RE-606 v1.3 BUILD GUIDE

Here are the general build notes for the RE-606, before starting please visit https://shop.re-303.com/build-it-2/ and download the BOM, we've made a few changes here and there so you should grab it before you start.

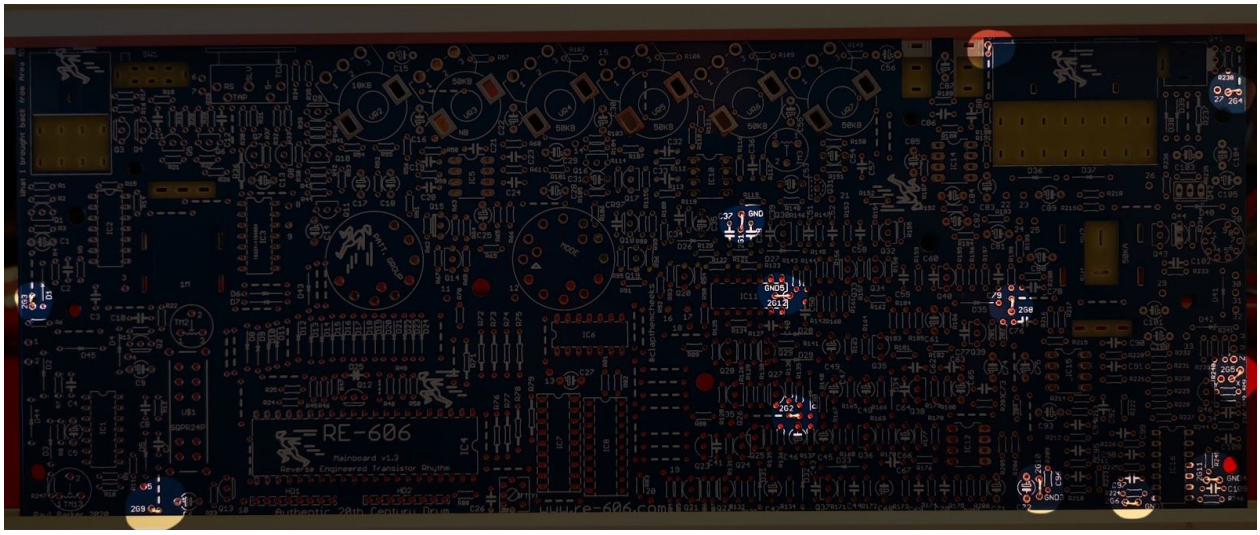
For placement you can refer to the TR-606 Service notes

There's a couple of things to attend to before building so check all of this short document before you start.

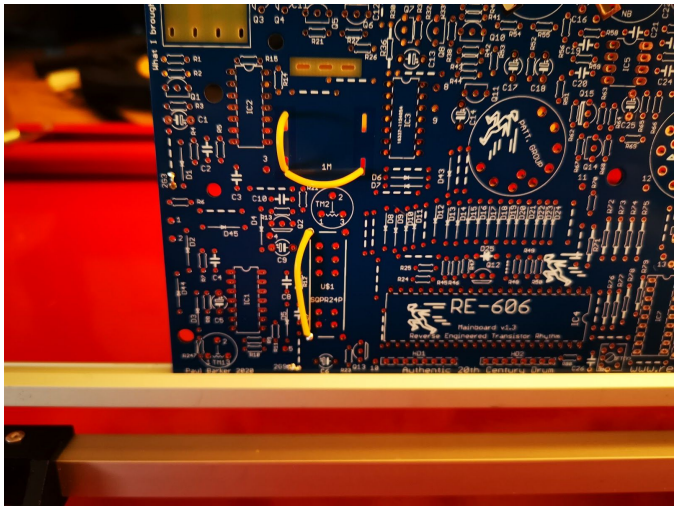
What's in the bundle?

1x pcb set (3pcs) mainboard, switch board and tom board.
1x pot set, 1x tempo,1x vol, 1x control 10kb, 5x control 50kb
1x rotary set (2pcs)
1x sumida coil and adaptor
1x jack set (6pcs)
1x sync switch
1x power transistor set (3pcs)

Fit the RF jumpers

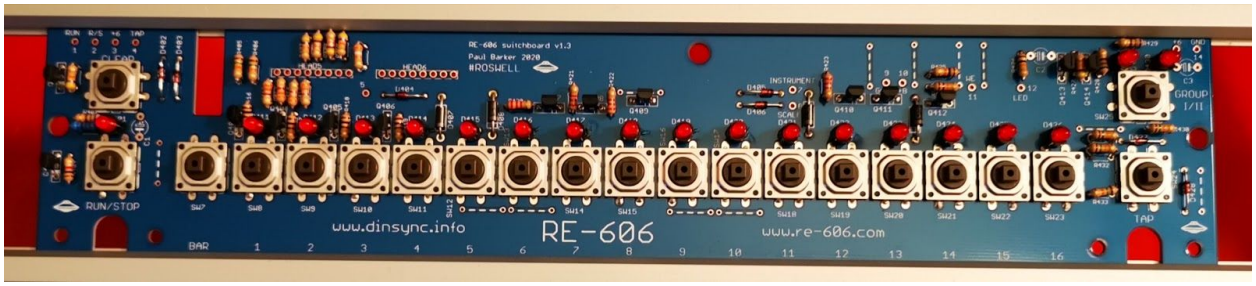


The regular wire jumpers do not need to be fitted as they have been fixed in the copper layer, however you still need to manually connect 12 points for the RF layer. See the image above for the 12 points and how they should be connected (zoom in for details).



