HIGGS
PARTICLE PHASER

Build Document last updated november 2024
Version 1.0 2024

The Higgs Particle Phaser is a unique CMOS based phaser that has an ADC that turns the analog signal into a digital 1-bit stream and then uses shift registers to create the phase shifting.

Have fun building and playing the Higgs Particle Phaser!

A finished build (prototype #2)

Power

Input voltage - 9V DC
Current draw - 100mA (Make sure that your PSU can supply enough current)

Controls

Potentiometers

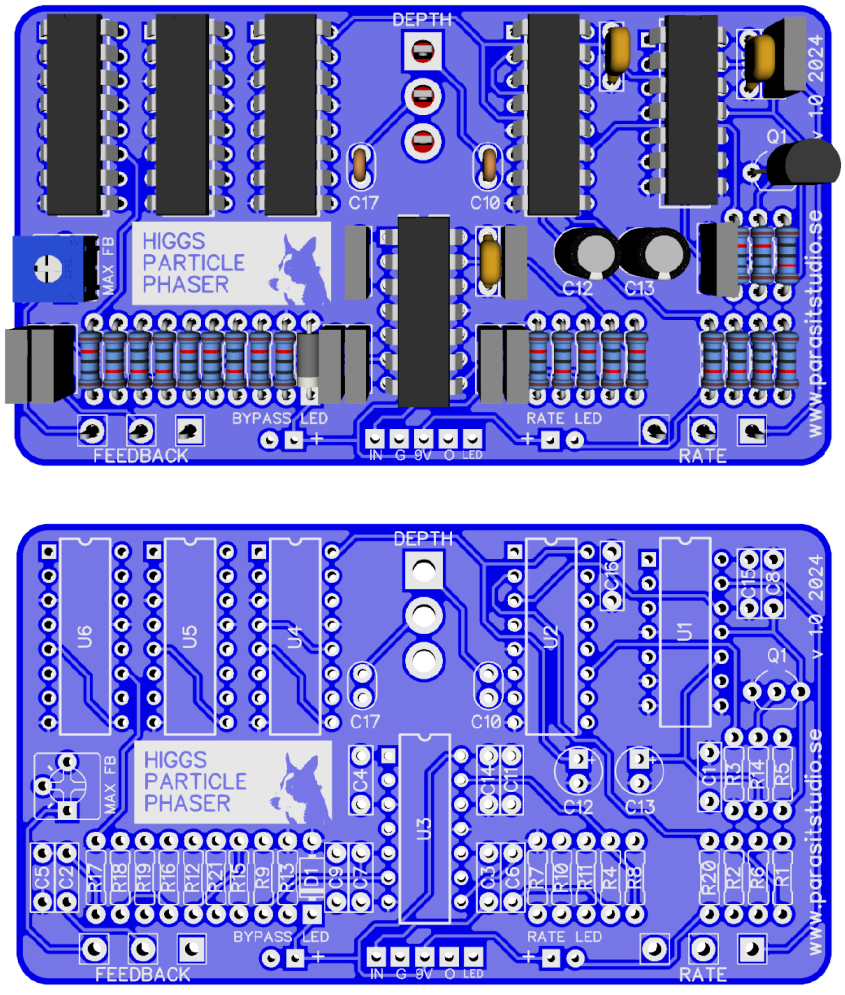
- Rate – LFO rate/speed
- Feedback – Controls the feedback of the phaser

Switches

- Depth – changes the depth of the phasing

The populated PCB

Here's a 3D render approximation of what the fully populated board should look like (except that the IC's should be in sockets).



The PCB measures 85mm wide x 48mm tall

General building tips

- Just follow the Bill of Materials and solder the low profile components first.

  Recommended order: resistors and diodes, chip sockets, trim pot, multilayer and ceramic capacitors, film box capacitors, electrolytic capacitors, pots and switches, offboard wiring (jacks and the 3PDT switch). Bend the legs of the components alittle bit to prevent them from falling out, or use tape to hold them in place while soldering.

- Pay special attention to the orientation of the LED's, diodes and the electrolytic capacitors.
- Always use sockets for IC chips to avoid heating them directly. It also makes it much easier to swap them out if needed.
- CMOS chips are very sensitive to static charges and can be easily damaged. It's a good idea to wear a anti-static wristband. Or at least don't wear a woolen jumper and pat your dog while building, and keep the circuit away from rugs... Put the chips in last, after everything else is soldered in place.
- Break off the small tap on the potentiometers, so they can sit flush against the top cover.
- Make sure that the backside of your pots are covered so they don't short anything on the PCB. If you not have pot covers I recommend pvc electrical tape.
- When it's time to solder the potentiometers, switches and buttons I recommend having the enclosure/lid prepared to make sure that they line up with the holes.

