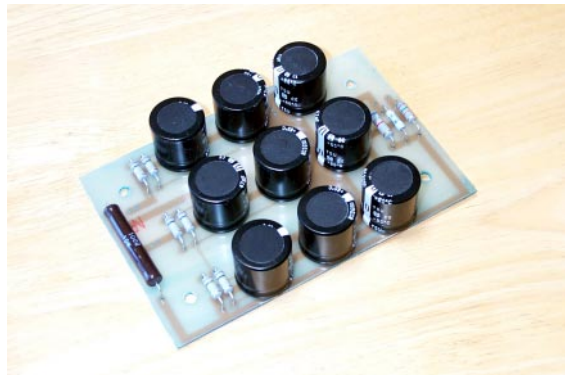




Sheldon Stokes
5600 Montañó Road NW
Albuquerque, NM 87120
(505) 897-1305
stokes@spinn.net
<http://www.quadesl.com>

Heath W-5M Capacitor Board



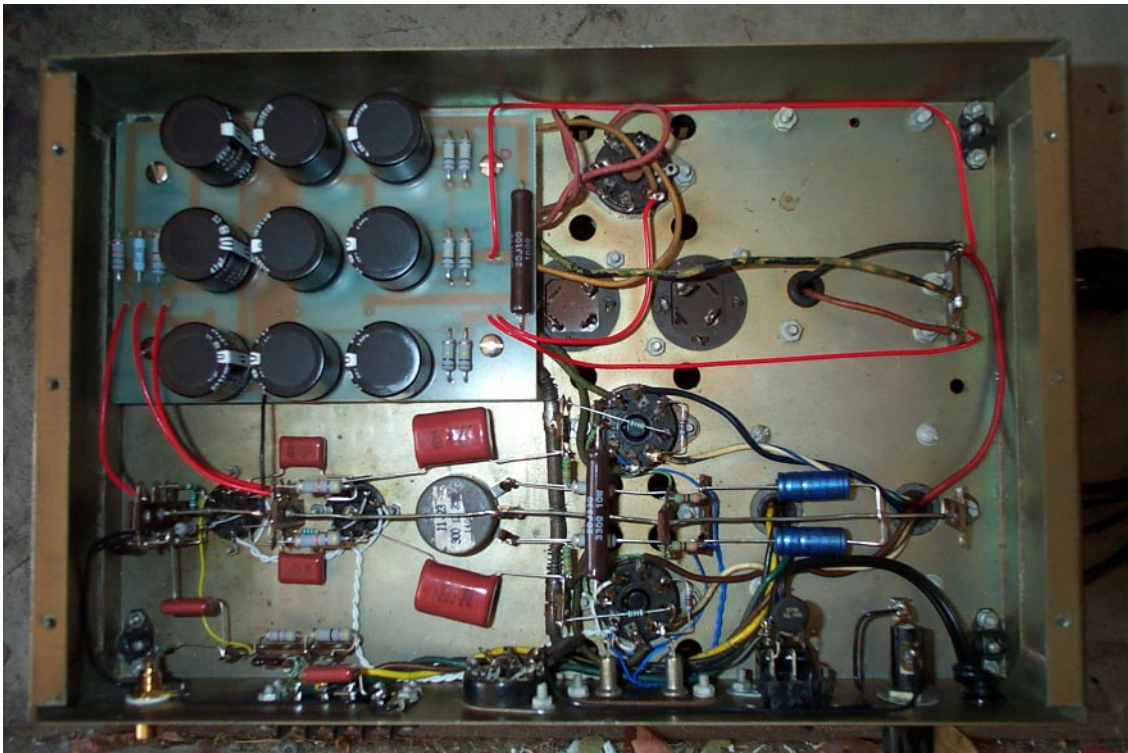
Heath W-5M Power Supply Capacitor Board

This board replaces the 2 large can capacitors and a handful of large axial lead electrolytic caps in the power supply section of the Heath W-5M amplifier. Exact replacements for the original can capacitors are impossible to find. The values needed can be built up with various replacement caps, but the installation of all the needed capacitors tends to make the amplifier messy. This circuit board uses readily available circuit board mounted capacitors. There is a large range of values and voltages available in this form factor. The board easily holds similar capacitor values to the original can caps with room to spare. The capacitor values can be increased substantially and still fit on the board and in the amplifier. As an added bonus, the cost of all the caps needed is almost the same as a single modern twist lock capacitor.