If you don’t have pot brackets you’ll want to connect these temporary points by the tempo pot. Also if you don’t yet have a scale switch then place a temporary patch between the top and bottom left of the switch chassis footprint.

Switchboard



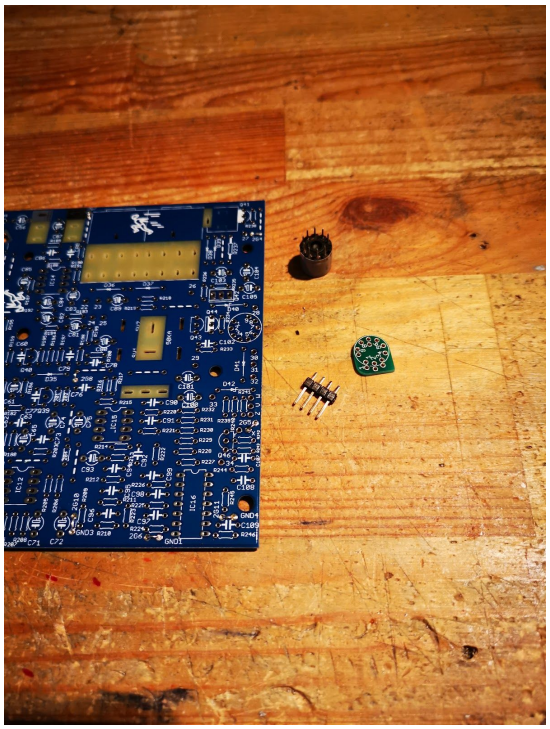
My resistors are too large (but sufficient for the picture, also the electrolytic caps aren't placed yet)

Build the switchboard, all of the part numbers are in the 400 range, so quite easy to find in the BOM. However the three electrolytic capacitors were labelled wrong, these are as follows