  I recommend that you solder only the middle pin of each potentiometer to the PCB (so that the placement matches the silkscreen on the PCB and the pot stays in place when you turn the PCB around but still has some wiggle room). Screw in the toggle switches in the enclosure and then put the PCB with the pots into the enclosure so that everything fits and finally solder the rest of the pot pins and the switches from the component side of the PCB.

- *The oscillator IC can become quite hot in this circuit, as the oscillator is running at a very high frequency (but well within max operating limits). It's nothing to worry about, but feel free to add a small heatsink if you want to.*

Higgs Particle Phaser BOM (Bill of Materials)

Resistors		Capacitors		IC's	
R1	12K	C1	22nF	U1	CD4013
R2	10K	C2	22nF	U2	CD4046
R3	10K	C3	22nF	U3	CD4069 (UBE)
R4	10K	C4	15nF	U4	CD4517
R5	1K	C5	33nF	U5	CD4517
R6	1.5K	C6	100nF	U6	CD4517
R7	1M	C7	100nF		
R8	1M	C8	100nF	**Transistors**	
R9	82K	C9	1nF	Q1	3N3904
R10	100K	C10	47pF		
R11	100K	C11	68nF	**Potentiometers**	
R12	100K	C12	47uF	RATE	C50K
R13	100K	C13	47uF	FEEDBACK	C50K
R14	100K	C14 **	100nF		
R15	100K	C15 **	100nF	MAX (trimpot)	50K ***
R16	22K	C16 **	100nF		
R17	22K	C17	100pF	**Switches**	
R18	22K			DEPTH	SPDT on/on
R19	4.7K	**Diodes**			
R20 *	10K–22K	D1	1N4001		
R21 *	10K–22K	1x LED Rate and 1x Bypass LED (2x LEDs)			

BOM Notes

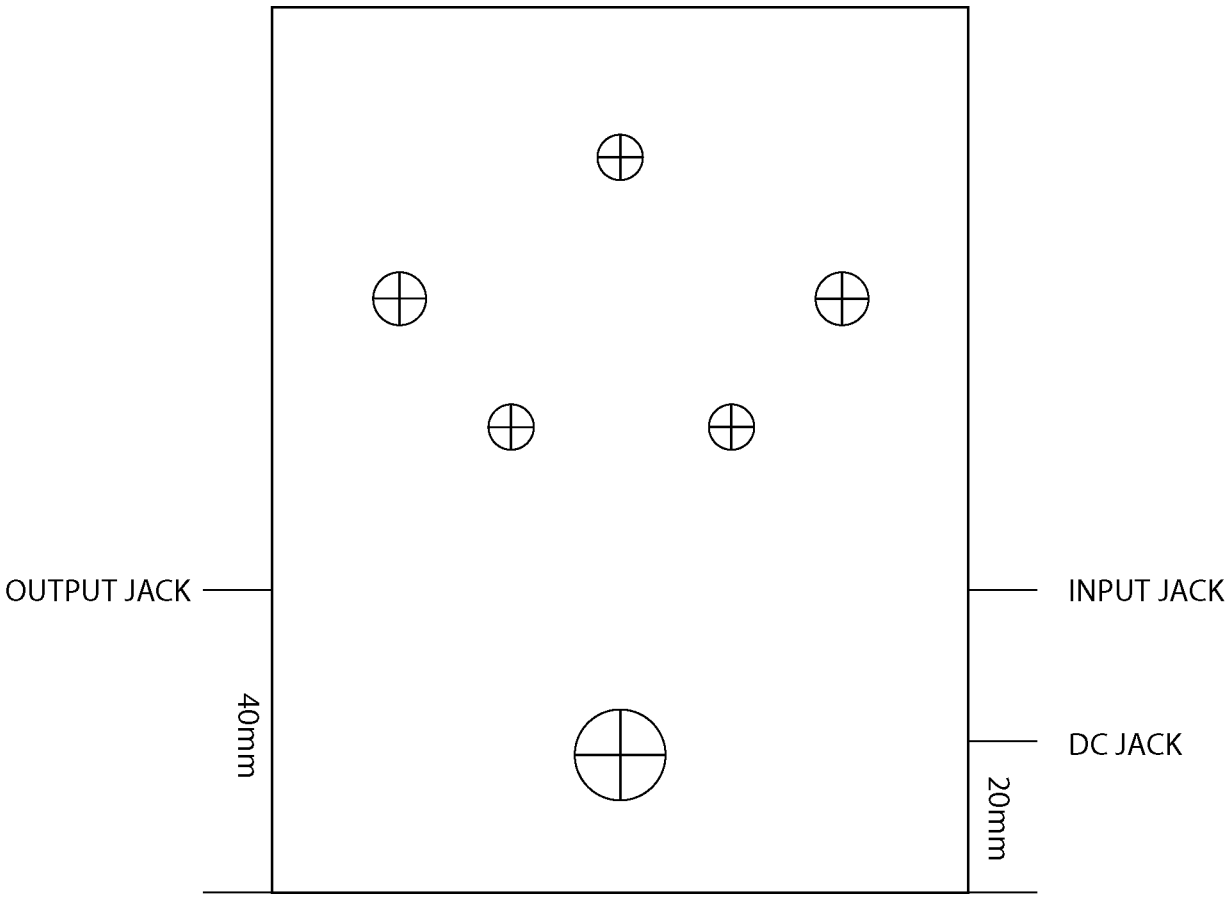
- * = current limiting resistors for the bypass LED and the rate LED. *The rate LED should be a minimum 10K value to reduce ticking for the rate LED. I recommend using a "super bright" type of LED and a high resistor value.*

