JHAIBLE TAU PHASER (Euro)

Jürgen Haible's Tau "Phaser is a redesign of the legendary and extremely rare Tau "The Pipe" Flanger - inspired by Aries, ARP and Tau circuits. A bit of history: the original Tau phaser was developed by Dennis Colin and Ken McNeill in 1975 and released by Tau Systems of Newton, New Hampshire around 1980. Dennis Colin designed the ARP 2600 synth as well as the Aries synth.

Jürgen Haible's design was inspired by those phasers, but replaced obsolete components and added a number of features and improvements. The R*S kit is an adaption of the Haible Tau phaser for Eurorack format.

Features

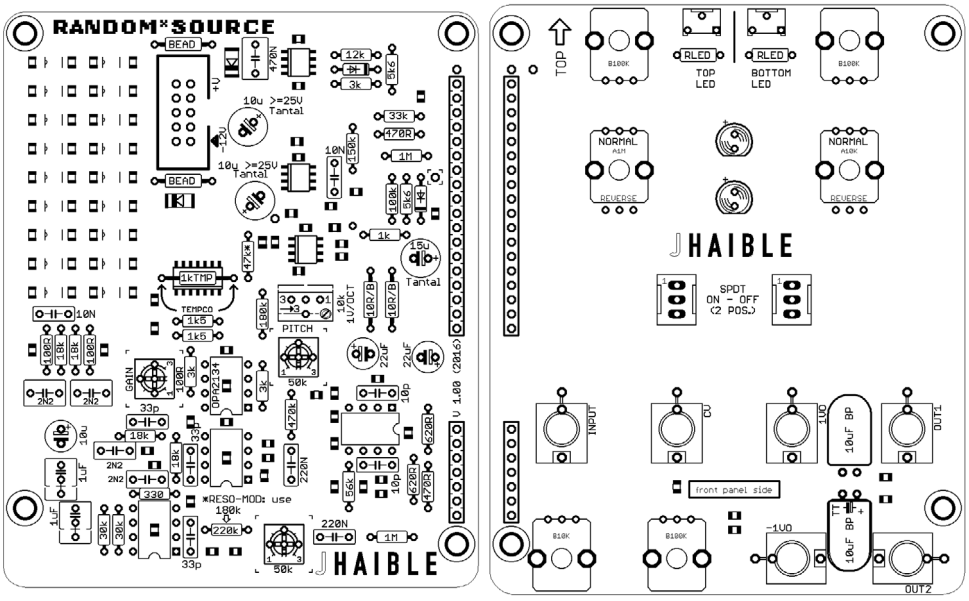
The original design by Jürgen Haible used matched transistors that are obsolete today and usually film caps for the filter poles. The R*S version already contains matched transistor arrays in SMT as well as low tolerance C0G/NPO capacitors for superior audio performance. Portions of the circuitry have been changed to SMT to reduce the board size for Eurorack and to avoid obsolete or hard to find parts.

Extensive voltage control has been added: while the original version only has a 1V/Oct input, the RS version offers an additional -1V/Oct input (providing the opposite behavior) and an additional variable CV input (attenuverting). All these can be used simultaneously for crazy sci-fy effects.

Front Panel / Controls

- Audio input with level pot
- **Extended CV control**: 1 V/Oct, -1V/Oct and an (attenuverting) CV input
- 2 Outputs: normal and alternative (allows pseudo-stereo output)
- Internal LFO with speed control - LFO DEPTH blends from 100% LFO to 100% PITCH + CV control
- PITCH knob for manual phase shifting
- FEEDBACK knob for resonance control up to self oscillation
- A switch for high pass filtering of the phaser LFO control voltage (turning the triangle LFO into a sine shape to avoid clicks in the signal)
- A switch for inverting the effect signal (creates another output spectrum by swapping high and low frequencies)
- 2 LEDs showing the LFO speed

The R*S kit consists of a Tau (main) pcb and a matching panel pcb which serves as an interface to the front panel. Bot pcbs version 1.00 are partly SMT and partly through-hole (TH) with all the required SMT parts already reflow-soldered - i.e. only the TH parts have to be added and soldered:



Please note:

- The SMOOTH switch has been implemented as by Jürgen's design - it's effect (on the internal LFO control) is very subtle, most likely you will only notice a difference at fast LFO speeds.
- The (main) Tau board was designed by Jürgen Haible to be powered by +/-15V (or 18V AC). However, the Euro version has been adjusted to be powered by a **+/-12V stabilized PSU** only. Any given for the main pcb in this document assume such **+/-12V power supply.**
- Random*Source has aquired all rights to Jürgen Haible's electronic heritage and is the only legitimate source for Jürgen Haible's designs.