C401 is C1
C402 is C2
C403 C3

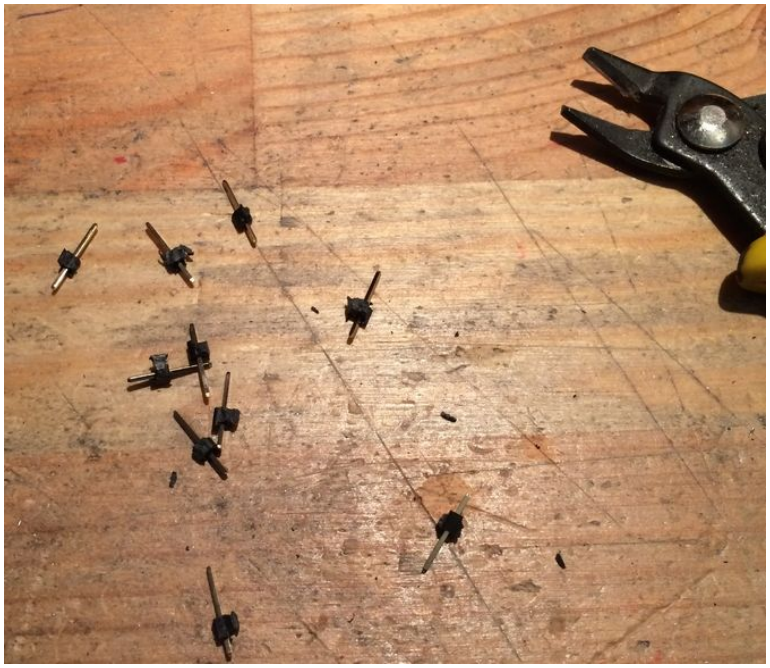
Power section

CARE MUST BE TAKEN WITH THE COIL

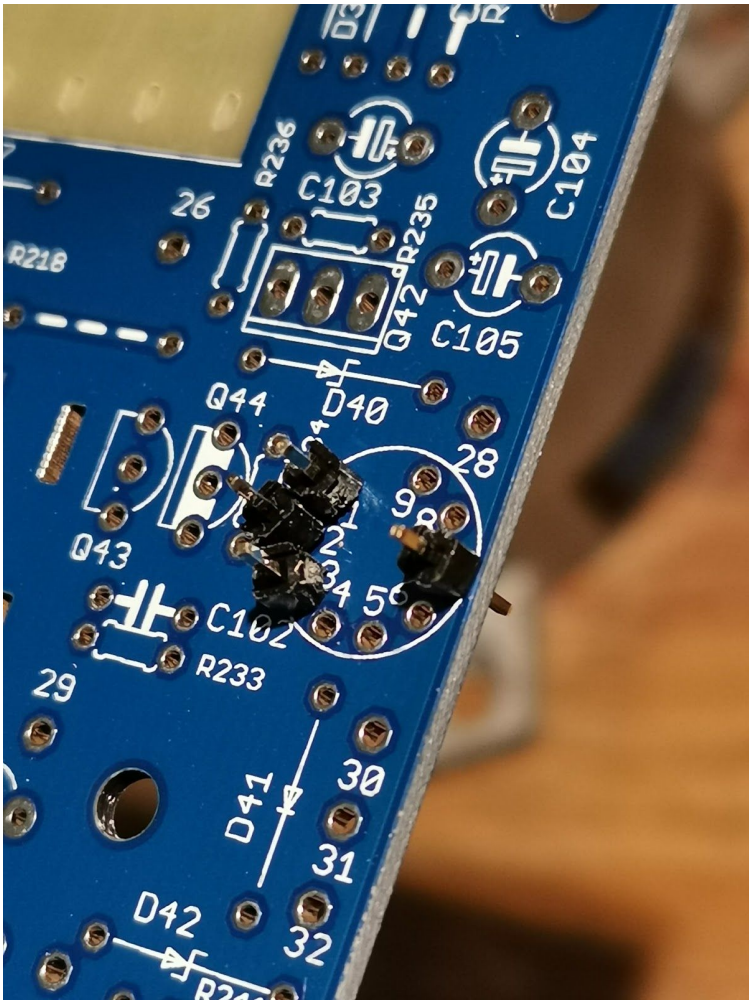


The coil is a 30 year old "new old stock" part, so care should be taken when soldering, the pins themselves will not take much to break or overheat.

Before we attach the coil, take the header



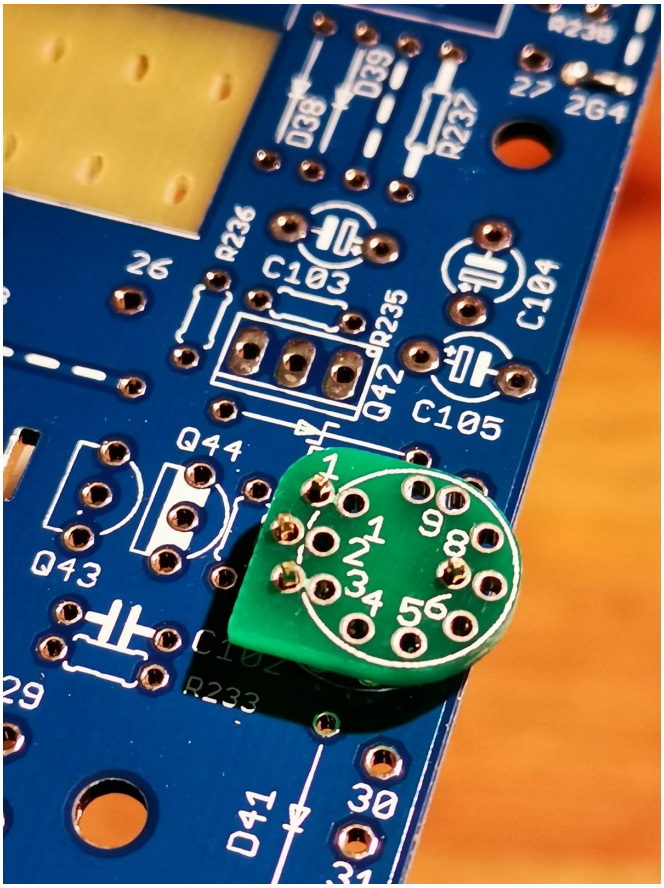
And cut it up into single legs.



Now place the 4 pins with the long part into the hole in pin locations 1,2,3 and 7 as shown in the image.

DO NOT SOLDER ANYTHING TO THE MAIN BOARD YET!

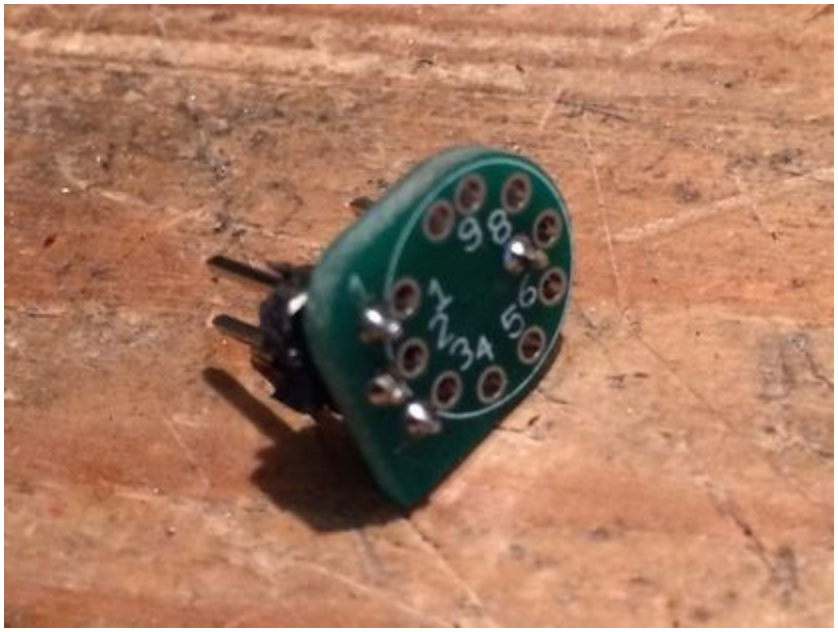
(we are just using it to hold the pins for us)



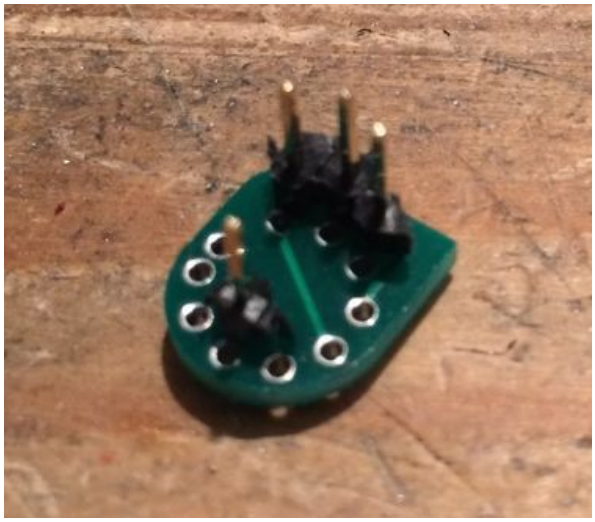
now place the adaptor on top of the pins as shown. Check that the orientation is correct and that there is a clear place for the coil to sit.

