Panadapter Interface

Rights & Permission

Version

This document is for printed circuit board version 0.1a

Overview & Features:

The Panadapter Interface circuit board is designed to provide an internal connection from a transceiver's first IF to an external SDR receiver so that along with several freely available software packages a panadapter can be realized for the transceiver.

- Works with any radio that has a first IF in the 50 to 70 MHz frequency range
- Won't adversely impact receive sensitivity
- Low power consumption typically 5 to 6 mA
- Can be configured to work with a wide range of supply voltages from 5 to 14 V.
- Stable with or without a SDR receiver attached to the interface output.
- The PCB back side is a ground plane only enabling solder mounting to most any ground shield within a transceiver.

Circuit Description

Component designations refer to the schematic and PCV layout diagrams for this version.

High Impedance Input Buffer

Q1, the input buffer, has its emitter biased at close to 1.5 volts by input voltage divider R1 and R2. R3 sets the emitter current to about 1.5 mA, sufficient to drive the 50ohm filter. C10 and C2 are input and output DC bias blocks. C1, L6, and C11 provide supply filtering for the input buffer.

Filter

The filter section is made up of L1 to L5 and C3 to C5. It's a nodal inductor-coupled third order band pass with a center frequency 67MHz and a bandwidth of about 15MHz. It was chosen as a compromise between complexity, its impact on PCB size, and realizable commercially available components. The center frequency was chosen to make sure the radio's first IF at 68.33 MHz is near the top end of the passband.

Post Filter Gain & Output Buffer

Q2 provides post filter gain of about 6 to 8 dB. R4 and R5 set the emitter bias voltage to about 1.5V, R7 along with AC bypass capacitor C7 sets the emitter current to about 2mA. R6 and R9 set Q2's gain along with the feedback network C8 and R10 which also serves to set the stage input impedance to match the filter's 50 ohm output. Q3 is the output buffer voltage follower. Its emitter current is set to about 1.5 mA by R8. C9 provides output coupling to the DVB dongle. L7 and C12 provide supply decoupling for the amplifier and driver stages.

Assembly Guidelines

Assembly Equipment you'll need

- Surface Mount Device (SMD) soldering kit
    - Temperature controlled soldering iron with fine tip. Insure the tip is grounded!
    - Small vise to hold the PCB
    - Liquid no clean solder flux

    - SMD Tweezers
    - Isopropanol 99%
    - Cotton swabs
    - Solder wick and solder sucker
- Electrostatic Discharge (ESD) protection - mat - wrist strap
- Needle nose pliers
- Wire cutters
- Voltmeter with leads suitable for probing surface mount components.
- Five volt regulated power supply capable of delivering 0.1A.
- X-Acto knife or utility knife with snap-off blades

Before You Begin

- Check to insure all the components needed are at hand.
- Print out this document. To help locate components on the printed circuit board, print out the circuit schematic, and the PCB top component layouts.
- Watch the part sizes. Most components are 0603, however the filter inductors may vary and the PCB is capable of handling a variety of sizes.

Assembly Tips

1. Use two pair of tweezers, one for pulling up the hold down tape on the parts sheet and opening the SMD part carrier, and the other for mounting components on the PCB. This prevents the tape glue residue from getting on the tweezers used for SMD mounting work.
2. If adhesive does get onto the tweezers' tips, parts will stick to them. If this happens wet a cotton swab with isopropyl alcohol. Spread the tweezers tips to clamp around the swab and draw them across the swab to clean the tips.
3. If there is more than one part of a value, remove the part from the strip then return the remaining parts to the parts sheet in their original location and cover them with the strip of tape. The tape used to hold the components to the sheet is special Scotch removable tape so that the parts can be returned to the strip and tacked down or repositioned several times.
4. Orient all resistors in the same direction so that they can more easily be read and identified after installation. Use the same orientation as is used on the PCB silk screen.
5. As you go through each assembly section check off components in the square box beside the component identifier as you install devices.
6. The ground plane is typically a good heatsink. If one side of a component is to be soldered to ground, solder the non-grounded side first. If you have a temperature controlled soldering iron set the tip temperature a little higher for the ground plane solder joint. Typically 350 to 370C will work better than 300.

PCB Assembly

All components are mounted on the top side of the printed circuit board. Use the schematic and PCB top layout as an assembly guides.

1. Install all top side components

- ☐ C1 1nF
- ☐ Q1 BFR520
- ☐ R3 1k
- ☐ R1 12k
- ☐ R2 10k
- ☐ C10 1nF
- ☐ C2 1nF
- ☐ C3 200pF
- ☐ C4 200pf
- ☐ C5 200pF
- ☐ R5 10k
- ☐ C8 1nF

- ☐ C6 1nF
- ☐ R4 12k
- ☐ C11 1uF 6.3V
- ☐ R10 330R
- ☐ R6 68R
- ☐ R7 1k0
- ☐ C7 2.2nF
- ☐ C9 1nF
- ☐ R8 1k6
- ☐ R9 1k2
- ☐ C12 1nF
- ☐ Q3 BFR520
- ☐ L7 4.7uH
- ☐ L1 33nH
- ☐ L3 33nH
- ☐ L2 43nH
- ☐ L4 150nh
- ☐ L5 150nH

Functional Verification Test 1

This test verifies that the interface board transistors are biased correctly.

a. Solder a supply wire to P3, the 5V on Rx solder pad.
b. Power the interface board by connecting the supply wire to a 5.00V power supply, and a return wire to ground
c. Turn on the power supply and measure the voltage at the emitter of Q1. It should be 1.5 V +/- 300 millivolts
    - ☐ Emitter Q1 voltage. _____ V.
d. Measure the voltage at the emitter of Q2. It should be 1.5 V +/- 300 millivolts
    - ☐ Emitter Q2 voltage. _____ V.
e. Measure the voltage at the emitter of Q3. It should be 2.6 V +/- 300 millivolts
    - ☐ Emitter Q3 voltage. _____ V.

Functional Verification Test 2

If an RF signal generator and power meter are available measure the gain of the interface at 68.33 MHz.

a. Solder RF coax cable pigtails with suitable RF connectors to the PCB's RF In and RF Out ports.
b. Connect the RF generator, power meter and power supply leads to the DUT.
c. Set the RF generator to 68.33 MHz with an output power of -40 dBm.
d. Measure the RF at the output, it should be -33 dBm +/ 3dB.
    - ☐ Power at panadapter interface output _____ dBm.

Final Assembly – General Guidelines

Final assembly depends of course on the radio.
Most of the solder mask has been removed from the bottom of the PCB so the board can be soldered to just about any ground metal shield within the transceiver. If soldering is the choice, do perform the functional verification tests before soldering the PCB in place.

Many DVB-T dongles use MCX female connectors at the antenna port.
This is a compact and fairly reliable RF connector. The bulkhead female version of this connector is recommended for use on the transceiver as it's fairly small compared to other connectors so can be put on the transceivers rear panel where other RF connectors wouldn't fit. Also the MCX female bulkhead doesn't leave a protruding connector thread on the transceiver that could be damaged during transport. A short male to male MCX jumper cable can then be used between the transceiver and the dongle.

If the first IF is close to the PCB RF input a small jumper wire can be used instead of a coax cable. This was done for the FT817 installation, where a 1 cm piece of 30 AWG wire wrap wire was first soldered to the RFIN pad on the PCB. The PCB was installed against the PLL shield sidewall so that the RFIN was opposite the IF filter connection point. In this case all that was needed was about 3mm wire length to connect the PCB to the filter.