

SymAsym - “The Sequel”, AAK’s DTV PCB Assembly Instructions

This document is intended to provide all the information necessary to build my PCB. The document includes step-by-step assembly and testing instructions, the latest amplifier and power supply schematics, matching transistor circuits, and picture illustrations.

I'll divide the instructions into different categories starting with assembly tools.

Assembly Tools:

1. Needle nose pliers for bending leads to secure components to the board before soldering.
2. A fine tip wire cutter to cut component leads after their soldered to the board.
3. Soldering gun of a least 40 to 60 watts. If the temperature is controllable set it to 700F for small signal transistors, resistors, small caps, rectifiers, and output transistors. 800F for soldering the PS caps, terminals, and fuse holders.
4. Magnifying glass to make sure there's no shorts caused by solder slobber.
5. Isopropyl Alcohol and tooth brush to remove soldering flux from the board.
6. A multimeter or DVM to measure resistance, voltage, and setting quiescent current (Iq). It should be able to measure hFE for matching small signal transistors.
7. An oscilloscope would be nice but not necessary. If no mistakes are made most likely you wont need one.

Assembly Tips:

1. Check each component for the correct value before inserting them into the board. Check the package for the correct part value to make sure the right component was sent. Use a DVM to check resistor values when in doubt.
2. Check to make sure the components are inserted in the correct position. Refer to the Parts Placement Diagram on Page 5.
3. Check polarized components like capacitors, diodes, zener diodes, and LEDS for the correct orientation. Refer to the Assembly Instructions section for additional information.
4. Bend leads away from adjacent pads when securing components to the board to help prevent shorting when soldering.
5. Don't over solder especially in the amplifier section. Use just enough solder to cover the pads.
6. Be careful not to overheat the small signal transistors (SST). Set the soldering gun to about 700F and quickly solder the transistor leads to the pad.
7. Solder only a few components at a time. Trying to solder around to many leads makes it more difficult to solder and could result in poor solder connections and shorts.
8. Take your time don't rush the assembly.

Assembly Instructions:

1. Start the assembly by inserting the resistors first. I left R5, R6, and R7 for later since they're standing. I like keeping the board as flat as possible to help keep the components in place while soldering. For the bias pot R36, the small adjustable screw should be on the opposite side of the R36 label near J1.
2. Insert the non-polarized capacitors. These are the small Wima caps including C7, and C8, and Mica caps C15-C19.
3. Insert the small-polarized capacitors. Verify that each capacitors positive lead is inserted into the pad hole with '+' labeled next to it.
4. Insert LEDs, L1-L5. **Insert the longer of the two leads (anode) into the pad hole with a circle labeled around it.**
5. Insert zener diodes, ZD1, ZD2 and ZD3. Note the correct orientation labeled on the board. **Note: For the 6.8V zener diode ZD3 insert the lead that's nearest to the black band (cathode) into the pad with a circle labeled around it.**
6. Insert the SST including matched pairs Q1&Q2, Q3&Q4, Q5&Q6, Q9&Q12, and Q10&Q11. **Note: If you use the 2SK170 for Q1&Q2 they need to be oriented in the opposite direction then how it's labeled on the board. The flat side of transistor should face inward towards the large PS caps.** If you plan on mounting the drivers Q20&Q21 to the main heat sink refer to step 10.
7. Insert R5, R6, R7, rectifiers D1-D4, terminals T1-T5, fuse holders, and large PS caps C3-C6. I like placing R7 underneath the board to provide additional clearance around ground terminal T4. Set the solder gun to about 800F when soldering terminals, fuse holders, and large caps. Solder the top and bottom of each terminal and fuse holder for added strength. **The rectifier's metal tabs should face inward towards the AC terminals. For the large PS caps insert the positive lead in the pad hole with '+' labeled next to it. For the fuse holders make sure the tabs face outward.**
8. Jumper J1 to J2 with a 22 to 26-gauge 1.6" (41mm) jacketed wire. The J1 pad is located right below the adjustable pot R36, and between Q8, and R29. The J2 pad is located between R32, R30 and L3. Place and solder the jumper underneath the board. Refer to the pictures in Post #1 of the DiyAudio builder's thread for reference.
9. Solder to each Q17 (BD139) pin a 2" (50mm) jacketed wire so that it can reach far enough to mount onto Q22 for thermal sensing. I like trimming the BD139's leads about 0.25" before soldering the wires to them, and then insulating the solder connections using 1/16" heat shrink tubing about 5/8" in length to help prevent the pins from shorting. Before soldering the wires to the board trim the wires to the correct length by first turning the board upside down placing the BD139's backside up over where about Q22's mounting hole would be. Lay the wires flat over the board so they don't overlap each other, then bend the leads towards the pads at a 90-degree angle and trim the wires even. Now strip the ends of the wires about 3/16" then insert the ends into the pads and solder from underneath. You may need to pull the wires out slightly to make a good solder connection. Refer to the pictures in Post #1 of the DiyAudio builder's thread for reference.

