W0EB/W2CTX DSP Audio Filter Construction Manual V3.02.1

W0EB's build of the filter with the 1.8" Color touch Screen showing the initial "Splash" screen and a bezel made from PC board material .

This "Stand-Alone" DSP Audio Filter is loosely based on one originally called "The GI1MIC $20 Dollar DSP Audio Filter" by Gareth McClean, GI1MIC. It uses the considerable audio processing capabilities of the relatively inexpensive little PJRC Teensy 3.2 MPU, a 1.8" Color TFT Display with Resistive Touch Screen and a small audio output amplifier to

drive headphones or a larger amplified speaker. The main PCB was designed by Jim Sheldon, W0EB, the original software was reworked and greatly enhanced by Ron Pfeiffer, W2CTX and the PCB layout was engineered by Jim Giammanco, N5IB. The only controls are the 1.8" resistive touch screen, a rotary encoder with pushbutton switch and the on/off/volume control.

The Triumvirate Skonk Works (TSW) is hoping to offer this filter in partial kit form. All parts would be included EXCEPT the Teensy 3.2, case and parts external to the PCB (encoder, amplifier, audio and power jacks). Also not included would be the internal cabling to the controls, jacks and display. An adapter to make connection to the display and breakout boards for the encoder and audio jacks WILL be included.

The board is designed with input/output headers for ease of connection, only one SOIC14 SMD IC, one SOT23 (2N3904 or equivalent) transistor and a few 0805 sized SMD capacitors and resistors. The parts are also included to provide a socket for the Teensy 3.2 for easy replacement if ever necessary.

CONSTRUCTION:

Let's get the hardest part out of the way first. U2 is an SOIC14 size 74HC14 Hex Inverter (only 2 inverters are used here) with fairly wide lead spacing so careful builders should have little difficulty installing it. Be sure to orient pin 1 properly as shown on the board diagram in the appendix.

It would help if you were to use some ChipQuik "Tack Flux" to flux the PCB's pads for U2. There is usually enough solder actually on the pads from the manufacturers "tinning" process that very little or no solder will be

required if the pads are fluxed prior to soldering. Using a fine point soldering iron with around 750 degrees F heat, carefully place U2 over its pads and tack pin 1. Check to make sure the other pins are aligned with their pads and tack pin 7 or any pin on the other side. Once you have the orientation and positioning correct, carefully tack all the rest of the pins. When U2 is tacked in place, you should, using small diameter solder, carefully solder each pin to its pad prior to the resistance check. The pads are spaced wide enough this should be relatively easy. It has been discovered that using the “flux and tack” method doesn’t always achieve good connections. Now, take your DMM and on the DIODE TEST or low resistance range, check continuity between each pin and the other end of its trace. Use minimal pressure on the IC pin to make sure you aren’t pressing it down, making contact in case it ISN’T soldered. Pressing too hard on the pin may make it contact the pad even though it wasn’t properly soldered. Once you are satisfied all pins of the IC are firmly soldered to their proper pads, using commercial flux remover or Isopropyl alcohol, carefully clean any flux from around the chip. Give it a final visual inspection before proceeding with the rest of the construction.

Q1, the 2N3904 (labeled 1A) transistor. Place a tiny amount of solder on the collector pad for Q1. Holding the transistor in place with a tweezers, carefully tack solder the collector pin. Now make sure the base and emitter pins are roughly centered over their respective pads and solder these using a fine point iron @ 750F and small diameter solder. Check your connections.

C5, the .01uF 0805 ceramic capacitor. Carefully solder C5 in place now so you don’t get it mixed up with the .1uF 0805 ceramic capacitors installed next.

C1, C4, C7, C8, C10, C11 and C12. Solder the .1uF 0805 ceramic capacitors in place at their respective locations. Check your connections.

C2, C3. 10uF 1206 tantalum electrolytic capacitors. THE BANDED END IS THE POSITIVE END of these capacitors, make sure you orient them so the _+ end is over the + marked pad on the PCB. Carefully solder them in place and check your connections.

C6 and C9. These may be either 10uF 1206 tantalum electrolytic capacitors OR they may be 10uF 16V 1206 Non Polarized ceramic capacitors. If the tantalums are supplied, see C2 and C3 for polarization. For the polarized capacitors, make sure the banded end is over the + marked pad on the PCB. Solder them in place and check your connections.

R1. Carefully solder a 470K 1/8W 0805 SMD resistor at R1.

R2. Carefully solder a 3.3K 1/8W 0805 SMD resistor at R2

R3. R3 originally was a 3.3K 1/8W 0805 SMD resistor but it was found the 74HC14's 2nd Schmitt Trigger inverter was not triggering reliably with this resistor in place so it was decided to eliminate R3. Solder a small wire jumper (or bridge the pads with solder) across the pads for R3 and make sure no surrounding traces other than the one to C7 are shorted to either pad.

P1 through P8. Solder the proper headers in each of the areas silkscreened on the PCB. P6 and P7 are shown oriented WRONG on the v3.20 PCB silkscreen. The polarizing post (thicker, short white line on the silkscreen) should be oriented to the left of the connector, NOT on the right

as shown on the v3.20 silkscreen. Convention says Pin 1 should be ground or negative and these two are incorrectly shown with Pin1 as Positive.

As you install each header, solder 1 pin first and, while re-heating that pin, adjust the headers so the pins are vertical and the plastic bases are flush and square with the PCB. Once you are satisfied with the orientation of that header, solder the remaining pins. Do this will all headers and carefully inspect your work to ensure there are no shorts or unsoldered pins.

