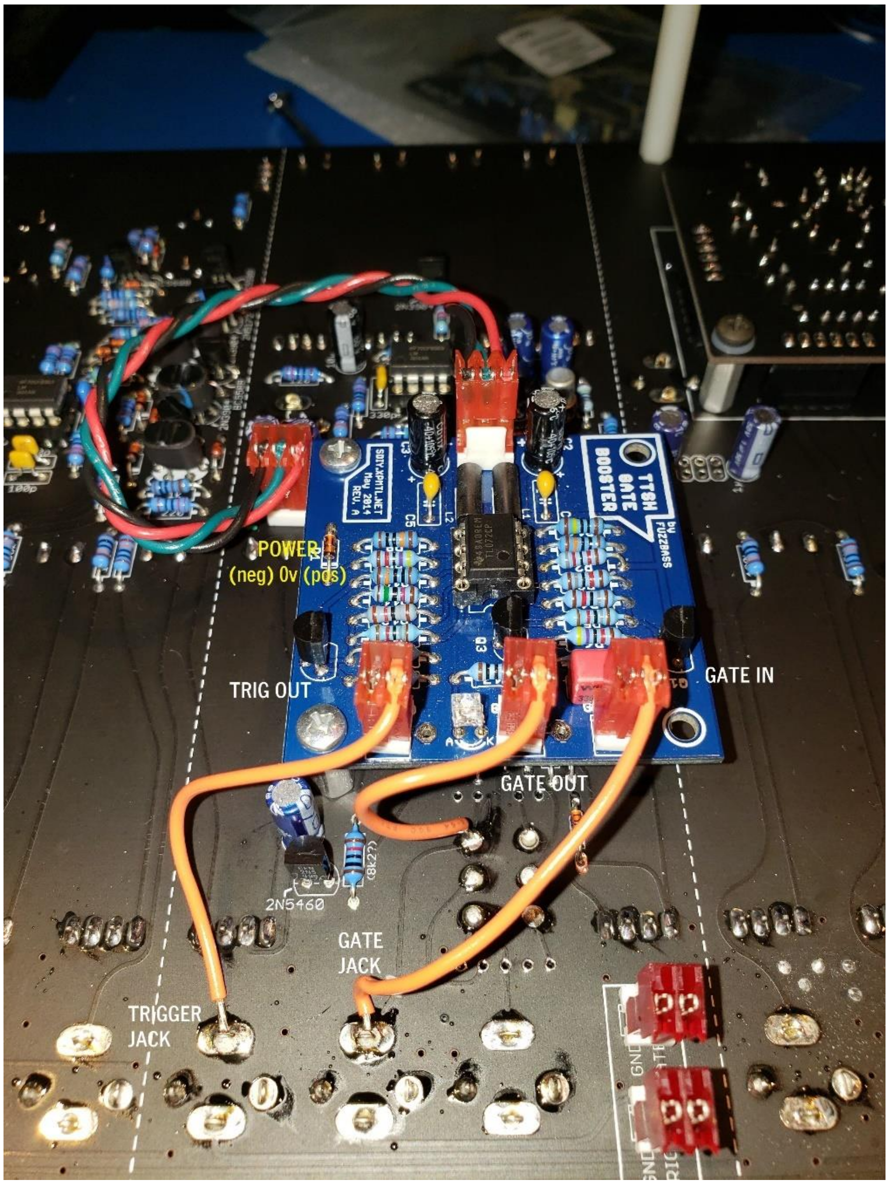

Figure 10 - Standard Gate Booster Installation. On the power jumper here, red=V+ and black = V-

Signal Connections:

- Gate In > Gate Jack (tip)
- Gate Out > Gate Switch (Pin 1)

- Trigger Out > Trigger Jack (tip)

Modification: External Control – Electronic Switch

If you install only one mod, it should be this one. It is utterly simple, reversible, requires no parts, and provides sub-oscillator capability. This re-routes the control input of the electronic switch from the output of the clock to the tip of the External Clock In jack. When nothing is connected to that jack, the Electronic switch control input is still routed to the clock. When an alternate signal is applied to the External Clock In jack, the Electronic Switch will be controlled by it. To do this, find the two resistors, desolder and lift the lower ends of each, and tie these in common to the tip of the External Clock In jack.