- ** = should be a multilayer ceramic capacitors (yellow capacitors)

- *** = Max feedback trimpot. A 100K trimpot works fine as well, but a 50K give alittle more fine control.

- Things that are not included in the BOM list: enclosure, input and output jacks, DC jack, 3PDT stomp switch, led bezel and knobs.

Drill Template (1590BB)

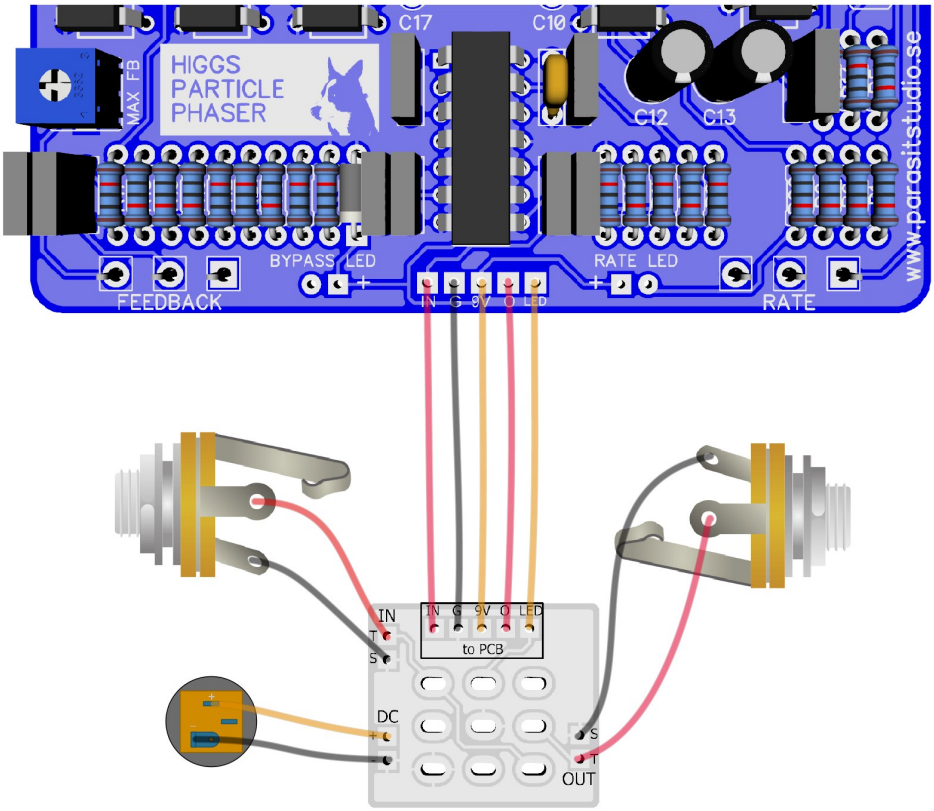


- Use at your own risk! This template is approximate.
- Make sure your printer isn't doing any scaling (100% print size).
- Jacks are measured from the edge at the front/top of the enclosure (as they were drilled on the prototype), but you can drill the positions for the footswitch, DC jack and input/output jacks to your own preference.

- Typical drill sizes are:
  - switches / LED bezel (for a 3mm LED) - 6mm
  - potentiometers - 7mm
  - DC jack / 3PDT footswitch – 12mm
    (8mm for lumberg style DC jacks)
  - input/output jacks (Neutrik style) – 9,5mm
    (9mm for Lumberg style jacks)

Measure and confirm before drilling!

Off Board Wiring



The top row of connections on the 3DPT daughterboard connects directly across to the main PCB as shown.