U1. Next the socket must be made for U1. This is formed from two 14 pin female socket strips and one 5 pin female socket strip.

The easiest way to make this socket is to solder the 14 pins on each side of your Teensy 3.2 to the Teensy itself (or order the Teensy 3.2 with pins). If you order it with pins, you will still have to solder a 5 pin male header across the end away from the USB connector on the Teensy 3.2 as some of these pins are used for the filter.

Once you have all the pins soldered properly on your Teensy 3.2, plug each side into one of the 14 pin female sockets and the 5 pins across the back into a 5 pin female socket. Next, carefully insert the sockets into the PCB with the USB connector on the end shown by the PCB silkscreen. Make sure all pins go into the respective holes in the PCB and none are bent over. Turn the board over and CAREFULLY solder all socket pins on the underside of the board. Inspect your work to make sure there are no shorted pins and no unsoldered pins. Carefully remove the Teensy 3.2 from its socket and set it aside for now.

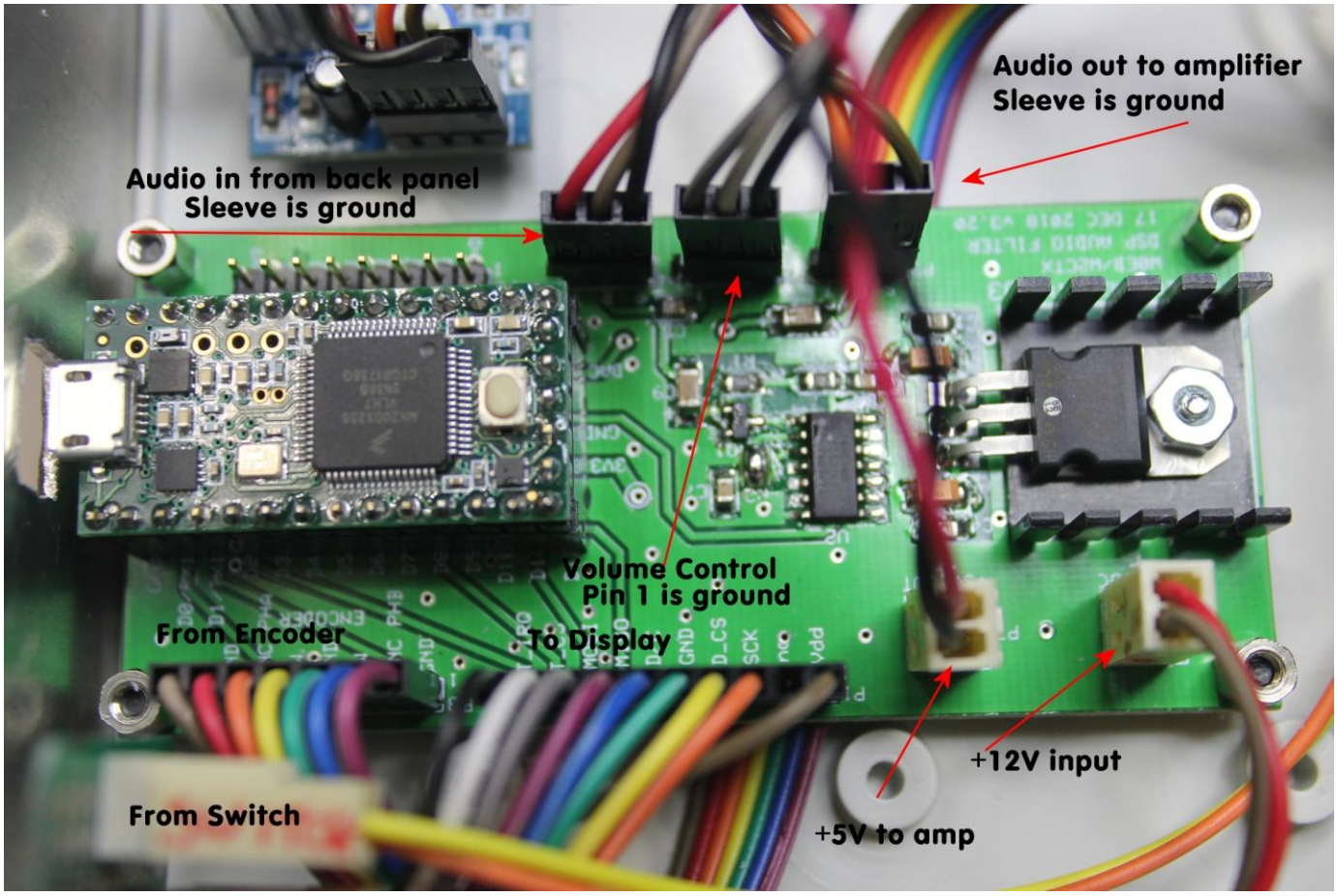
U3. U3 is mounted on the top side of the board along with all the other components. Carefully form the leads on the 7805 regulator so they fit in the proper holes and the back of the regulator is flat on the PCB with the hole in the tab centered over the hole in the PCB. Make sure the hole in the board and the hole in the 7805 tab line up. Secure the 7805 to the PCB with a 4-40 by 3/8" screw and nut with the head of the screw to the underside of the PCB and the nut on the top side of the 7805 and tighten the screw. Finally, solder the 3 terminals on the underside, check your connections and trim the protruding pins. (The 7805 does NOT require a heat sink so none is included – if you feel you need one you will have to supply it yourself.

If you haven't already done so, using the Arduino IDE;s (V1.8.5 or later) "TeensyDuino" interface to your computer, compile and upload the latest version of the TSW filter software to the Teensy 3.2 which will be available in the "Files" section of the TSW Website, www.w0eb.com.