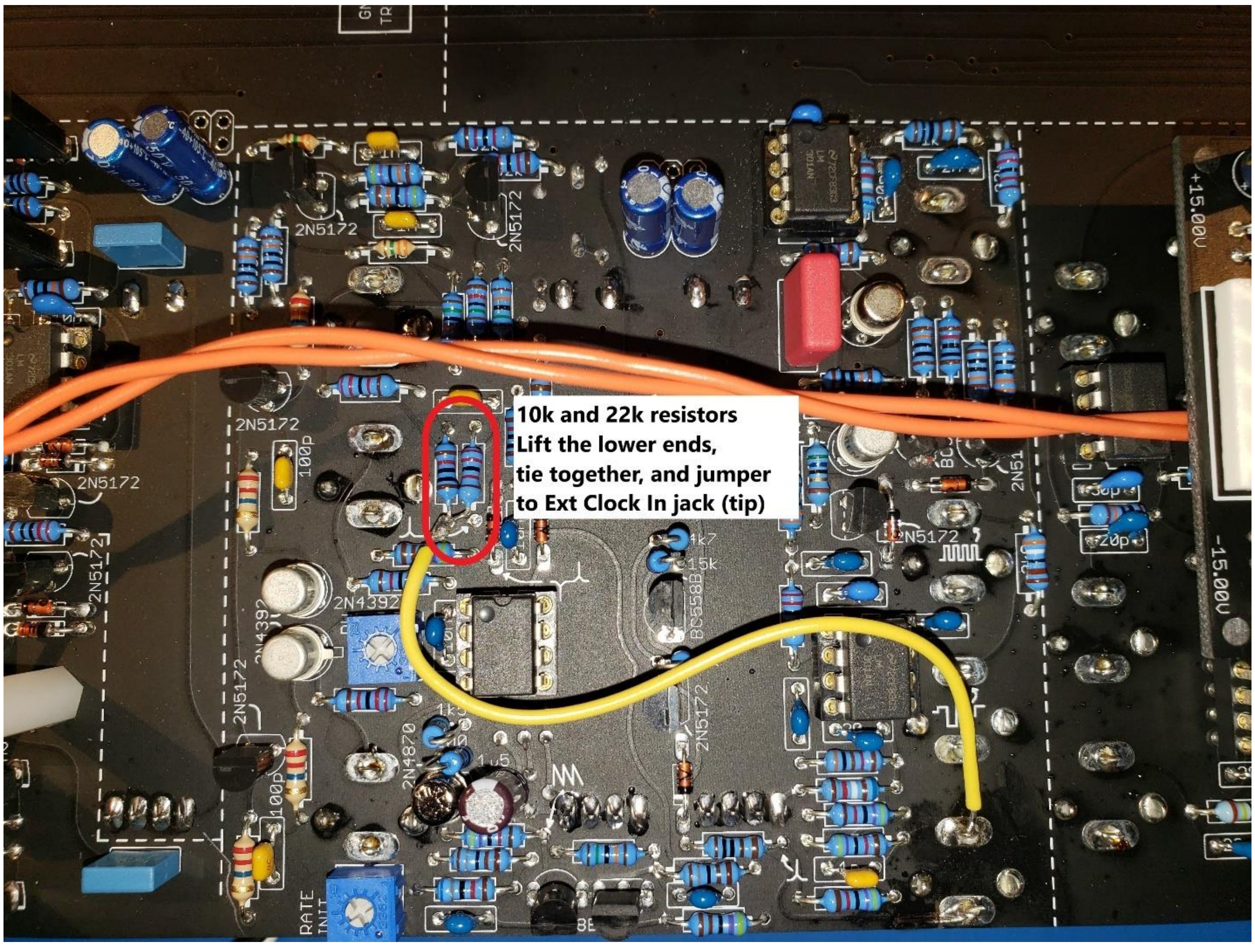

Figure 11 - Electronic Switch Mod

If you then patch up the Electronic Switch thus:

- A > Output first Voltage Processor (top slider in the middle i.e. +5V)
- B > Output second Voltage Processor (it's slider in the middle i.e. -5V)
- External Clock In > VCO1 square wave

… a -1 octave sub-oscillator, tracking VCO1, will be available at Jack C on the Electronic Switch. If you process this with just a wee little bit of Lag, it will be converted into a near-triangle wave which can be mixed in post-VCF in the VCA's second audio input, for Moog Taurus I type sounds.

