BSCORE DEV PEDAL PCB BUILD MANUAL



Figure 1. BSCORE Dev Pedal PCB

Table of Contents

- 1.Schematic Diagram
- 2.Off-Board Parts Wiring Diagram
- 3.Top Layer Layout
- 4.Bottom Layer Layout
- 5.On-Board Part List
- 6.Off-Board Part List
- 7.Printable Enclosure Drill Template
- 8.Printable Enclosure Label
- 9. Building Guide

Contents

1. Schematic Diagram



Figure 2. Schematic Diagram of BSCORE Dev Pedal Circuit

2. Off-Board Parts Wiring Diagram



Figure 3. Off-Board Parts Wiring Diagram

3. On-Board Part List

Note: C11 uses multi-layer ceramic capacitor (MLCC) because it should have maximum height of about 5 mm when soldered onto the board, you can use value between 0.1 to 1uF for this capacitor.

ID 	NAME	DESIGNATOR	QTY
1	BSCORE Module + MALE-HEADER RIGHT-ANGLE-2X15	BSCORE-HDR	1
2	10uF/16V electrolytic capacitor	C1,C2	2
3	4.7uF/16V electrolytic capacitor	C3,C4,C5,C6,C7,C8	6
4	47nF Capacitor	C9,C10	2
5	100nF Multilayer Ceramic	C11	1
6	1nF Capacitor	C12,C13,C14	3
7	1N4002/1N4001/1N4007 Diode	D1	1
8	1N4148 Diode	D2	1
9	LED 3MM (RED)	D-AUX,D-MAIN,D-POWER	3
10	MALE HEADER-1×4	D-HDR	1
11	MALE HEADER-1×2	DCIN-HDR,DGND-HDR,INL-HDR,INR- HDR,MICIN-HDR,OUTL-HDR,OUTR-HDR	7
12	MALE HEADER 1×3	MIDIIN-HDR,MIDIOUT-HDR,UART-HDR	3
13	B10K, right angle PCB-Mount potentiometer	P1,P2,P3,P4,P5,P6	6
14	10R , 0.25W Resistor	R1,R2	2
15	330R , 0.25W Resistor	R3	1
16	47R , 0.25W Resistor	R4,R5	2
17	220R , 0.25W Resistor	R6	1
18	100K, 0.25W Resistor	R7,R8	2
29	1K, 0.25W Resistor	R9,R10,R13	3
20	1M , 0.25W Resistor	R11,R12	2
21	DPDT ON-OFF-ON Toggle Switch	SW-MIDITTL	1
22	SPDT ON-ON Toggle Switch	SW-POWER,SW-PROG	2
23	LM1117T-3.3 Voltage regulator IC	U1	1
24	H11L1 optocoupler	U2	1

4. Off-Board Part List

Note: Use isolated 3.5 mm TRS sockets for J1-J4 (like shown in the Figure 4), since the board has separate audio ground and digital ground to minimize the digital noise. For audio input-output (J5-J8), use mono or stereo socket with small case overhead (like shown in the Figure 5), so it could fit in this critical spacing design.

DESIGNATOR QTY

2

4

4

1

- 1 SPST Momentary Footswitch FS1,FS2
- ISOLATED 3.5 mm TRS socket J1,J2,J3,J4
 MONO audio socket 6.35 mm J5,J6,J7,J8
- 5 ISOLATED 2.1 mm DC socket JD



Figure 4. Isolated 3.5 mm TRS Sockets



Figure 5. Small Case-Overhead 6.35 mm Audio Sockets

5. Top Layer Layout



Figure 6. BSCORE Dev Pedal PCB Top Layer Component Layout

6. Bottom Layer Layout



Figure 7. BSCORE Dev Pedal PCB Bottom Layer Component Layout

7. Printable Enclosure Drill Template (125B Enclosure)



8. Printable Enclosure Label (125B Enclosure)

CTRL 1	CTR	RL 2	CTRL 3				
CTRL 1	CTR	RL 2	CTRL 3				
CTRL 1	CTR	RL 2	CTRL 3				
CTRI 4	СТР						
CIRL 4	CIN	łL 5	CTRL 6				
BSCORE DEV PEDAL							
< M	IDI O	N RUN	MIC > >				
< N. < UA	C. ART	PROG	;>				
< R-OUT	OI	F	R-IN >				
MAIN	POV	VER	AUX				
			E(MONO)-IN >				
	21	DJ					
	51	B2					

9. Building Guide

- 1. Make sure the enclosure has been drilled, painted, and labeled.
- 2. Solder all components of the top layer, including all the header connectors. Make sure the LM1117-3.3V is mounted on the board to have as short as possible height.
- 3. Solder all bottom layer resistors (R9, R10, R13), those are all bottom layer components except the potentiometers, LEDs, and switches.
- 4. To reduce the mechanical stress on the PCB when mounted the the enclosure through by potentiometers and switches, follow these steps:
 - 1. Install all potentiometers (P1-P6) and toggle switches (SW-POWER, SW-MIDITTL, SW-PROG) loosely to the enclosure.
 - 2. Mount the PCB to the mounted potentiometers and switches and align perfectly with the enclosure, keep it aligned without soldering.
 - 3. Tighten the nuts of all the potentiometers and switches while keeping the board mounting aligned, still without soldering.
 - 4. Solder all potentiometers and switches except P1, P2, and P3.
 - 5. Cut the leads of P1, P2, and P3 as short as possible, so they have zero height before get soldered.
 - 6. Solder P1, P2, and P3 without excessive tin so it produces thin or flat soldering surface in the top layer. This will make the BSCORE module insertion possible/easier.
 - 7. Unmount the board and the potentiometers assembly from the enclosure.
 - 8. Re-solder the potentiometers P1, P2, P3 from the bottom to add more strength, but be careful not to let the tin flows to the other side.
 - 9. Mount the LEDs with the following steps:
 - 1. Twist the LEDs for D-POWER, D-AUX, and D-MAIN before mounting them to the PCB, as shown in the figure 8. This twisting enables omni-directional horizontal adjustment after get soldered to PCB.
 - 2. Mount each LEDs and adjust the height to get aligned with the adjacent switch as the reference.
 - 3. Solder the LEDs from both sides (top and bottom layer) to get maximum strength against pushing and pulling off the board.
 - 10. Cut all the excessive components leads.
 - 11. Re-mount the board assembly to the enclosure through the potentiometers switches, secure their nuts tightly.
 - 12. Mount the footswitches (FS1, FS2) to the enclosure, and wire them to the board using single female header cable jumper (Figure 9). See the Figure 3 for the wiring reference of the off-board parts.
 - 13. After the footswitches get wired, you can mount all other sockets (JD, J1-J8) to the enclosure and wire them to the board using cable jumper/single female header in

any order (Figure 10(A)). See the Figure 3 for the wiring reference of the off-board parts.

- 14. Insert the BSCORE Module to its header (Figure 10(B)).
- 15. Secure the enclosure with its base-lid, now the pedal is ready to use (Figure 11).



Figure 8. Lead Twisting Before Mounting The LED



Figure 9. Single Female Header Dupont Connector Cable Jumper



Figure 10. BSCORE Dev Pedal Assembly (A) Without The BSCORE Module, (B) With The BSCORE Module



Figure 11. Ready-to-Use BSCORE Dev Pedal