Hypno

Delay Kit by Pedal Markt

Apr 25, 2025
Rev1.0

Contents

1 Introduction

TODO

Enclosures for Hypno and other pedals in the Beastly Series were designed by Agata Fiz.

Figure 1: Hypno: oustide and inside

2 BOM – Bill of Materials

BOM is a document that lists the parts you'd need to build a project. Each row corresponds to a component with a certain value, for example, a 'ceramic capacitor with value 1nF.' There could be one or more actual physical parts per row, their designators are listed in the *Reference* column.

In the BOM *text in italic font* gives tips about how to mount or solder parts.

Table 1: BOM

	Ref	Value	Qnty	Description
Outboard				
	–	Enclosure	1	*Mount the DC jack, Footswitch and Lampshade into the enclosure before soldering*
	–	Lampshade	1	Small transparent plastic part for the LED, *mount in enclosure before putting the boards in*
	–	Rubber Ring	1	*Use it to keep Lampshade in place*
	–	DC Jack	1	Black plastic part with a nut, *mount in enclosure before soldering*
	–	DC Cable	1	Red and black cables in a JST connector, *cut to* $\approx 10cm$ *and solder to DC Jack once it's mounted in enclosure. Black wire to shorter lug, red to the longer one.*
	–	Audio Jack	2	*Only mount these in the enclosure together with the main board once they are wired up*
Main board, floor side				
	GND	Wire	2	$\approx 5cm$, black, *strip and tin the ends*
	IN	Wire	1	$\approx 5cm$, any color, *strip and tin the ends*
	OUT	Wire	1	$\approx 5cm$, any other color, *strip and tin the ends*
	R8	5.6k	1	Resistor
	R9	56k	1	Resistor
	R10	100	1	Resistor
	R11	68k	1	Resistor
	R12	100k	1	Resistor
	R19	470k	1	Resistor
	R20	1k	1	Resistor
	R21	6.8k	1	Resistor
	R2, R7	15k	2	Resistor
	R13, R14	1M	2	Resistor
	R15, R16, R17, R18	140k	4	Resistor

Continued on next page

Table 1: BOM (Continued)

	R1, R3, R4, R5, R6, R25, R26, R27	10k	8	Resistor
	U2	PT2399	1	Delay chip. *Please use socket. Orientation matters*
	U3	TL072	1	Dual opamp. *Please use socket. Orientation matters*
	C18	1u	1	Ceramic capacitor
	C20	2.2p	1	Ceramic capacitor
	C7, C13	560p	2	Ceramic capacitor
	C15, C24	100n	2	Ceramic capacitor
	J1	Power Socket	1	JST 2-pin m, in the bottom-left part of the board, *orientation matters*
	Q1, Q2	2N3904	2	NPN transistor, *orientation matters*
	U1	L78L05	1	Voltage regulator, *orientation matters*
	C14	10n	1	Film capacitor
	C5, C11	3.9n	2	Film capacitor
	C8, C9, C10, C12, C19	100n	5	Film capacitor
	C1, C17	1u	2	Film capacitor
	C6	100n	1	Electrolytic capacitor, *orientation matters*
	C3	330n	1	Electrolytic capacitor, *orientation matters*
	C26	100u	1	Electrolytic capacitor, *orientation matters*
	C16, C25	10u	2	Electrolytic capacitor, *orientation matters*
	C2, C4, C23	4.7u	3	Electrolytic capacitor, *orientation matters*
Main board, player side				
	–	Ribbon cable	1	Pads for that cable are in the bottom-center of the main board, *solder one end to main board, another to switch board,* ***make sure pin names on the two boards match, IN on one board is connected to IN on the other board etc***
	RV1 (Sink)	C10k	1	Potentiometer
	RV2 (Recall)	A100k	1	Potentiometer

Continued on next page

Table 1: BOM (Continued)

	RV3 (Lucidity)	B100k	1	Potentiometer
Switch board, player side				
	Rled	1k	1	*larger value will make the LED dimmer, values up to 6.8k are reasonable*
	–	LED	1	*Insert in PCB first. Solder last, once the main board is in the enclosure. Orientation matters*
	–	Footswitch	1	*Mount in enclosure before putting the boards in*

2.1 Note on values

Different kits and schematics designate values differently. For example, these usually mean the same value:
$2.2\,\mathrm{k\Omega} = 2.2k = 2k2 = 2.2 \times 10^3 Ohm = 2200 Ohm$
$4.7\,\mu\mathrm{F} = 4.7u = 4u7 = 4.7 \times 10^{-6} Farad = 0.0000047 Farad$

Table 2: Component values

Value	Multiplier	Unit
Resistance		
100 Ω, 100R, 100	1	Ohm
1 kΩ, 1k	10^3	Ohm
1 MΩ, 1M	10^6	Ohm
Capacitance		
1 pF, 1p	10^{-12}	Farad
1 nF, 1n	10^{-9}	Farad
1 µF, 1u	10^{-6}	Farad

3 Transistors

The circuit consists of just a handful of parts. Transistor selection and bias play a huge role in the sound of the pedal.

The stock transistors we use for Fur Face are BC108, Q1 biased to about 1.3 V, Q2 to about 5.8 V.

If you'd like to try out and hear different transistors, the best way is to solder sockets in place of Q1 and Q2 on the PCB. Swap transistors in those sockets without soldering them.

When inserting the transistors, please match their pinout to the expected pinout indicated on the PCB: E to emitter, B to base, C to Collector. To find out the pinout of a transistor, you could use image search, try to find its datasheet, or use a transistor tester.

3.1 Biasing transistors

Biasing transistors in this context means setting their collector voltage to a certain value, while there is no audio signal coming in.

Because Fur Face internally uses feedback for bias, you'd have to go back and forth a couple of times between Q1 and Q2 to make sure the voltages are right. That's because changing the bias of one of the transistors affects the other.

Here's the biasing procedure:

- Set your multimeter up for measuring DC voltage. We'll be measuring values between 0 V and 9 V;
- Clip or touch the black (ground) lead of the multimeter to any ground point in the circuit. For example the lug on the audio jacks that's connected via a black wire to GND on the PCB;
- Starting with transistor Q2, then moving to Q1, then back and forth between the two a couple of times:
  - Clip or touch the red (hot) lead of the multimeter to the collector of the transistor;
  - Make sure you can see there's some voltage readout on the multimeter;
  - With a small screwdriver twist the screw of the trimpot above that transistor, so that the voltage gets close to the desired value (Q1 to about 1.3 V, Q2 to about 5.8 V). The change is going to be slow initially,

but as the voltage grows, the increments are going to be more and more significant;

4 Schematic

5 Revision History

Revision	Date	Author(s)	Description
1.0	Apr 25, 2025	AS	Created