Input jack sleeve → ”S” IN pad *the lug that connects with the inner ring of the jack*
Input jack tip → ”T” IN pad *the lug that connects to the tip bracket on the jack*

Output jack sleeve → ”S” OUT pad *the lug that connects to the inner ring of the jack*
Output jack tip →”T” OUT pad *the lug that connects to the tip bracket on the jack*

DC jack negative → ”–” DC pad *the widest lug, or the short lug (Lumberg style jack)*
DC jack positive → ”+” DC pad *the outer lug if it's a 3 pin DC connector, or the long lug if using a Lumberg style jack*

If you are not using the 3PDT daughterboard PCB, have a look at the offboard wiring diagram here (fig1/3): wiringrev3.pdf (parasitstudio.se)

Troubleshooting

There's always a chance of running into trouble. To minimize error, follow the BOM and general building tips carefully. Take your time and don't rush. Take a break now and then. Use good solder, and it helps to have a decent soldering station insted of a cheap iron.

Musikding DIY kit

If you have bought the Musikding DIY kit and have recieved a faulty faulty, incorrect or missing component, please contact musikding.

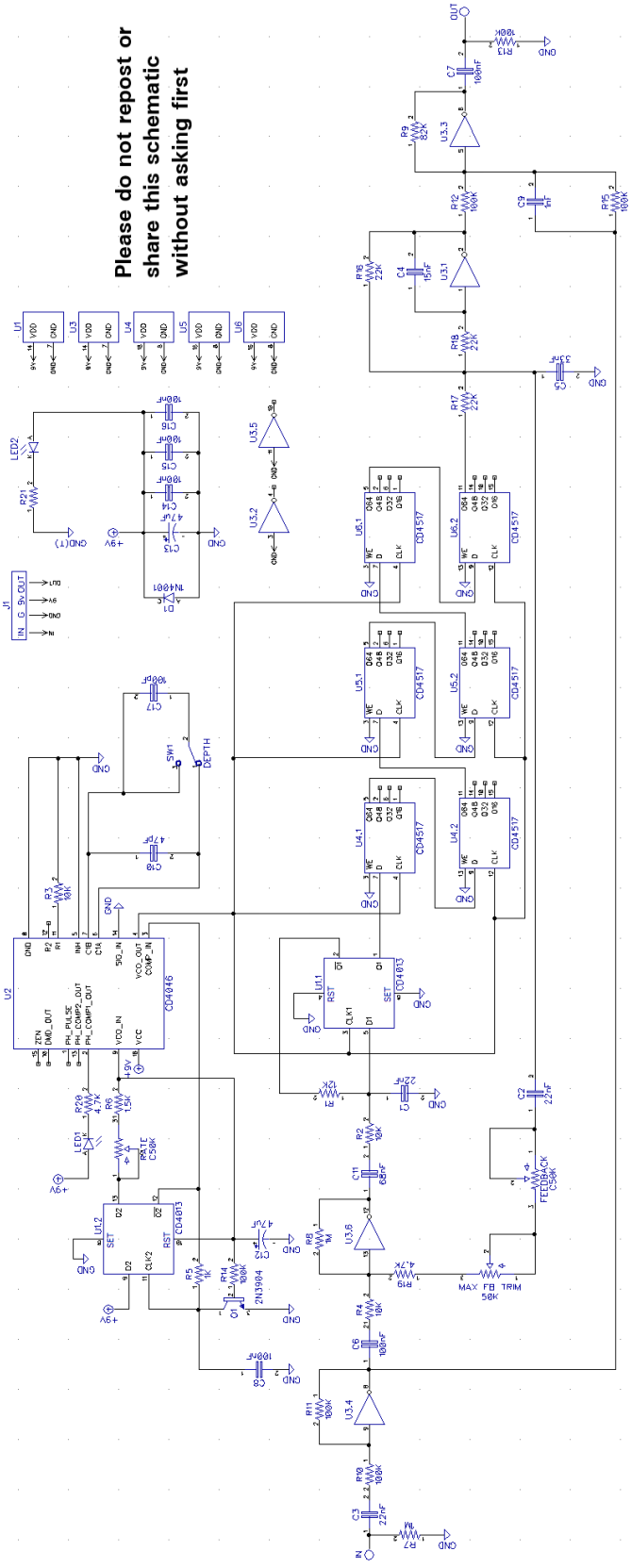
Contact us

Terms of use – please read

PCB's and circuit designs from Parasit Studio are intended for personal use only. No commersial use. It's not ok to build and sell these pedals without permission. However, it's totally ok to build a few pedals and give to your friends and bandmates. After all, that's what this hobby is about. :)

www.parasitstudio.se
parasitstudio@gmail.com

Schematic


Please do not repost or
share this schematic
without asking first