Now, following the information on the Teensy pinout sheet supplied with it from PJRC, cut the tiny jumper on the underside of the Teensy 3.2 labeled "Cut to separate VIN from VUSB, if using a battery charger or external power for USB Device mode." If you do not cut this jumper, the voltage from the internal power supply (7805) would conflict with the voltage supplied by the USB port during programming/reprogramming and could possibly destroy the Teensy's circuitry. This HAS happened so best not ignore this step.

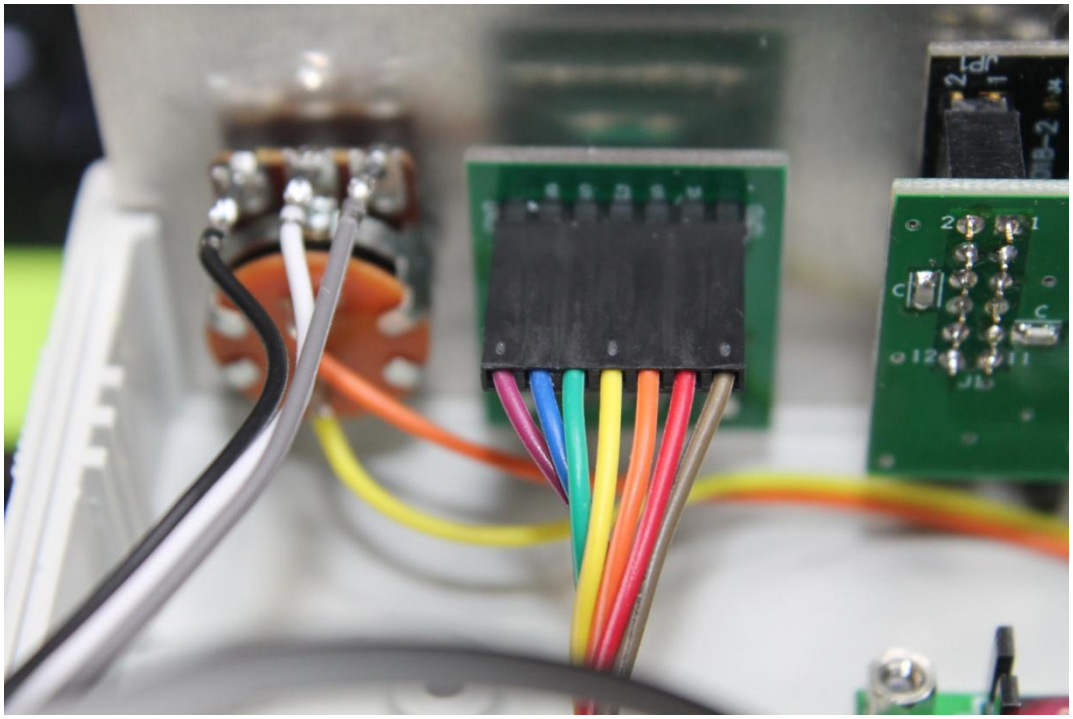
That completes construction of the filter PCB itself. The next section deals with the 1.8" touch screen display, audio input and output connections and mounting it all in a case of your choosing.

Interconnect information: The following images are really for reference only. Each installation will be different – follow the schematic and board layout disgrams. Observe polarity on all connections.

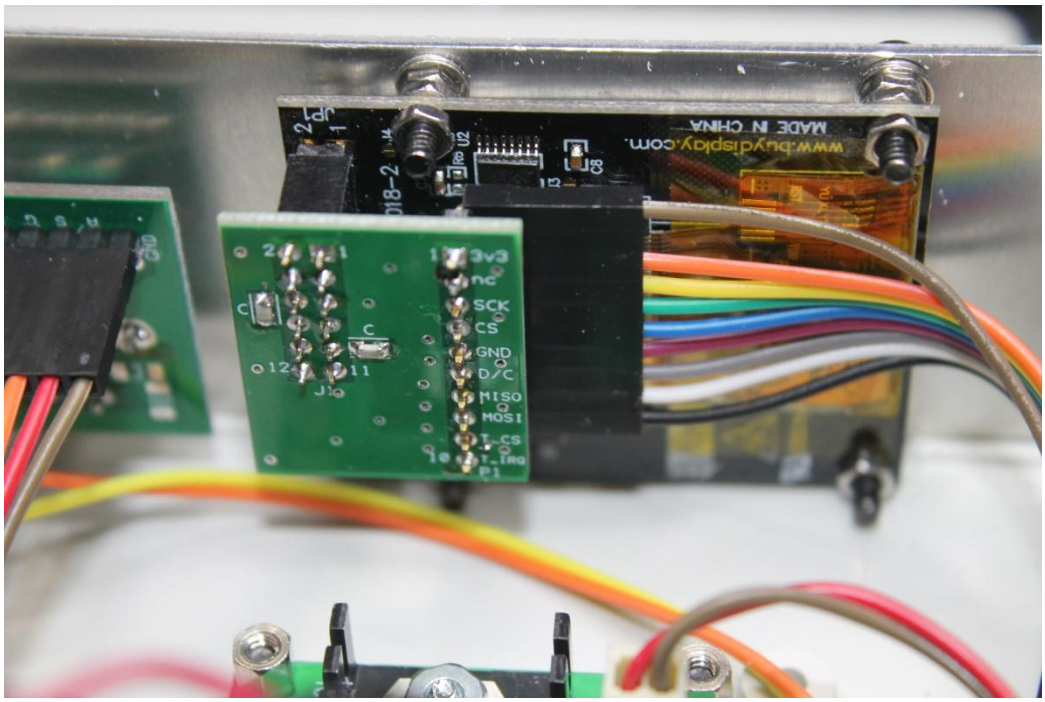


I used DuPont female connectors and Molex style connectors in the initial build, but each person is free to use whatever they choose. You could even leave out the pin headers and solder the wires direct to the PC board. Connectors though, make removal and replacement of the various controls much easier.