10. Insert drivers Q20&Q21. Here you have two options. Either insert the drivers from the top side, and use the small heat sink listed on the BOM, or from underneath to mount the drivers to the main heat sink (MHS). I prefer the former. If you select the later, first mount the drivers on the MHS allowing enough clearance between the mounting hole and board edge so that you can easily access the mounting screw. Then holding the board above the drivers align the pins with each pad. Use 3/8" standoffs at each corner of the board where the mounting holes are to provide an even spacing between the board and the main heat sink. Then tack on the pins to the pads with a small amount of solder from the topside to hold the drivers in place. Then remove the board and solder the drivers from underneath. I'd suggest leaving out Q18&Q19 for later to provide some extra wiggle room for the solder tip when tacking on the drivers.
11. Next output transistors Q22, Q23, Q24 and Q25, and mounting thermal tracking transistor Q17 to Q22. First mount the output transistors (OT) to the MHS, and then holding the board above the OT align the pins with each pad. Use standoffs at each corner of the board before soldering to provide an even spacing between the board and the main heat sink. Then solder the OT pins to the pads from the topside, and then remove the board and solder from underneath. When remounting the OT to the MHS mount Q17 to Q22.
12. Lastly attach a single ground wire from one of the boards ground pads located near C7 and C8 to the MHS.
13. Now clean the flux from the board by dipping a toothbrush in isopropyl alcohol. A PCB cleaner can be used as well. Wait until the boards have dried before testing.

Testing Procedures:

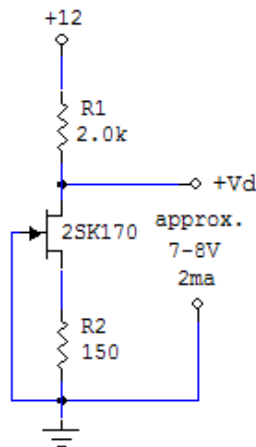
1. First lets check the power supply, **make sure the fuses are not inserted**. Then check the rectifiers for the correct orientation; the metal taps should face inward towards the AC terminals. Then connect the transformer's two AC outputs, and Center Tap (CT) to the input terminals labeled AC, and CT on the board. Then attach one of the multimeter probes to fuse holder clip FC3 and the ground probe to the GND terminal and power up the transformer. Check for the correct positive voltage then probe FC1 the correct negative voltage. **Note:** The PS LEDs will not light up unless the fuses are in place.
2. Without the fuses inserted solder a 100-ohm resistor across each fuse holder. These 100-ohm resistors are used to monitor current draw during testing, and provide component protection if a mistake was made in the assembly process.
3. Now with a small screwdriver adjust the bias pot R36 fully counter-clockwise until you hear it click. **The trimmer screw must be turned counter-clockwise 25 full turns to ensure the amplifier's output section is set to it's minimum bias of about 0ma.** If not the 100-ohm resistor may go up in smoke.
4. Now attach the multimeter probes across one of the 100-ohm resistors as illustrated in Fig 1., and set the meter to about 20V. Now supply power and check the voltage across each 100-ohm resistor. It should measure somewhere between 2.5 and 3.5 volts, and the three green LEDs in amplifier section should light up with nearly equal intensity. If the voltage is within range then check the output DC offset by connecting

the multimeter probes to the Gnd terminal, and speaker output terminal. It should measure less than 50mv. I measured less than 5mv in the new amps that I built. It varies depending on how well the SST are matched.

5. Now remove the 100-ohm resistors, insert the fuses and power up the boards. Then set your multimeter to 200mv, and attach the probes to Q22 and Q23 emitters to measure the correct I_q bias. Initially it should be zero or near zero. Now adjust the bias pot R36 clockwise slowly until you measure about 55mv. It may take several turns until you see the bias start to increase. Then wait a bit while the amp warms up, and then readjust to 55mv. It's ok if the bias drifts 2 or 3 millivolts. Then check the DC offset again to make sure that it's less than 50mv. If it all checks out the amps are now ready for some music. Good Job!
6. **If one or both of the 100-ohm resistors smoke in step 4, or the voltage is higher than 3.5 volts or less than 2.5 volts, post your observations on the DiyAudio.com builders thread and will start diagnosing the problem.**

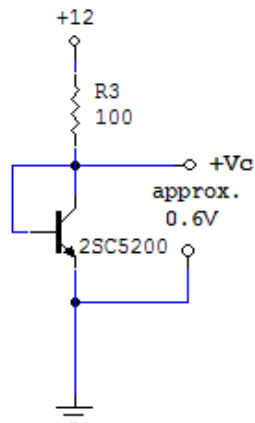
Transistor Matching Circuits:

Jfet Matching Circuit



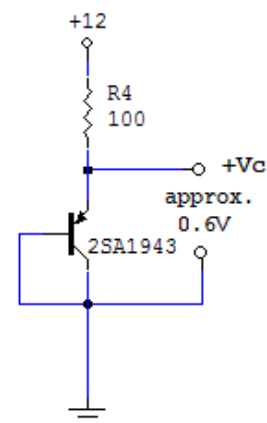
For Q1,Q2 (2SK170)

NPN Bjt Matching Circuit



For Q22,Q24 (2SC5200)

PNP Bjt Matching Circuit



For Q23,Q25 (2SA1943)