If everything looks good, solder the adaptor to the pins.

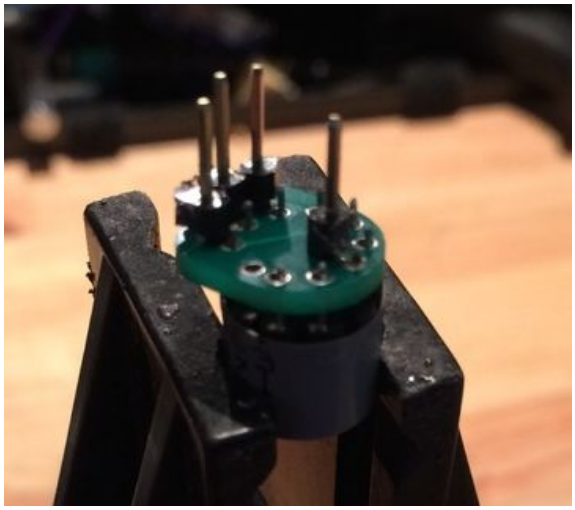
DO NOT SOLDER ANYTHING TO THE MAIN BOARD YET



remove the adaptor and check your work

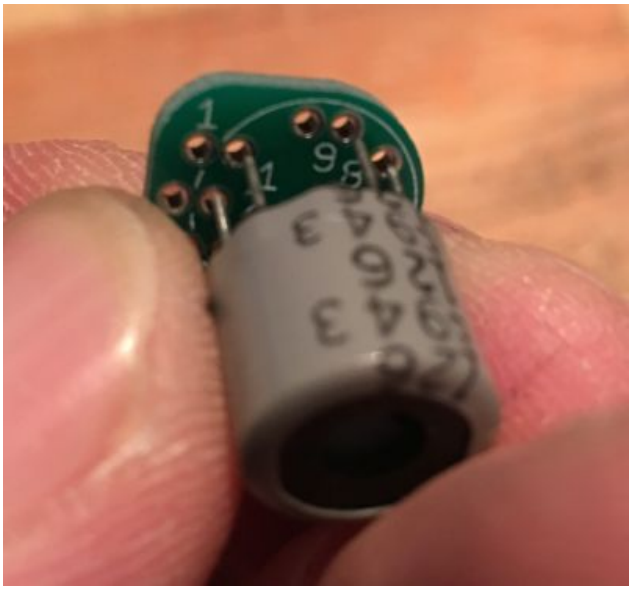


here's another angle

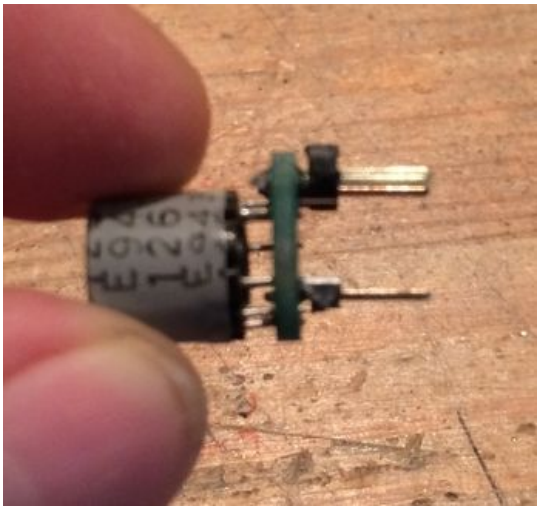


now place the adaptor on the coil as shown, **ONLY SOLDER ONE LEG** to the adaptor

BE SURE TO ALIGN PIN1 OF THE ADAPTOR (first pin in the gap) TO PIN 1 ON THE ADAPTOR (**do not align the gap from the pin9 side**, as there is an extra support pin next to pin 9 we don't use. It won't fit directly this way but can be accidentally forced to fit.)



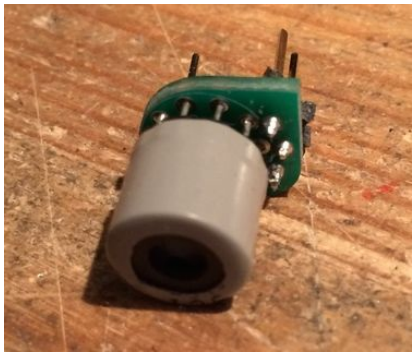
Note the empty hole between pin 1 and pin 9.



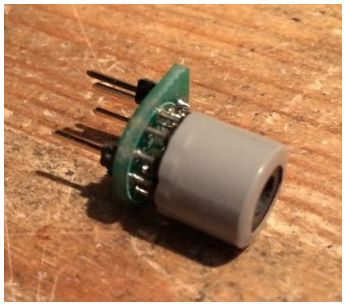
now carefully heat that soldered pin and adjust so that the coil is straight and does not touch the adaptor anywhere and also that the legs of the coil do not protrude as to be able to touch the main board.