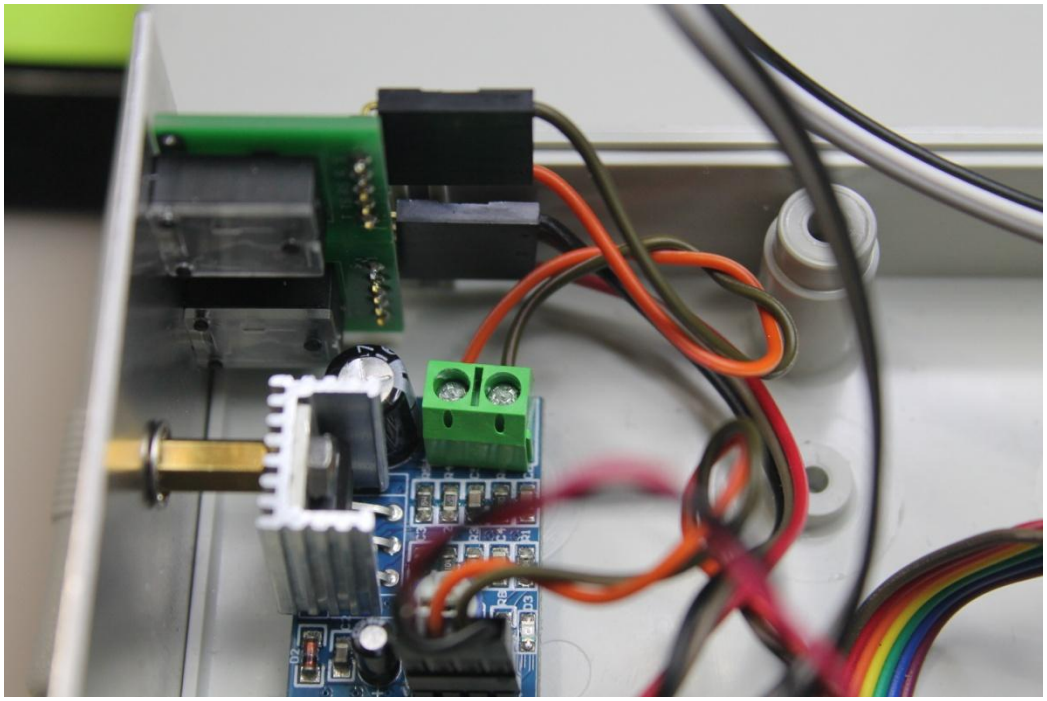
Encoder breakout and volume control/on-off switch connections



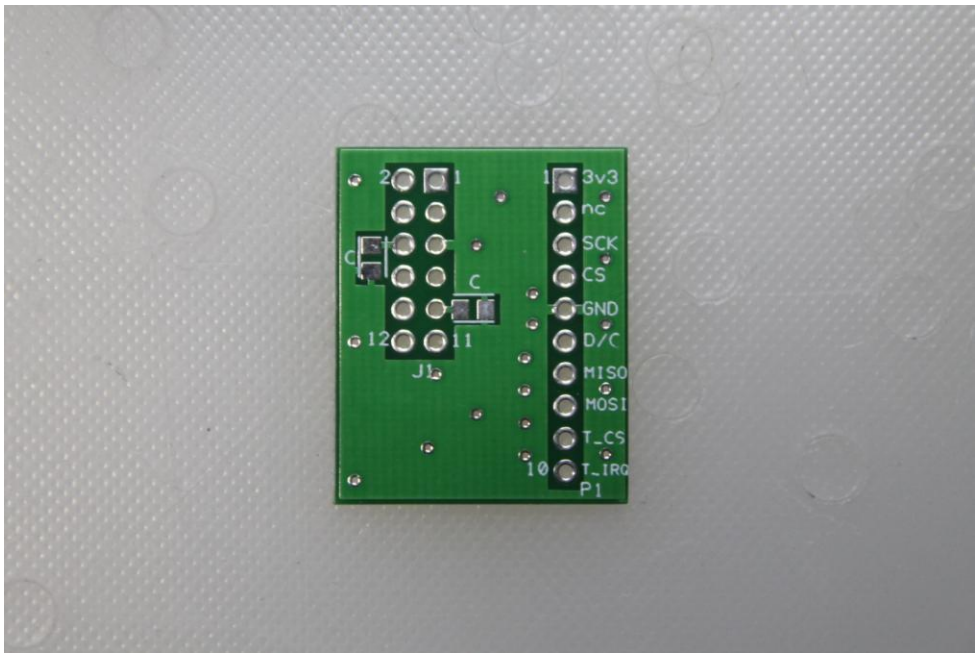
Connections to the encoder and display using small PC adapter cards which will be supplied with any kits TSW sells & delivers