Modification: Adding a MIDI-Implant

The author has no experience with the MIDI-Implant device, aside from answering questions related to a variant of the Gate Booster that acts as a pass through for the MIDI-Implant. So, this section will not describe the installation or configuration of a MIDI-Implant, but rather, discuss the connection of CVs generated by the Implant. The main complexity arises due to the sort of botched design of the front panel KBD CV jack.

The TIP-Switched contact of the KBD CV (the panel pitch CV input) jack is connected to 0V in the middle layer of the main board. This means you cannot cut a trace to defeat this. This is fine for situations where the TTSH is only being controlled externally. However, the arrangement is sub-optimal for internal MIDI to CV devices (or an actual 3600 keyboard controller). The problem was introduced in TTSH V2, and sprang from the best intentions. The idea is that when no pitch CV source is connected, the VCOs and Filter pitch control will be anchored at 0V. Without that, the VCOs will motorboat. This change however, renders the CV header on the back of the main board useless, unless a dummy plug is inserted into the front panel jack.

A better arrangement would have been to connect the TIP-Switched pin of the of the KBC CV jack to the CV pin of the header, and then instruct builders to install a shorting jumper on that header, if only using external CV sources.

Oh well…

To connect a pitch CV that is generated inside the case, such as a with a MIDI to CV device (or wiring up a 3600 keyboard inside the case), do the following:

1. Ignore the CV header on the back of the main board.
2. Before installing the KBD CV jack, bend its TIP-Switched contact out and away from the board. Make sure the pin is not in contact with the pad or the metal shroud around the jack. A wee bit of heat shrink will help here.
3. Connect the pitch CV from the MIDI device to the TIP-Switched contact on the KBD CV jack.
4. Make sure your MIDI device is always powered up when the TTSH is. It will now be the default pitch CV source, but defeatable via the front panel KBD CV jack, or individual VCO/VCF CV jacks.
5. Gate Output from your MIDI device should be connected to the TIP-Switched contact of the Gate Jack. The MIDI device will be the gate source when the Gate switch is in the up position, and will be defeatable via the Gate jack. Both the MIDI and External gate signals will be processed by the Gate Booster, if present. Don't use the MIDI device to generate a trigger CV. Instead, let the Gate Booster do it.
6. Configure your MIDI device for multiple trigger mode, to make best use of the ADSR.

Modification: Improving Calibration Accuracy

Bourns makes four-turn trimmers that fit the existing trimmer footprints. They are costly, about $6US at the time of this writing. These are not included in the Synthcube kit or BOM. You may install these in the VCO trimmer locations for Frequency and 1V/Octave calibration, to improve precision. Keep in mind that these merely introduce a gear between the adjustment screw and the resistive element. The resistive element is the same. You don't really get four times the accuracy, but it is still a significant improvement. The parts are:

- Bourns 3339P-1-253LF (qty:3)
- Bourns 3339P-1-104LF (qty:3)

Modification: Increase Adjustment Range: LEDs

The standard adjustment range for the LED trimmer does not allow for extinguishing the LEDs. The range may be expanded, without affecting overall brightness, to allow adjustment all they way to off. Replace LED-R1 (3k3) in the main LED driver section with a wire jumper.

Modification: Reverb Improvements

Improvements to the reverb system fall into two general categories: changing the character of the spring unit, and improving the overall noise rejection.

Substitution

The TTSH reverb driver and return amplifier are compatible with any Accutronics (or clone) model whose second and third digit in the model number are "AB". The Synthcube kit and BOM specify a 1AB2A1B. This is a two spring, medium decay unit in a short pan. The fifth digit "A" specifies a grounding configuration where both input and output have a common ground with the pan. This ground configuration is required for operation in the TTSH.