Bill of Materials

Trimmers

2	2K or more	LED - on panel pcb -	Trimpot (Bourns 3362P or Vishay T73YP202KT20 or anything that matches the footprint) to adjust the LED brightness. Pick value depending on LED. Optional - use trimpot or LED resistors (not both!)
1	100R	GAIN	Trimpot - Single Turn should suffice (Bourns 3362P or Vishay T73YP...)
1	10k	T1VOCT	Single Turn or Multi-Turn
2	50k	RESO, TR-PITCH	Single Turn

Resistors (1%)

2	RLED		* alternative to LED trimpots * pick according to LEDs, 2k to 5k should work for normal low current LEDs These LED resistors are in addition to Jürgen's design, i.e. they are optional to lower the brightness, i.e. you should be able to use small values like 100R, usually 1k or so should be fine. Use UPRIGHT trimmers so you can access them from the top.
2	100R	R13, R14	
1	330R	R15	
2	470R	R6, R57	
2	620R	R31, R33	
1	1k	R67	
2	1k5	R47, R48	
3	3k	R16, R17, R63	
2	5k6	R61, R62	
1	12k	R68	
4	18k	R9, R11, R21, R25	
2	30k	R7, R8	
1	33k	R65	
1	47k*	R53	*to be changed (only) if you use a different TEMPCO size, i.e. not 1K.
1	56k	R40	
1	100k	R60	
1	150k	R66	
1	180k	R49	
1	180k / 220k	R4	Resonance-Mod (original value is 220k)
1	470k	R26	
2	1M	R1, R59	
2	BEAD	F3, F4	Ferrit Bead
2	10R/B	F1, F2	10R or Ferrit Bead
1	1k TEMPCO	R52T	+3300ppm/K ...3500ppm/K, available from THONK

Capacitors

2	10p	C5E, C5F	C0G/NPO
3	33p	C2, C5A, C5B	C0G/NPO - optional - you should be able to omit them when using OPA2134 or OPA2227 - install if needed to avoid self-oscillation or when in doubt.
4	2N2	C5, C6, C5C, C5D	C0G/NPO
2	10N	C7, C53	C0G/NPO
2	220N	C1, C9	Film
1	470N	C54	Film
2	1uF	C3, C4	Film (e.g. WIMA)
1	10u	C8	Electrolytic
2	10u Tant.	CP1, CP2	min. 25V
2	10uF BP	C10, C11	BIPOLAR / NON-POLAR for audio use Electrolytic (or 22uF) >= 25V, 2.5mm ls e.g. Nichicon MUSE, Mouser: 647-UES1V100MEM
1	15u Tant.	C52	Tantalum
2	22uF	CB1, CB2	Electrolytic

ICs

4	OPA2134	U1, U2, U3, U4	Alternatively, use OPA2227 (fancier!) for U4
2	1N4148	D1, D2	Diode

Potentiometers (Alpha 9mm vertical pcb mount)

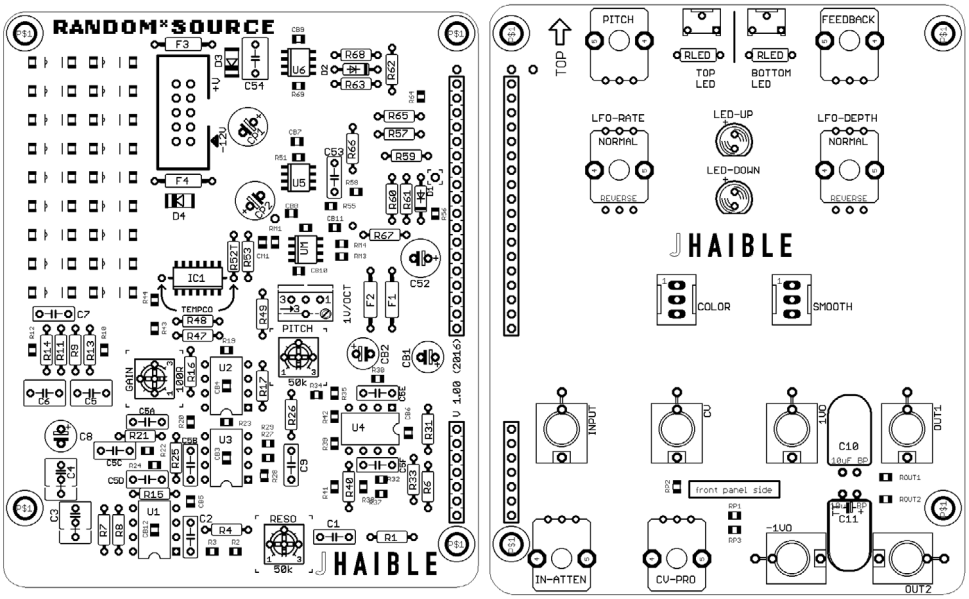
available from Thonk, Tayda

1	A10K	LFO-DEPTH	10k logarithmic
1	B10K	IN-ATTEN	10k linear
3	B100K	CV-PRO, FEEDBACK, PITCH	100k linear
1	A1M	LFO-RATE	1M logarithmic - sets range of LFO speed knob