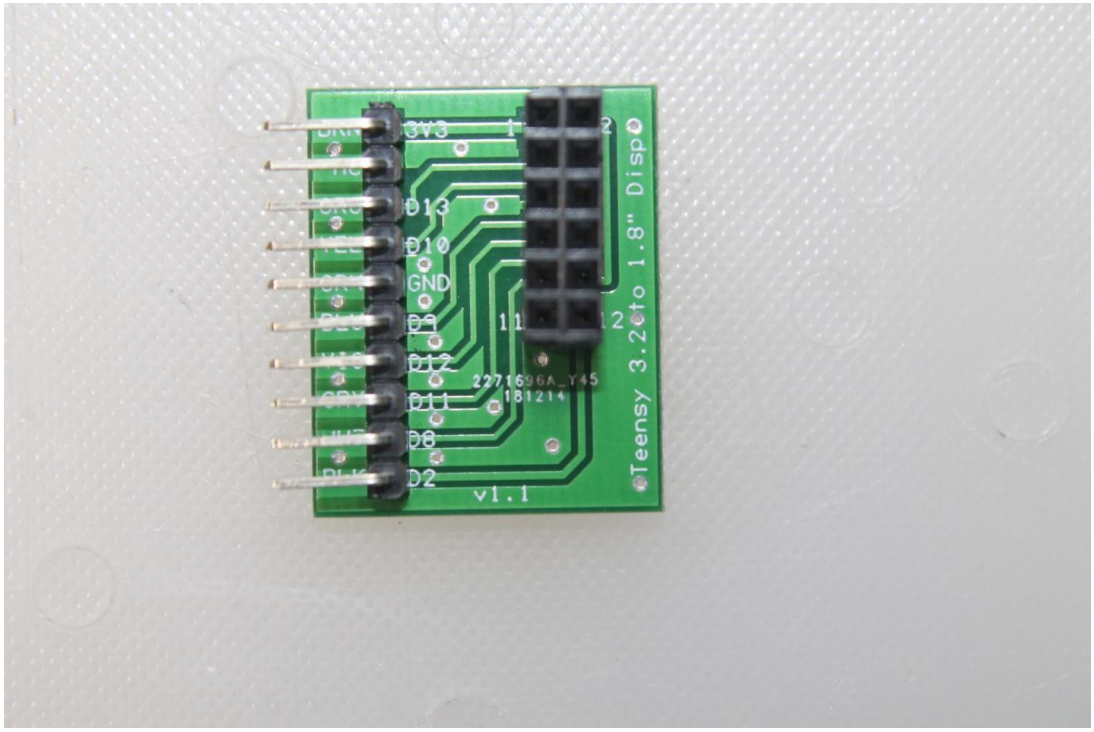
Audio I/O connections using 2 of TSW's Stereo Jack breakouts and the little eBay audio amplifier (less than $5) that works nicely. The Teensy's ADC output is NOT sufficient to drive a speaker (or even headphones for that matter)



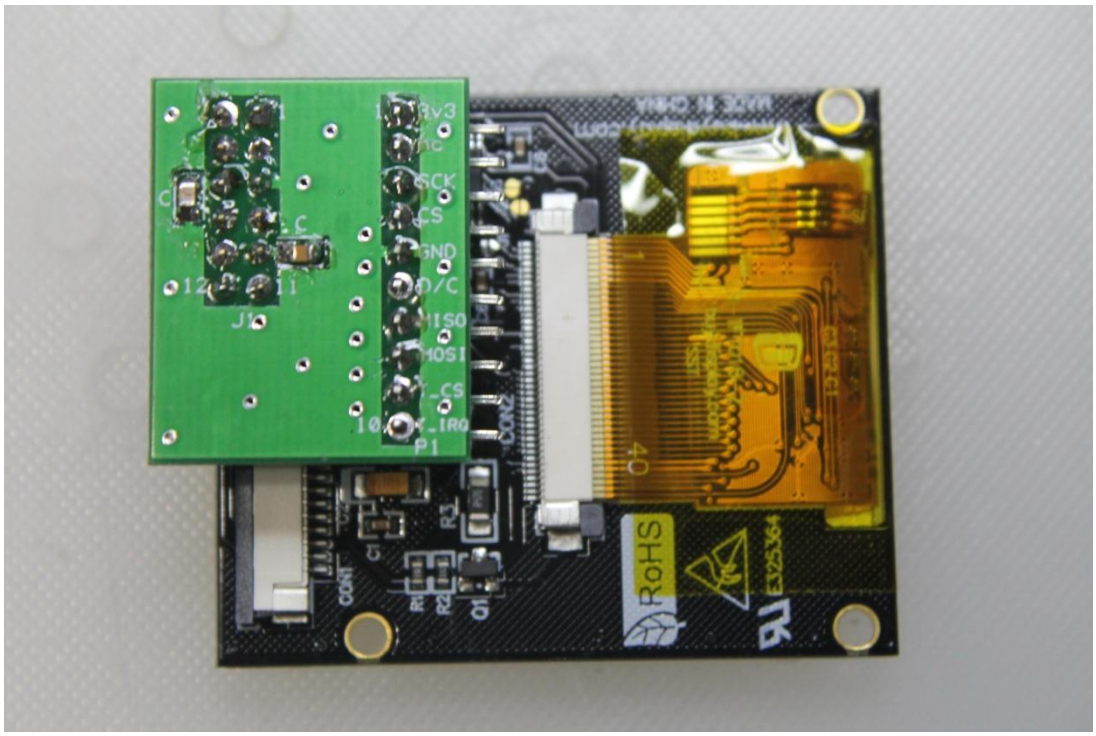
Display connector adapter PCB (supplied with the kit)