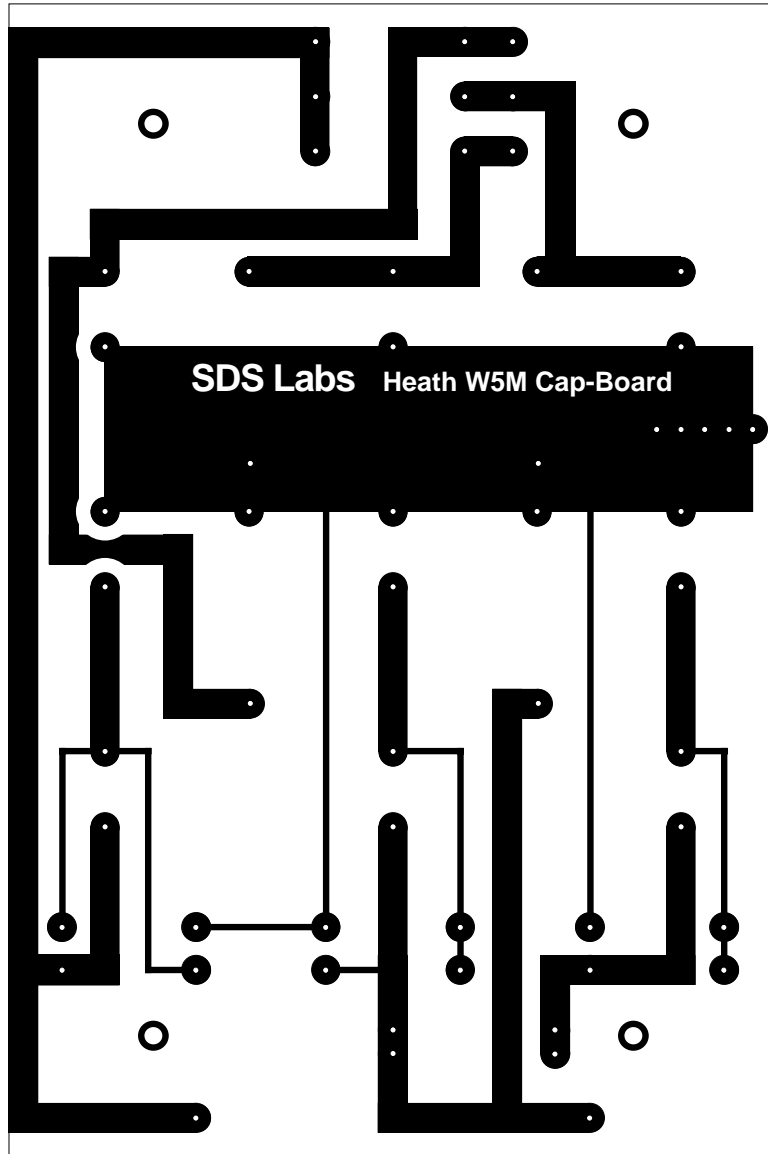
This board bolts to the 8-32 power transformer mounting bolts using 1/2" threaded standoffs (available from Digikey). The board is exactly 6 inches wide by 4 inches high.

Quantity	Value	Digikey No.	Cost
6	100 K Ω 1 Watt	100KW-1-ND	\$1.14
1	100 Ω 10 Watt	20J100-ND	\$1.05
1	6800 Ω 2 Watt	6.8KW-2-ND	\$0.23
1	15 K Ω 1 Watt	15KW-1-ND	\$0.19
1	22 K Ω 2 Watt	22KW-2-ND	\$0.23
9	47 μ F 450 Volt	P6438-ND	\$28.08
4	0.1 μ F 630 Volt (optional)	P3521-ND	\$4.72
		Total:	\$35.64