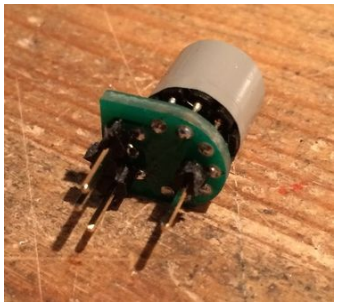
DO NOT BEND IT STRAIGHT AS YOU MAY BREAK ONE OF THE PINS

if it looks like the picture then solder the remaining pins.

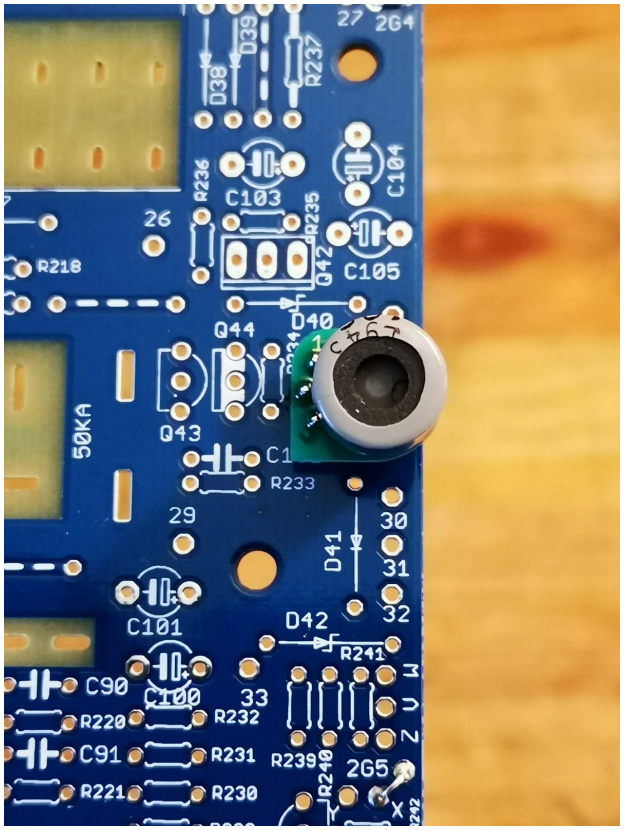


Inspect your work,

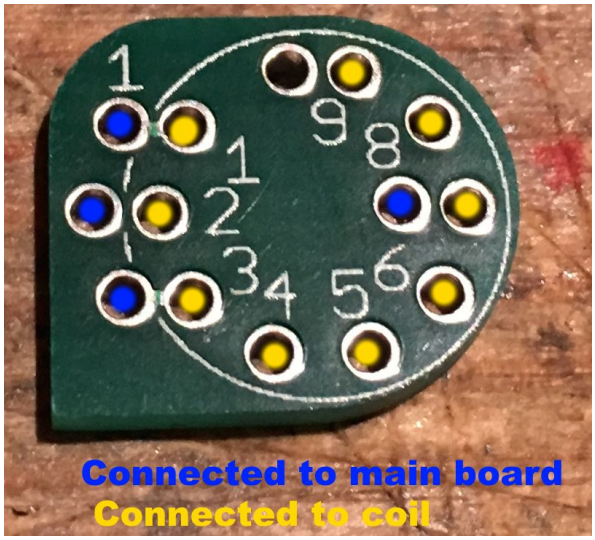




Place and solder the coil and adaptor on the main board.



The orientation of the pointer on the adaptor must be bottom left corner!



Adaptor reference

Power Section

Now that you have placed the sumida coil you can place the rest of the power section parts.

Resistors

WIRE JUMPER x 1, R237
33R x 1, R233
3.3K x 1, R234
1.5K x 2, R235, R238
15K x 1, R241
27K x 2, R239, R242
82K x 1, R240
100K x 1, R236
180K x 1, R243

Capacitors

2.2n / 16v x1, C102
10uf / 16v x 4, C9, C101, C103, C105
47uf / 16v x 1, C100
100uf / 16v x 1, C104

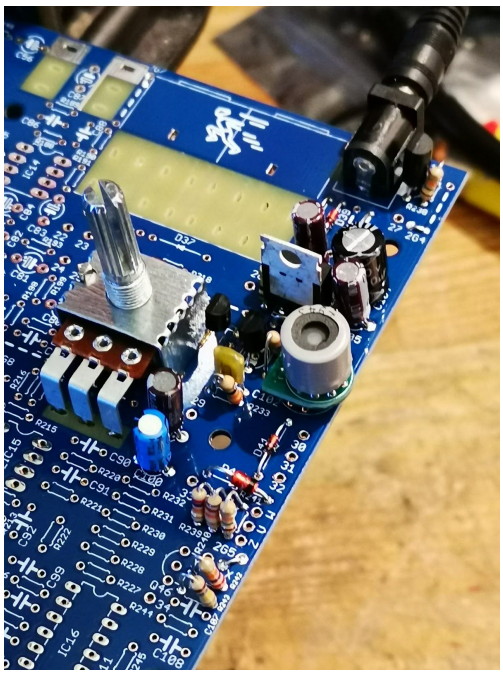
Transistors

KSA916Y x 1, Q41
TIP30C x 1, Q42
2SC945P/2SC536F/2SC603 x 1 Q43 (do not solder yet)
KSD1616A x 1, Q44

Diodes

1SS133/1N4148 x 5, D38, D39, D44, D45, D41
TZX15C x 1, D42
TZX6V8C x 1, D40

Power test



Place DC jack.

