BG Micro
Geophone Display Kit

Instructions

1. Solder R9 into the board first. Be extra careful about the value and installation, as this part will not be accessible after the LM3915 socket is soldered in. Note Figure 3 and the color codes on the Parts List.

2. Solder the remaining fixed resistors into the board (R1-R3, R5-R8). Note Figure 3 and the color codes on the Parts List.

3. Solder the IC socket into the PCB. Pay special attention the notch on the socket and the notch shown on the PCB. It should also be flat on the board.

4. Solder C1 and C2 into the board. Allow some strain relief on the leads.

5. Solder J1 and J2. Pay careful attention to the orientation of the connectors shown in Figures 1 and 2. They should also be flat on the board, no space between their bottoms and the PCB.

6. Insert the IC into the socket. You may have to gently squeeze the pins of the IC slightly together to get the IC to fit into the board, but don’t overdo it. It is critical that the notch on the IC match both the socket and the picture shown on the PCB.

7. Install the LEDs as shown in Figures 1, 2, 4. It is recommended you install them one at a time. Be sure they are flat to the PCB.

8. Solder R4 into the board, matching the orientation shown in Figure 2. The part should be flat on the board. For maximum gain turn R7 adjustment fully clockwise.

9. Inspect your work. Mistakes caught now may prevent burned out components later. Solder joints should be shiny and flow between the leads and PCB.
10. Install the power and signal (audio) leads into their connectors. Power PCB with 6VDC to 15VDC (12V recommended). Your display board is now ready to use!

**Optional – Bargraph Display Mode**

The IC in this design can light the LEDs as a bargraph display, but the printed circuit board (PCB) design cannot support this. A simple modification will allow you to use this kit as a bargraph.

The problem is R2 does not have enough wattage to handle the power needed. Replacing this resistor with a 1 Watt or 2 Watt resistor (not provided) will allow the kit to do this function. Be sure the wires on the replacement resistor will fit in the PCB holes.

The kit will jump from a standard draw of about 15ma to 105ma at 12VDC when all the LEDs are lit.

**Instructions**

1. Either leave R2 out of the circuit when assembling the kit or remove R2 from the printed circuit board.
2. Bend the wires as shown on Figure 5. This is to prevent the wires from the new resistor do not short out on the printed circuit board anywhere.
3. Install R2 on the bottom of the PCB. Figure 6 shows how this should look when finished. Don’t forget to clip the leads coming through on the component side of the PCB.
4. Install a jumper wire between the two holes labeled “Mode” on Figure 3 and Figure 4.

---

Figure 5 - 2 Watt Resistor replacing R2

And you are done!