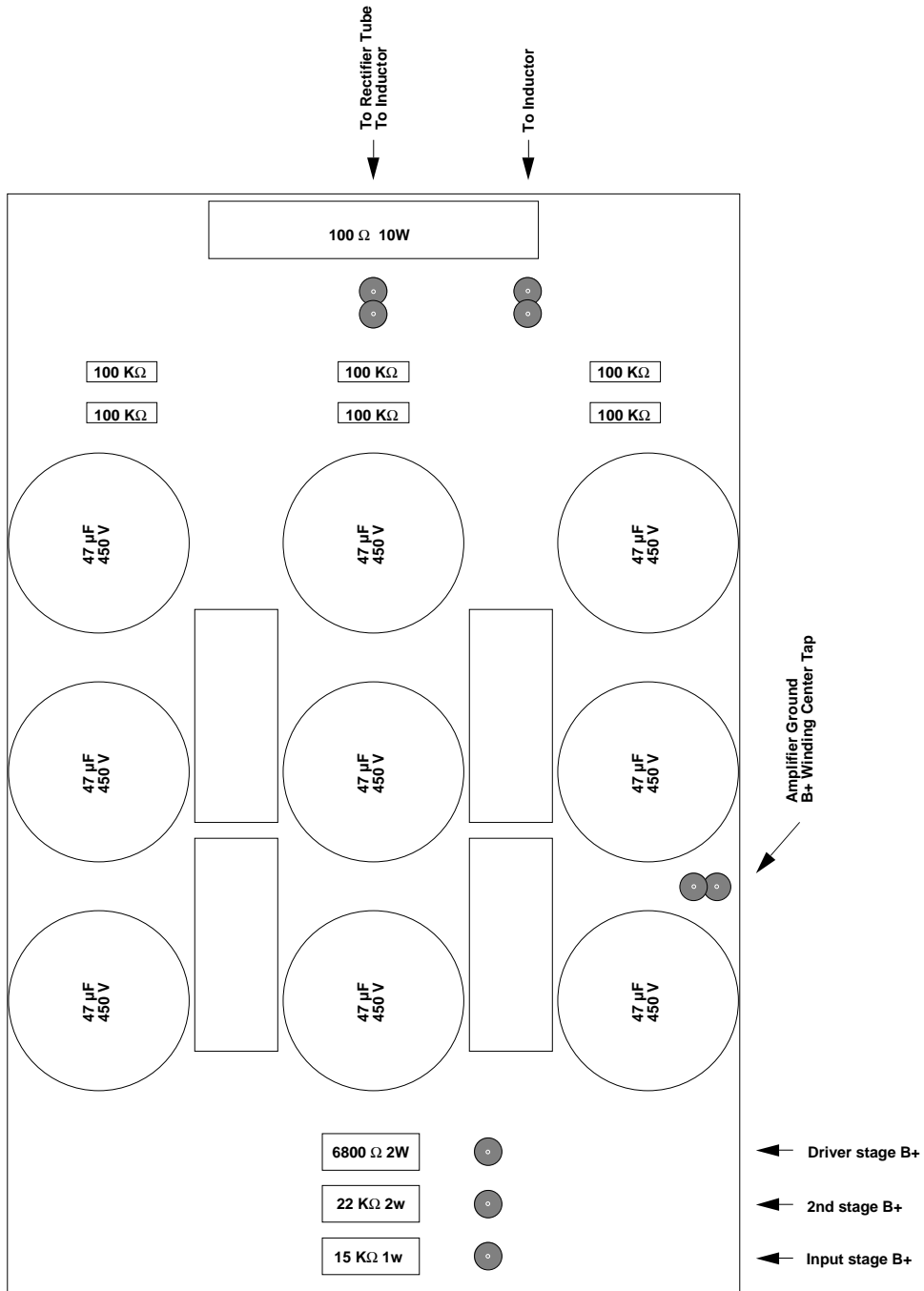
Capacitor Board Parts List (Nearly Stock Values)