Place power switch/vol pot (metal bracket isn't really needed on this side if you only have one)

Place Q43 so that it makes contact but do not solder it in.

Connect your DC adaptor (9v center pole negative, 500ma)

Turn on power, measure these points.

Point 32 gnd
Point 28 6v
Point 33 15v

If all voltages are good turn off power and solder Q43.
If you get something substantially different try another transistor at Q43, proceed to solder it in when voltages are good.

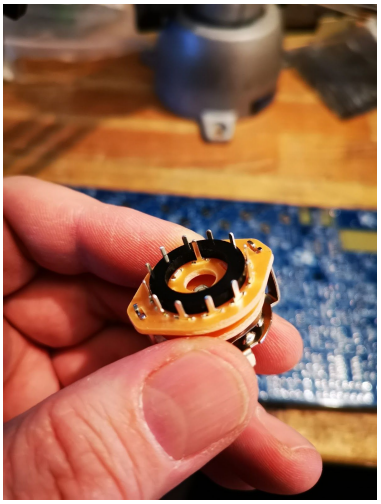
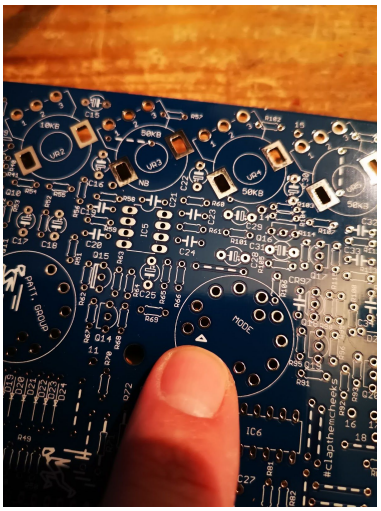
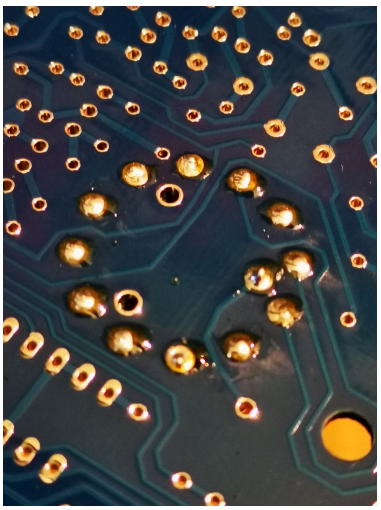
Next Place these diodes and resistor **(before patt / group rotary!)**