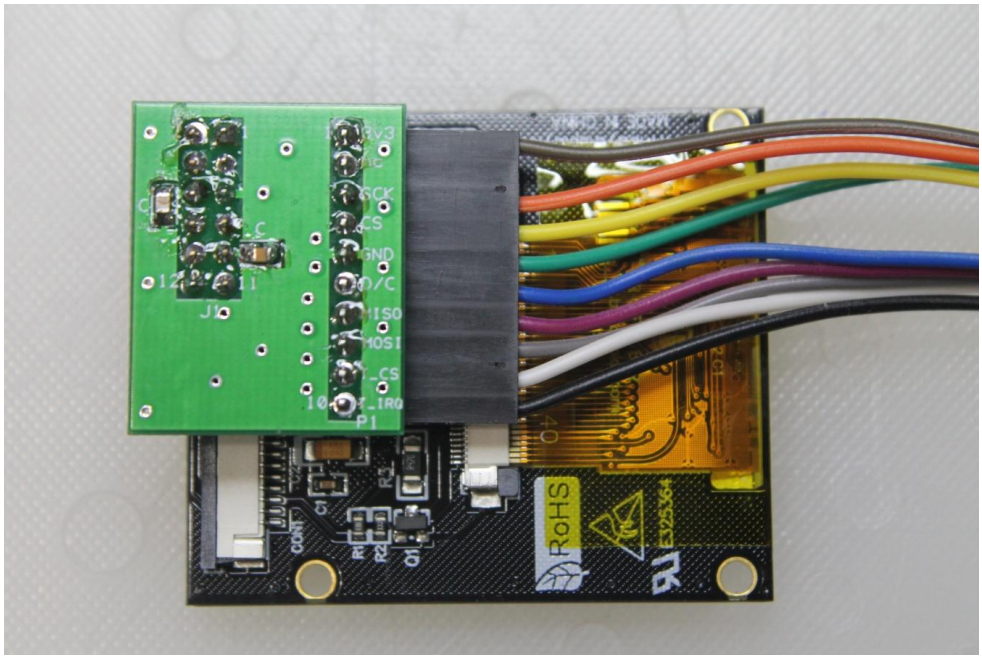
User will need to supply the dual row 12 pin jack and the right angle 10 pin connector as well as the two 47pF caps shown in the next picture.

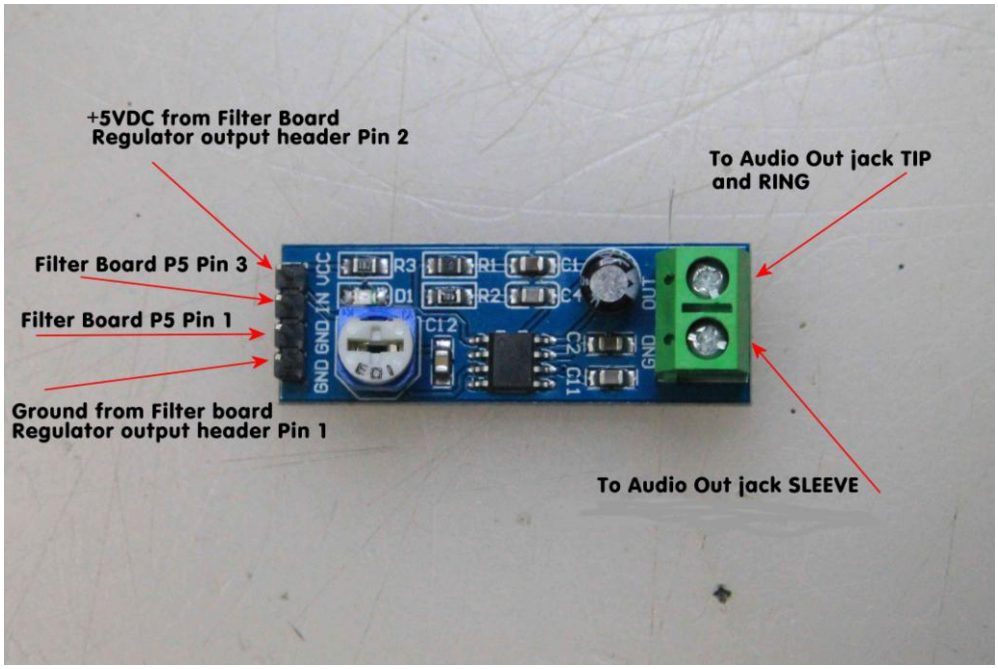


Two 47picofarad 0805 SMD capacitors need to be installed at C1 and C2.



10 pin DuPont female makes connecting display to the filter a breeze.





Supplied Audio Amplifier and connection diagram.