Misc

6	Knobs		matching pots, e.g. Davies
2	Switch SPDT 2-Positions	(ON - NONE - ON) or (ON - OFF)	Sub-Miniature Switch, e.g. Mountain Switch (Mouser: 108-0042-EVX)
6	Thonkiconn Jacks		3.5mm Jack Sockets (PJ301M-12) from Thonk
2	LED lenses 5mm		VCC, Mouser 593-3000R (red), 593-3000A (amber) ...
2	LED 5mm	low current (max) 2mA	pick color to suit LED lenses
1	Euro Power header		MTA-100 power connector, Reichelt: WSL 10G
1	SIL header(s) 18 pins (10 + 8) 7 pins		pin connectors, linking main pcb to component pcb - using precision strips allows to break off pieces as needed. Use as indicated on panel pcb - only connect the pads / pins within the white boxes!

Silkscreen with Reference Designators:



Power Connector

The module is designed to be powered using a standard Eurorack 10-pin DIP header (pinout +12V / GND / GND / GND / -12V with the **red stripe on the cable indicating the -12V side**).

Resonance Modification

By reducing the input resistor of the resonance stage (R4) from 220k to 180k, the resonance can be increased up to self oscillation. This allows for additional interesting sound effects. This is a modification suggested by Jürgen Haible.

Building

1. Use a side-cutter to separate main pcb and component pcb.

2. Main pcb and component pcb are to be connected through precision SIP sockets and pins (pin headers). It is recommended to use the pins on the main pcb (facing down, soldered from above) and the pin sockets on the component pcb (standing up, soldered from the front panel side). Break or cut off the pieces you need and stick them together.

3. Attach screws / spacers (10mm or hight matching the pin headers to connect) to the panel pcb (this gets more difficult once the component pcb is connected to the front panel) and insert the pin headers connecting the pcbs. Main pcb and panel pcb should form a nice sandwich (don't solder yet). Check that you didn't leave out any pins / holes and that the sockets are all on the same side (component pcb). Solder all the pins in while keeping the sandwich together - this avoids any misalignments.

4. **Carefully** separate the sandwich - if you used precision sockets, this may not to too easy - they stick together nicely (giving a good connection).

5. Mount the Thonkiconn jacks, the pots and the switch onto the panel pcb. Pots should sit on the side facing the front panel (as marked on the board). Don't solder them in yet.

6. Insert and solder the BP (Audio) Caps (C10, C11) and the LED trimmers (or resistors - not both!) onto the panel pcb. The BP caps should lie (flat) between the Thonkiconn jacks.

7. Carefully mount the panel pcb (with the pots etc. inserted) onto the front panel. You may then have to wiggle each pot a bit to get the pots through. Make sure the threads of the pots go through completely and the pots sit right at the front panel. Screw the jacks, pots and switches to the the front panel to make sure of that.

8. Once everything is nicely in place, solder the pots, jacks und switch onto the component pcb (while the front panel is attached).

9. Build the main board with all through-hole-parts required, beginning with the resistors, caps etc.

10. Connect a power cord supplying +12V, GND, GND, -12V to the power-header on the main board and double check the direction of the power header before you turn power on. You should be ready to go now :-)

Calibration

The main pcb contains a couple of trimmers. Jürgen's original setup advice is simple:

> *"Really, you can't do much wrong with these trimmers. The range of each is so small that there is only minor adjustment. Best start with each in mid position, apply signal, and then adjust!"*

I found that with the PITCH trimmer you may be able to overdo it, turning the module into a pure noise (actual white noise!) source - this is very audible and you just have to turn back a bit.

Tracking: If you install the Resonanace-mod describe above, you get the phaser to self-oscillate, but might find the tracking way off - calibrating the generated wave to **1V/OCT** is a bit tricky, as the PITCH trimmer seems to have a big influence on the 1V/Oct behavior: I found that first adjusting the PITCH for (roughly) 1V/Oct and then finetuning with the IV/OCT trimmer worked best. Of course the PITCH trimmer affects the overall pitch range, so make sure you are (still) happy with the "normal" phasing behavior before focusing on the fine-calibration of tracking. And don't expect oscillator accuracy here: Jürgen mentioned somewhere that tracking is not so great anyway.

Power Consumption

Power consumption: <=80mA @ +12V and <=80mA @ -12V
Module width: 18HP, depth: < 35 mm

(Version 13 November 2016)