You may substitute any reverb unit so long as the second and third digit in the model are AB (input impedance ~10 ohms, and output impedance ~2K ohms). Observe, however:

> If the fifth digit (grounding) of the model number is not "A", the builder will need to modify the ground configuration of the pan such that the input and output sleeve connectors are common with the pan itself. This is usually just a matter of installing a wire jumper between the sleeve connectors on the jacks.

Noise Reduction

The signal level in the return from the reverb unit is extremely weak, and very close to the noise floor. Noise is induced in the sleeve of the return cable, and because the connection is not balanced, it can get added to the reverb signal. This is specifically a problem when the 0V rail is referenced to an external brick. Most bricks have a floating ground, and do not provide a shunt path for signals induced in the shields of audio connectors. The shield needing this shunt path the most is the one in the cable for the reverb return. The gain of the return amplifier is very high and so the noise will be brought up to audio level. There are two ways to improve this (in order of effectiveness): 1. Connect the 0V plane of the synthesizer to a true Earth ground; 2. Improve the rejection of noise in the return cable.

Earth Ground

Although not discussed in this guide, some builders substitute a linear bipolar power supply for the external brick and DC-DC converters. In this arrangement, there is a strong earth ground brought to the 0V reference in the TTSH. This approach yields inherently quieter reverb, but at the cost of introducing 50/60hz line noise to the reverb signal. When the TTSH is built using the DC-DC converters and an external brick power supply, the reverb can be particularly noisy. The solution is to bond 0V on the synth to true earth. This may be as simple as placing a patch cable from any jack on the synth to another system whose 0V plane is referenced to earth. Another approach, recommend here, is to add a grounding lug connection to the case.

It is left to the builder to determine the best way to bond the TTSH to Earth in the studio, so no particular solution is advocated here. In a typical studio, there are many places and options to tie the TTSH 0V plane to Earth.

In the author's studio, a banana receptacle (green, of course!) has been added to the case, and this is wired inside the case to the 0V pin on the barrel connector accepting the input power from the brick. On standard USA appliance outlets, the center screw on the wall plate functions as an Earth connection. A ground wire was constructed with a banana plug on one end, and a C type spade connector on the other. This cable runs between the screw on the wall plate to the banana plug on the synth.

Regardless of how the synth is bonded to Earth, if it is powered using the external brick + DC-DC converters, there will be a dramatic reduction of reverb noise when the Earth connection is established.

Improved Noise Rejection in the Wiring

When routing the reverb cable inside the case, keep in mind the biggest sources of induced noise inside the case, and try to dress the cable away from them. In order of noisiness: DC-DC converters, Noise Generator, Internal Clock, Ring Modulator.

The Sythcube kit includes a standard duplex audio cable assembly (a tape deck cable, in fact) for connecting the reverb. The kit also includes parts the builder may use to build cables for improved noise rejection. These parts consist of a length of RG174 coaxial cable, and four basic RCA plugs. This method reduces noise two ways: 1. RG174 coax is antenna grade wire specifically engineered for better screening, and improved isolation between the shield and the signal wire. 2. The connector cables (in particular, the return wire) can be cut to minimum length necessary – thus reducing the exposure to noise sources.

This is how reverb was wired up in old Fender guitar amps. The following steps assume the use of the standard steel case.

1. Install the reverb unit such that the return (red) plug on the pan is oriented closest the reverb jacks on the TTSH.
2. Cut two lengths of RG174: 14 inches and 20 inches (24 inches for type 4 or 9 reverb).
3. Strip the coax such that the inner copper core is exposed to the length of 1 inch, and the shield is cut back and exposed to the length of ¼ inch. See the photo:

Figure 12 - Cut and Strip the RG174 as shown

4. Twist the copper conductor, and flare out the shield wiring.

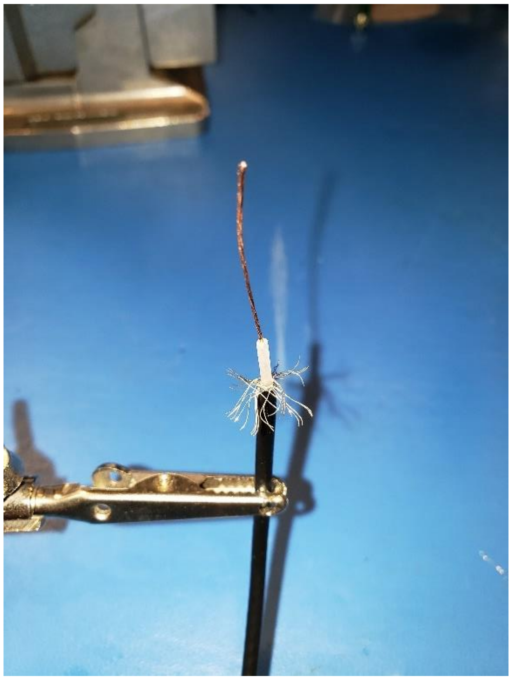

Figure 13 – Preparing the cable.

5. Insert the copper conductor and its insulated portion into the plug, such that the flared out shield meets the outer part of the plug, and the copper conductor comes out the hole in the tip of the plug. Clamp it and support the cable going in, so it does not sag.

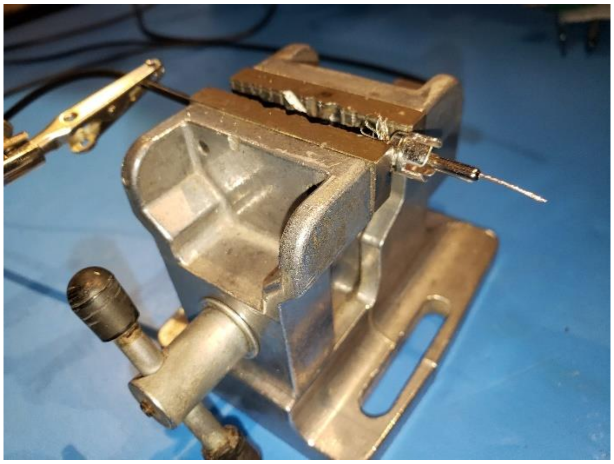

Figure 14 - Ready for Soldering the Tip.

6. Heat your iron to 625f, melt a bead of solder onto the iron's tip. Bring the iron to the tip of the plug, just beneath the opening and the copper wire. At same time bring solder to the opening. As the work heats up and the solder melts, it will be wicked into the inside of the plug. Apply a healthy amount of solder.

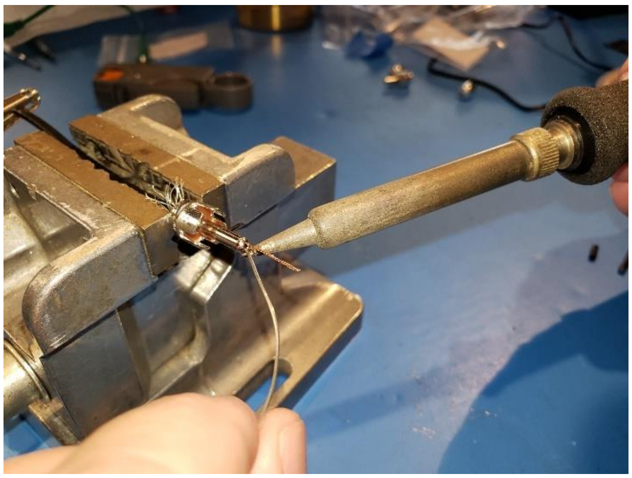

Figure 15 - Soldering the Signal Wire

7. Snip off the signal wire protruding from the tip of the plug.
8. Push down the shield wire onto the outer part of the plug, clamp the work and solder the shield to the outer plug.