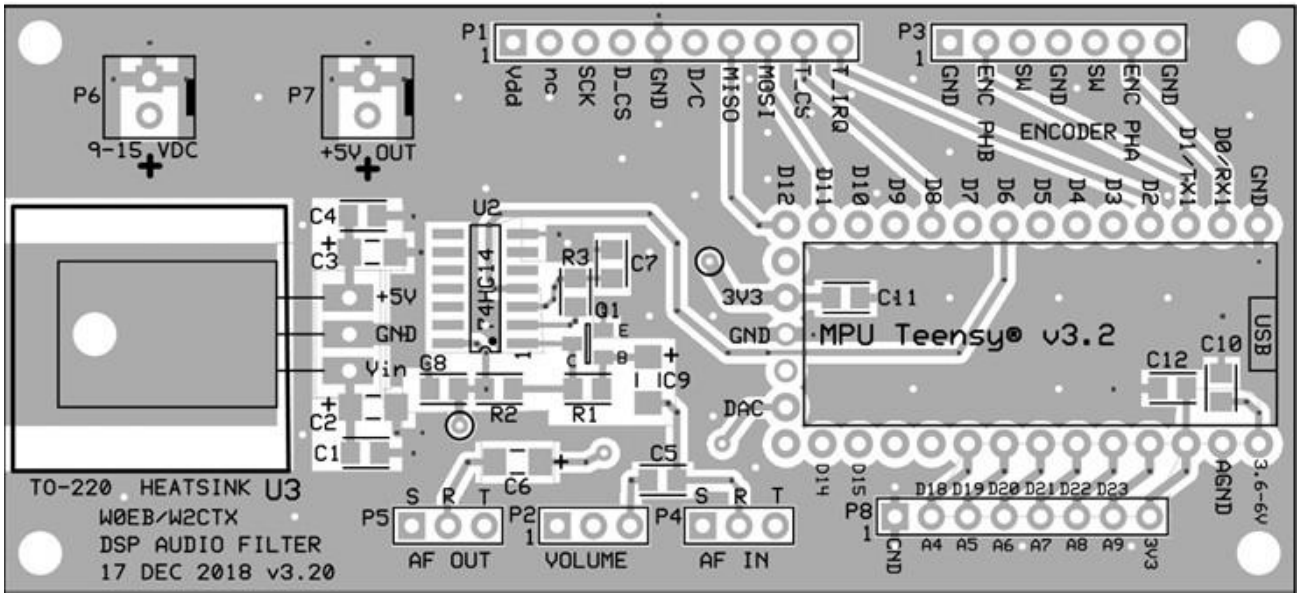
DSP filter design & interface software, GI1MIC, W2CTX.

PCB circuit design, interface hardware, schematic, W0EB.

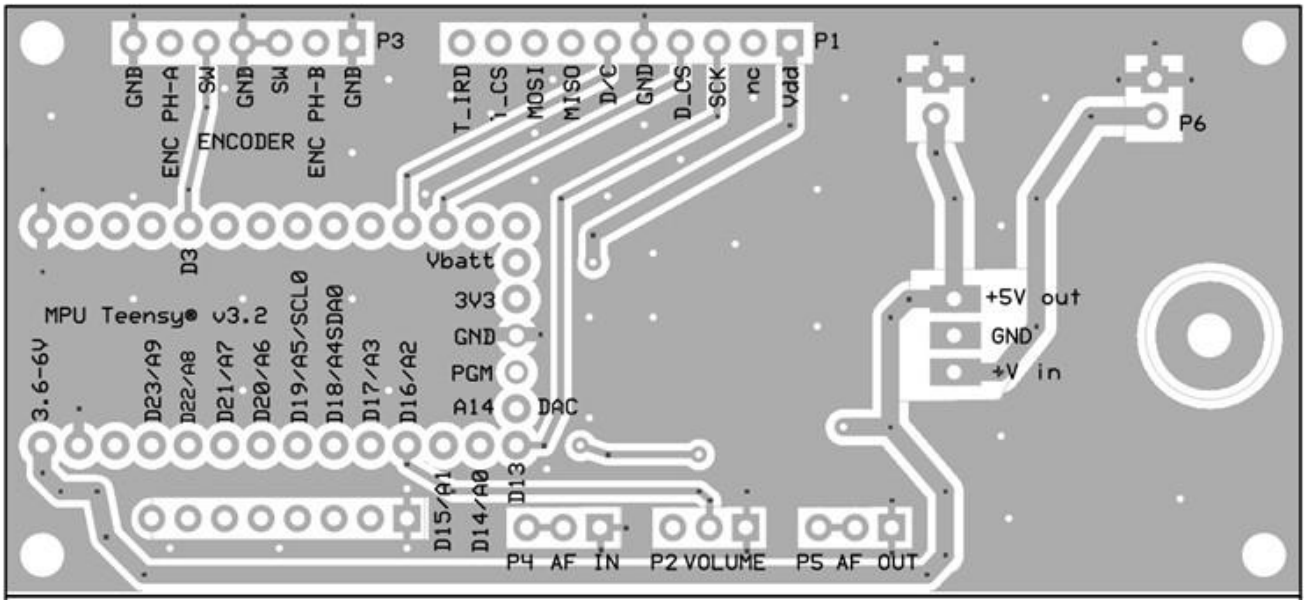
PCB layout & procurement, N5IB.