D1, D2, D3, D4, D5, D6, D7, D8, D9, D10, D11, D12, D13, D14, D15, D16, D17, D18, D19, D20, D21, D22, D23, D24, D25

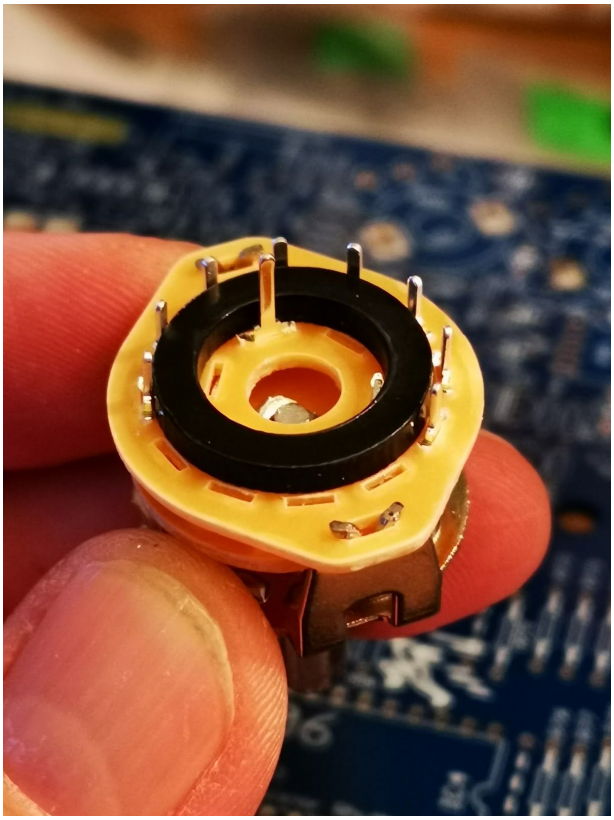
D43

Place R106

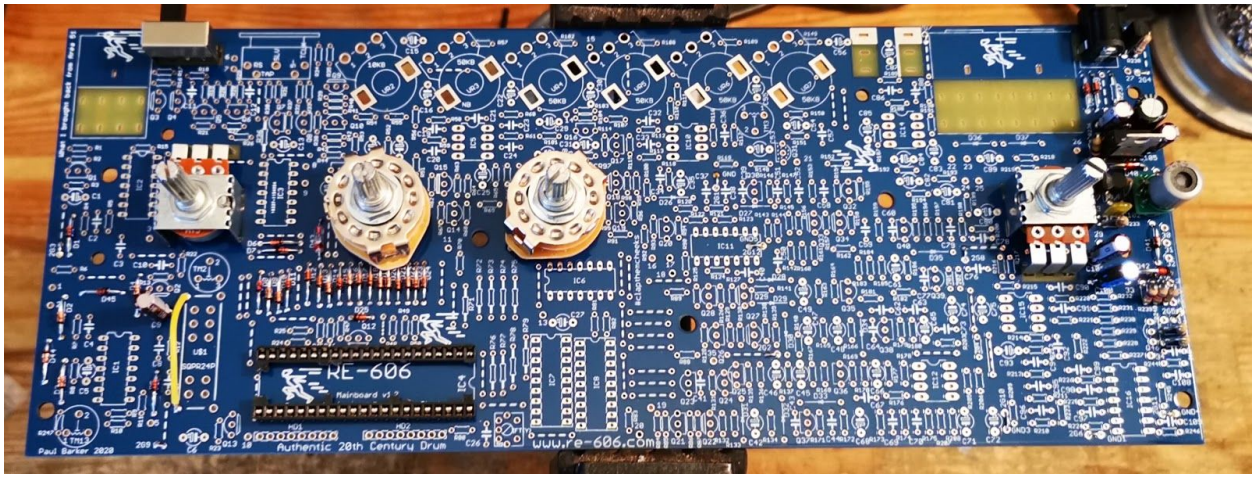
It’s recommended to print Luan's 3D spacers Of course you don’t need them but it will be a great help when soldering in the rotaries and especially the instrument select as it doesn’t have a full rotation of pins.

	Place the spacer on the underside of the 4 position rotary switch.
	Place the part and be sure to line up the switch pin to the hole marked by the small arrow.
	After soldering the underside should look like this.

You can also place the instrument select rotary now also.



As before, if you have a spacer place it now, you'll see now since there isn't a full rotation of pins why the spacer is helpful. You could use another non conductive material if you have something at hand, the spacer is 2.4mm tall.



Fit the sync switch now also. Here's a shot of the pcb so far, I added a socket for the CPU which we will be fitting at the end of the digital section.

Building the digital section

Fit these parts

Resistors

22R x 1, R36
100R x 1, R53
1K x 3, R2, R18, R35
3.3K x 1, R247
4.7K x 3, R39, R43, R44
10K x 3, R11, R23, R28
15K x 5, R21, R26, R37, R38, R80
22K x 4, R16, R17, R29, R40
33K x 6, R3, R19, R27, R32, R42, R71
47K x 7, R1, R12, R13, R14, R15, R22, R41
100K x 3, R20, R30, R33
150K x 2, R9, R10
470K x 3, R4, R5, R31
1M x 3, R7, R8, R34

Capacitors

Poly
0.068 x 1, C7
0.039 x 1, C8
0.001 x 1, C10
0.01 x 3, C4, C11, C12

Electro

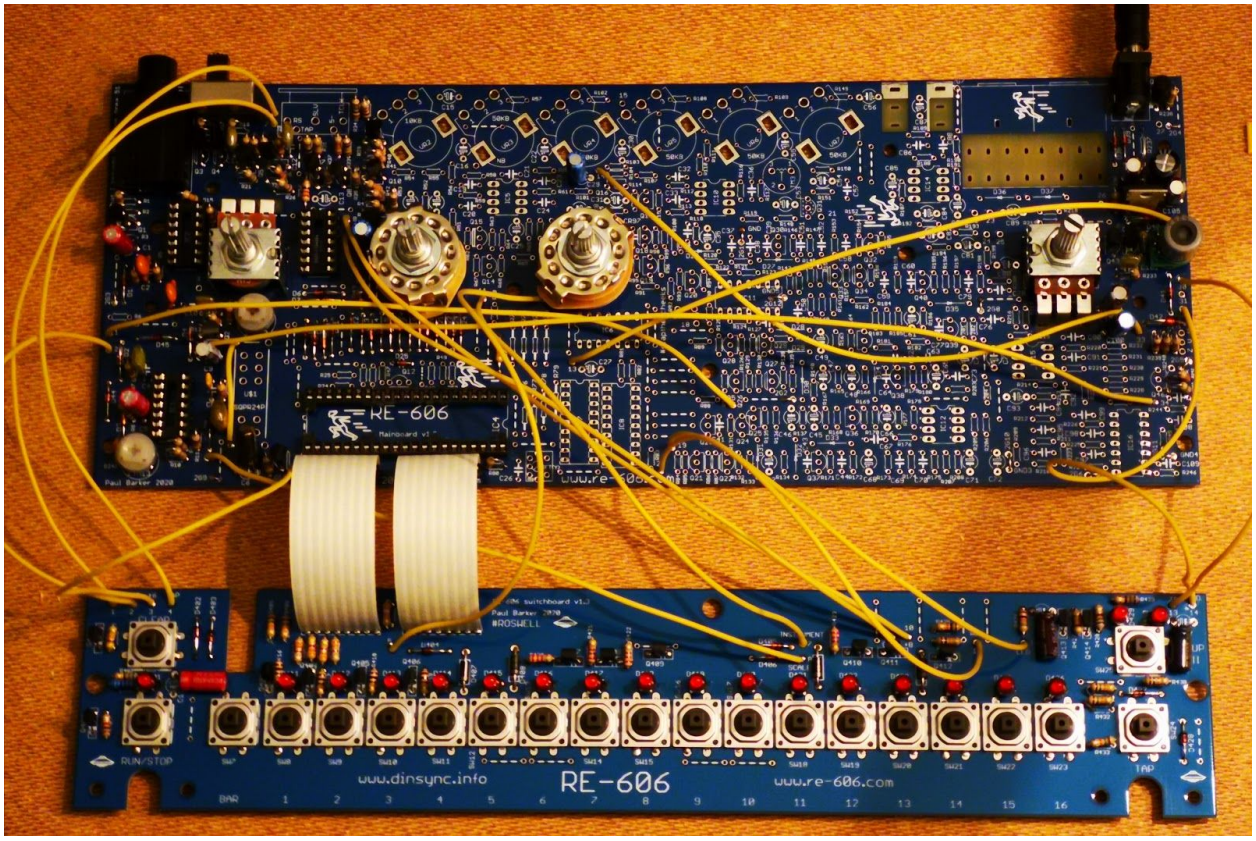
1/50 x 2, C1, C5
47/16 x 2 C14, C29
100/6.3 x 1, C6