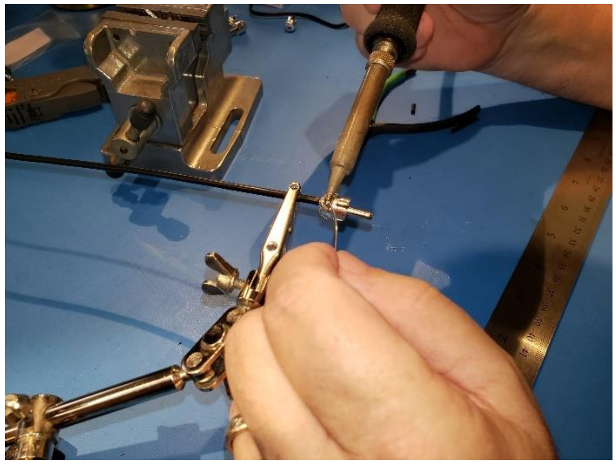

Figure 16 - Soldering the Shield

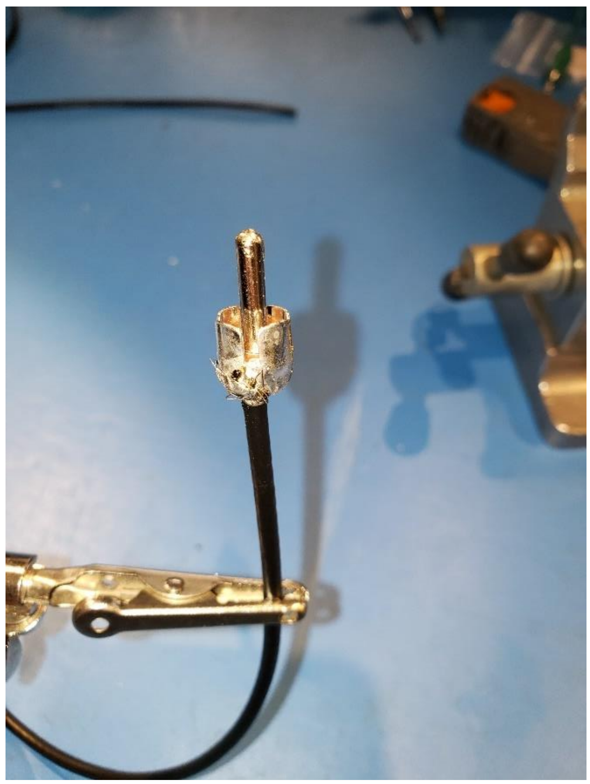

Figure 17 - A Finished Plug

9. Repeat three more times. Then, test the cable assemblies for shorts and continuity.
10. If there is excessive solder on the exterior of the tip, clamp the plug and scrape it off with a knife. Don't heat the work again!
11. It's a good idea to mark the shorter return wire in some way with red. Use either heat shrink tubing or paint. Mark both ends. I use enamel model paint.

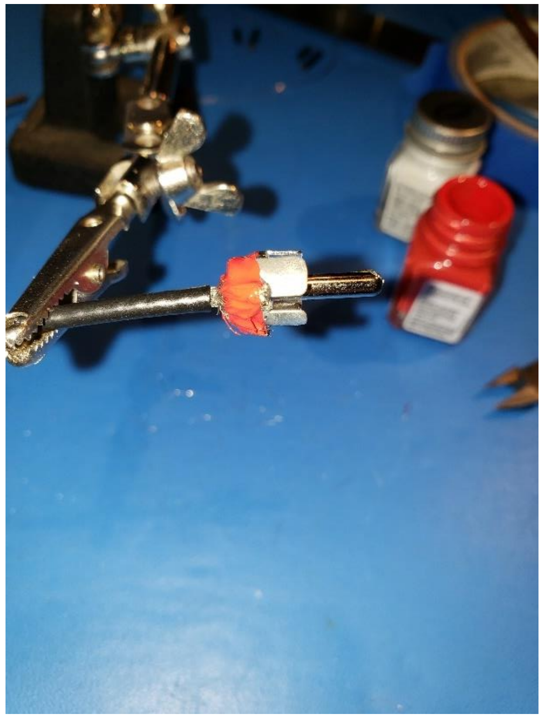

Figure 18 - Mark to ID the Short Return Cable