Appendix A: PCB TRACE PATTERNS

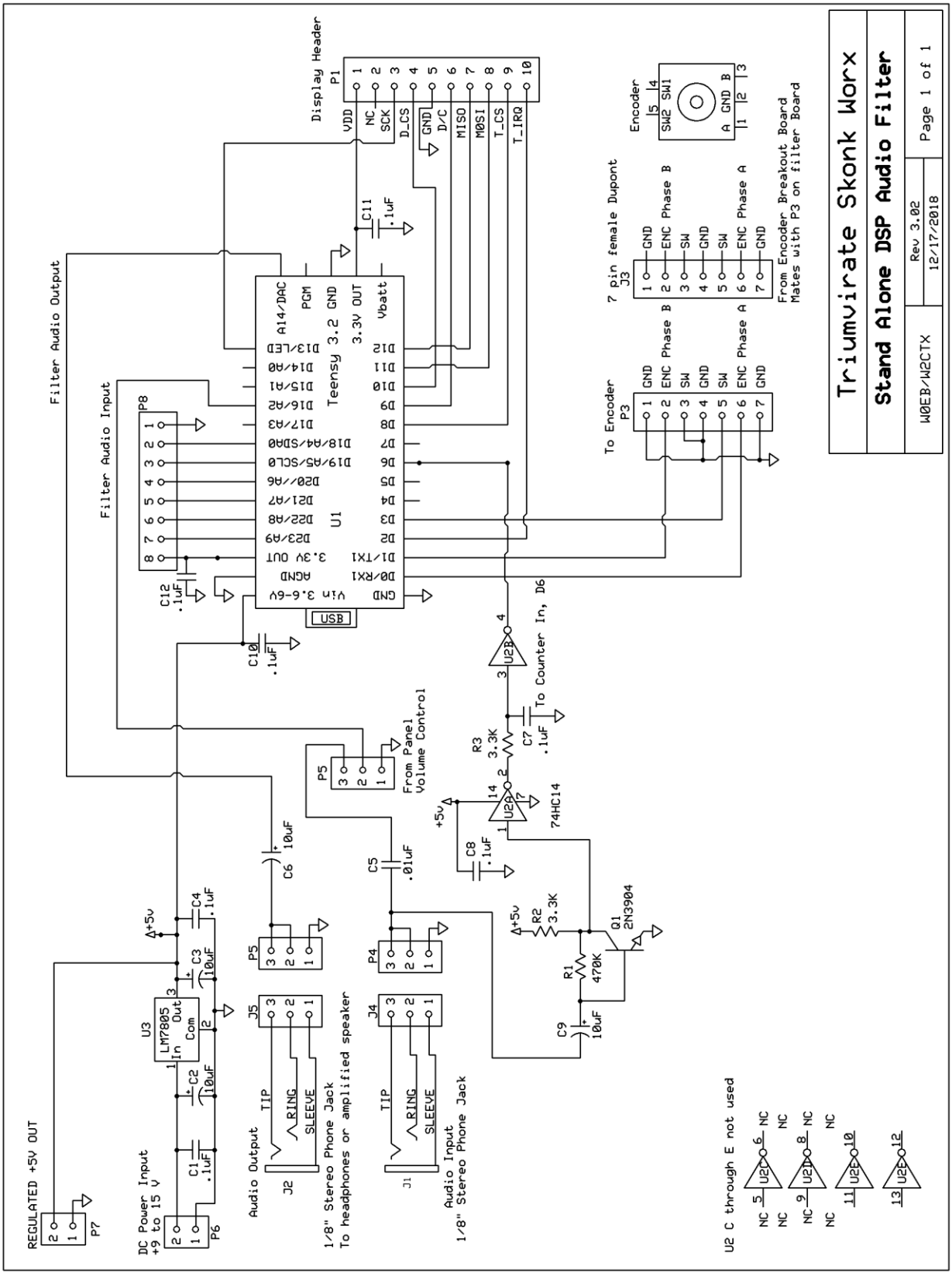
PCB Top traces



PCB Bottom Traces



Appendix B: CIRCUIT SCHEMATIC



Appendix C: BILL OF MATERIALS

Output Amplifier	5V-12V LM386 Audio Amplifier module from Amazon.com
C1, C4, C7, C8, C10, C11, C12	1uF, 50V SMD 0805 capacitor Tayda Electronics SKU A-3511
C2, C3	10uF 25V SMD 1206 Tantalum capacitor Mouser 581-TCJA106M025R0150
C5	. 01uF (10nF) 50V SMD 0805 capacitor Tayda Electronics SKU A-3507
C6, C9	10uF 16V NP 1206 Ceramic capacitor Digi-Key 76-6641-6-ND (can use 10uF 25V SMD 1206 Tantalum capacitor the same as C2 and C3.
P1	10 pin male header Cut from 40 pin header strips. Tayda Electronics SKU A-197
P2, P4, P5	3 pin male header (see P1)
P3	7 pin male header (see P1)
P6, P7	2 pin polarized Molex style male header Tayda Electronics SKU A-804
P8	8 pin male header (See P1)
Q1	MMBT3904-TP SOT-23 SMD transistor Digi-Key MMBT3904TPMSCT-ND

R1	470K 1/8W SMD 0805 resistor Tayda Electronics SKU A-3129
R2	3.3K 1/8W SMD 0805 resistor Tayda Electronics SKU A-3077
R3	Zero Ohms, Jumper across pads.
SKT1, SKT2	14 pin female (parts of socket for U1) Tayda Electronics SKU A-1308 16 pin female – cut 2 pins from each.
SKT3	5 pin female (part of socket for U1) Tayda Electronics SKU A-1304
U1	PJRC Teensy 3.2 (User supplied)
U2	74HC14D SOIC-14 SMD IC (Hex Inverter) Digi-Key 1727-2783-1-ND
U3	7805 5V Regulator TO-220 Tayda Electronics SKU A-179
PCB1, 2, 3 & 4	Filter PC Board, display adapter, encoder & and stereo jack breakout boards.
Volume Control w/switch	Tayda SKU A-4184
Encoder	Bourns PEC11R-4015K-S0024 Digi-Key PEC11R-4015K-S0024-ND
Stereo input/output jacks	Tayda Electronics SKU A-069
Knobs (2)	Tayda Electronics SKU A-324

Appendix C (cont): USER SUPPLIED PARTS

Power Jack	User supplied
Display	User supplied. www.buydisplay.com ER-TFTM018-2 Order with Pin Header 4-wire SPI interface and 3.3V power options. (Adapter PCB will be supplied with the kit of parts.)
Hardware	User supplied 4-40 or 3mm standoffs and screws to fit the PCB mounting holes and 4-40 to fit the display. Use 3/16" (across the flats) nuts as spacers between the panel and the display PCB to keep from shorting the interface plug to the panel. Insulating tape on the underside of the display PCB pin header pads would be a good idea.
Parts suppliers	www.digikeycom www.mouser.com www.taydaelectronics.com www.buydisplay.com www.amazon.com