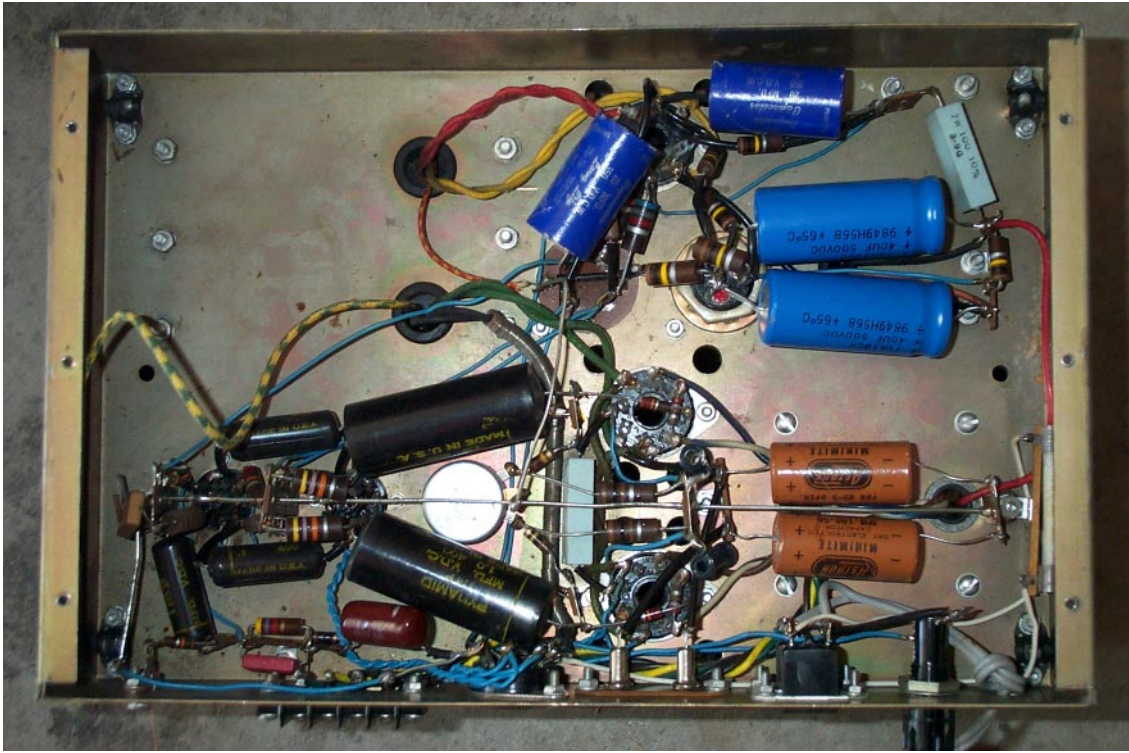
Capacitor Board Installed in Amplifier



Capacitor Board Layout



Capacitor Board Parts Placement



Stock Amplifier

Board Etching Tips

The artwork is printed onto transparency film from a laser printer, print it three times. Cut out two of the prints with about a quarter inch of clear space around the circuit board image. Then carefully tape these two copies to the uncut one after carefully aligning the traces of the overlay to the uncut sheet's traces. When finished, there should be three perfectly stacked copies. This increases the contrast of the final image. When a transparency is printed with a laser printer, there are usually holes in the black printed parts. And the blacks aren't all that black when it is held up to the light. Overlaying makes the blacks much more black, and gets rid of the holes. Now the artwork is ready to use. For double sided boards, the two sheets of artwork can be taped securely together on three sides after carefully aligning the traces on each side. this forms an envelope which the circuit board gets slid into. It's helpful to tape the board in place inside the envelope with a single piece of tape. This will prevent the board from shifting when it is flipped over to expose the second side.

This method uses GC positive sensitized boards and developer. The FR-4 fiberglass 1 Oz. grade board works very well (they can be gotten local electronics stores). The board emulsion is sensitive to UV light, A good source of UV to expose the board is a GE sunlamp. The sunlamp is hung so the bottom of the bulb is about 12" above the board. The exposure time is 9 minutes. With a yellow incandescent bug light-bulb on, pull the protective coating off the board and carefully align the artwork on top of the board. Then cover the artwork with a piece of glass to hold the artwork against the board (just like making a contact print in photography). Then turn the sun lamp on for 9 min. If a sunlamp is unavailable, the sun at noontime (on a clear day) can be used exposing the board for about 20 minutes.

The exposed board gets dumped into the developer which has been mixed up beforehand. The developer says to use a 1:9 concentration of developer to water, but a 1:5 mix can be used, which works faster and can yield slightly better results. However the timing is more tricky, so it is not recommended for the first time. Submerge the board into the developer (A photography developer tray works very well), and rock the solution back and forth over the board. The exposed parts will start to dissolve. The emulsion is green and it will wash away exposing the copper underneath. This is the tricky part. The board must be removed when all the emulsion is off the exposed areas. If the board is removed too soon, the emulsion won't be completely dissolved off the exposed areas and it won't etch, if the board is in the developer too long all the emulsion dissolves and all that is left is a bare board. With the 1:9 solution this time window is about a minute, with a 1:5 solution it's about 20 seconds. The board is removed from the developer and washed off with room temperature water, then scrape at an exposed area and see if there is any emulsion left there. if there is, place the board back in the developer for a few seconds. Repeat this as necessary until the exposed areas clear. With a little practice, it's pretty obvious when it's time to pull the board out. Do all the developing using the yellow bug light. When the board is done, wash it off and let it dry. Be careful of the emulsion, it's easily scratched, especially when fresh from the developer.

Next, drop the board into an etching solution. Ferric Chloride is available from the same electronic outlets where the GC boards and developer are purchased or from Radio Shack. Ferric Chloride is a nasty smelling, iodine looking, serious staining stuff. Pour out the developer from the tray, wash it out and add the etchant. Then put the board into the etchant and rock gently back and forth for about a half hour or so, until all the exposed areas are clear. Then remove the board and wash it clean. The emulsion can then be removed with acetone or alcohol.

Then all the holes need to be drilled in the board. A Dremel moto tool works well for drilling the small holes, a small drill press would also work.