Ceramic

220P x 2, C2, C3

TRANSISTORS

2SA733P/2SA733Q/2SA1115 x 5 Q2, Q3, Q4, Q9, Q13
2SC945P/2SC536F/2SC603 x 7 Q1, Q5, Q6, Q7, Q8, Q10, Q11



Interconnection wires and testing the digital section

We will need to connect the switchboard to test the digital section (leave the toms for now)

Use the following chart to make the interconnection wires, you will need to connect the mainboard to mainboard wires, and mainboard to switchboard wires.

It’s recommended to use headers so that you can detach the wires after the test, rather than soldering them direct as in the picture.

The lengths of wire needed are rounded up and also include 2cm for striping and tinning ends.

The following tables show source, destination points (highlighted in yellow) and wire lengths (highlighted in blue)

MAINBOARD	SWITCHBOARD	TOM BOARD	MAINBOARD	LENGTH (cm)
1			35	24
2	2			9
3	N/C	N/C	N/C	-
4			34	31
5	8			24
6	4			16
7	1			16
8	10			24
9	9			24
10	3			12
11	7			16
12	5			16
13			28	22
14			33	22

MAINBOARD	SWITCHBOARD	TOM BOARD	MAINBOARD	LENGTH (cm)
15		4		12
16		1		16
17		2		16
18		3		16
19	12			16
20	11			16
21		5		12
22		6		12
23		7		12
24		8		12
25		9		12
26		10		12
27			Battery +	
28			13	*
29		11		12
30	13			12
31			Battery -	
32	14			12
33			14	*
34			4	*
35			1	*

Power test

Before fitting the IC's below its time to power on and measure the voltages at the ICs, if they test ok then place the parts except the Pixie (CPU). It's recommended to fit sockets, but you can skip them if you are confident.

IC1 4584B 5v
IC2 4013B 6v
IC3 4001B 6v
IC4 PIXIE 6v (do not fit yet)

Fit these remaining parts.

TRIMMERS

100KB x 1, TM1
1M x 1, TM2

JACK

6.3mm x 1 RUN STOP

Now we are ready to install the CPU, if you are using an original CPU you will need to fit the parts marked in the BOM in red. If you are using a Pixie RE-CPU then you can leave those parts unpopulated.

Pixie installation,

Download the pixie manual here please note this is the current version at writing this and you may wish to visit the parent branch to get the latest version.

Basically you will need to solder the header pins to the cpu but leave the far right pins empty, then you need to solder a midi cable and/or header. Follow the Pixie manual for the details.

Once your CPU is fitted and the midi is ready the simple guide to install the firmware is as follows.

1: turn on the power, some random leds with be lit.

2: download sysex file from www.d650.cc and use a sysex tool to transmit it to midi in on the 606.

Before you start, if your midi sysex program has a throttle to limit the midi speed its a good idea to turn the transmission speed down a little.

I use http://www.snoize.com/SysExLibrarian/ a mac mini with a usb uno midi interface with the speed set for 500 ms pause between messages with success.

After you have sent the firmware sysex, wait a moment then cycle the power, with luck you now have a working sequencer section. If the tempo led isn't blinking, make sure the sync switch is set correctly.

If you have a blinking tempo led, you can use the service notes to trim TM1 and TM2, page 4 has the details. If everything went as planned, you can read the user manual (direct link to Roland site) to learn and test the basic functions. It's quite straight forward.

After you are satisfied, disconnect the interconnection wires. The rest of the build is simple, if you are an advanced builder you can probably just populate the rest of the BOM, do a final test and then build the toms board.

If you are less confident you should look on page 5 of the service notes where you can clearly see the parts based on the voice circuits, build one voice at a time and proceed to test each one before proceeding. This is the slow way but would be a sure fire way of finding any faults you may make otherwise.

Final notes

If something is not working, sign up on the forum at www.re-303.com and leave your questions in the re-606 section there, I'll try to help if I can.

At high gain here is some leak from the cymbal and some trigger bleed from the tom. This is minor and the TR also has a ton of noise at high gain. These may be fixed with bom adjustments or just down to the RF shield (which was pretty hard to recreate in eagle) Either way its nominal in normal use.

A big thank you to AFX303 for the BOM, he did most of the work there, I've made a few changes and notes but only minor. I used LM358 op-amps in my builds, I doubt these are critical to the sound. You should use metal film caps if possible for correct tone.

Also the germanium diode, I've used pulls from old TR's, this needs to be tested for subs, Pixie mentioned that with his CPU it might even work with a standard 1n4148 there (untested at writing)

The CR parts are just ceramic caps! (seems to be a design change at some point as there are no surface parts on my reference boards)

I really hope you enjoy your finished machine as much as I have been enjoying mine, I've had a bit of fun fitting an internal power bank in mine for beats on the go. It's awesome!

Now go get drumming!

Paul