

PAUL Construction Manual

PAUL (Packet and Allstar Universal Link) is a kit construction project that implements a radio interface to a Raspberry Pi running Allstar Amateur Radio VOIP networking software [HamVOIP](#), [AllstarLink](#) or Direwolf AX.25 packet modem/TNC and APRS encoder/decodersoftware [Direwolf](#)

An Allstar node can be implemented using PAUL, a UHF or VHF radio having (or can be modified to provide) some type of COS output and PTT input, a custom cable and a Raspberry Pi running the Allstar program.



Figure 1-PAUL (LED end)



Figure 2- The connector end of PAUL

PAUL offers more interface flexibility than our DINAH product which is designed specifically to interface to the 6 pin DIN connector commonly used for the packet (or data) connection on ham radio VHF and UHF radios.

PAUL features a standard female 9 pin dsub connector (DE-9) wired with the standard Kantronics packet interface pinout for the cable which plugs into your radio. A USB miniB connector is used for the cable connecting to your Raspberry Pi. Two potentiometers are provided to allow the user to decrease the audio signal levels between PAUL and your radio (no audio amplification or filtering is provided). Optically coupled solid state relays are used for the COS and PTT interfaces. Eleven solder bridge jumpers facilitate interfacing to different COS and PTT signal levels.

PAUL is packaged in a small black enclosure (80x50x20 mm) created from two aluminum extrusions. Custom 3D printed end caps are provided so the kit builder is not required to create any holes or cutouts

Designed by N8AR, PAUL is available as a parts kit or completely assembled.

Disclaimer:

This device controls equipment that could be damaged by said device or your device could damage PAUL if connected incorrectly. You are responsible for installing, configuring, testing and verifying that the device performs properly with your equipment in your environment. The developers cannot be held liable for any direct, indirect, consequential or incidental damages to other pieces of software, equipment, goods or persons arising from the use of this device.

By constructing this device, you accept the above terms of disclaimer.

Release Notes:

RELEASE	DATE	CHANGES
1.00	2021-04-12	Initial release
1.01	2021-11-12	Typo fixes
1.02	2021-18-12	Typo fixes

Table of Contents

PAUL Overview.....	4
Key Features.....	4
Degree of soldering difficulty – Easy.....	4
Degree of mechanical difficulty - easy (if you use the 3D printed plastic end caps).....	4
Required Materials.....	5
PAUL Kit.....	5
Tools.....	5
Step 1. Disassembly	6
Step 2. Configuring the Solder Bridge Jumpers	6
Step 3. Constructing PAUL	9
Step 4. Custom Interface Cable.....	11
Step 5. OPTIONAL - Cutting Holes in the Aluminum End Caps	11

PAUL Overview

Key Features

- Small, portable, USB-powered
- Pre-assembled SMD board simplifies construction
- Uses the CM119B or CM108B USB audio codec integrated circuit
- Two potentiometers (RV1 and RV2) to adjust (attenuate only) audio levels
- Rugged, RFI-resistant aluminum enclosure
- RFI filter capacitors
- Ferrite beads
- Uses standard female 9 pin dsub connector (DE-9) for radio interface
- Uses standard USB miniB connector for computer interface
- 3D printed plastic enclosure end plates minimize mechanical assembly effort
- Blank aluminum end caps included for modification and use in the event of RFI issues
- 11 solder bridge jumpers and extra parts locations facilitate custom configurations

Degree of soldering difficulty – Easy

- The DINAH PC board provided with the kit comes with all the surface mount parts installed. The builder installs three through-hole parts.
 - A 9 pin dsub connector
 - Two potentiometers

Degree of mechanical difficulty - easy (if you use the 3D printed plastic end caps)

- If you decide to use the metal end caps, using the printed end caps as templates, you will need to mark, drill and cut the required holes and openings.

Required Materials

PAUL Kit

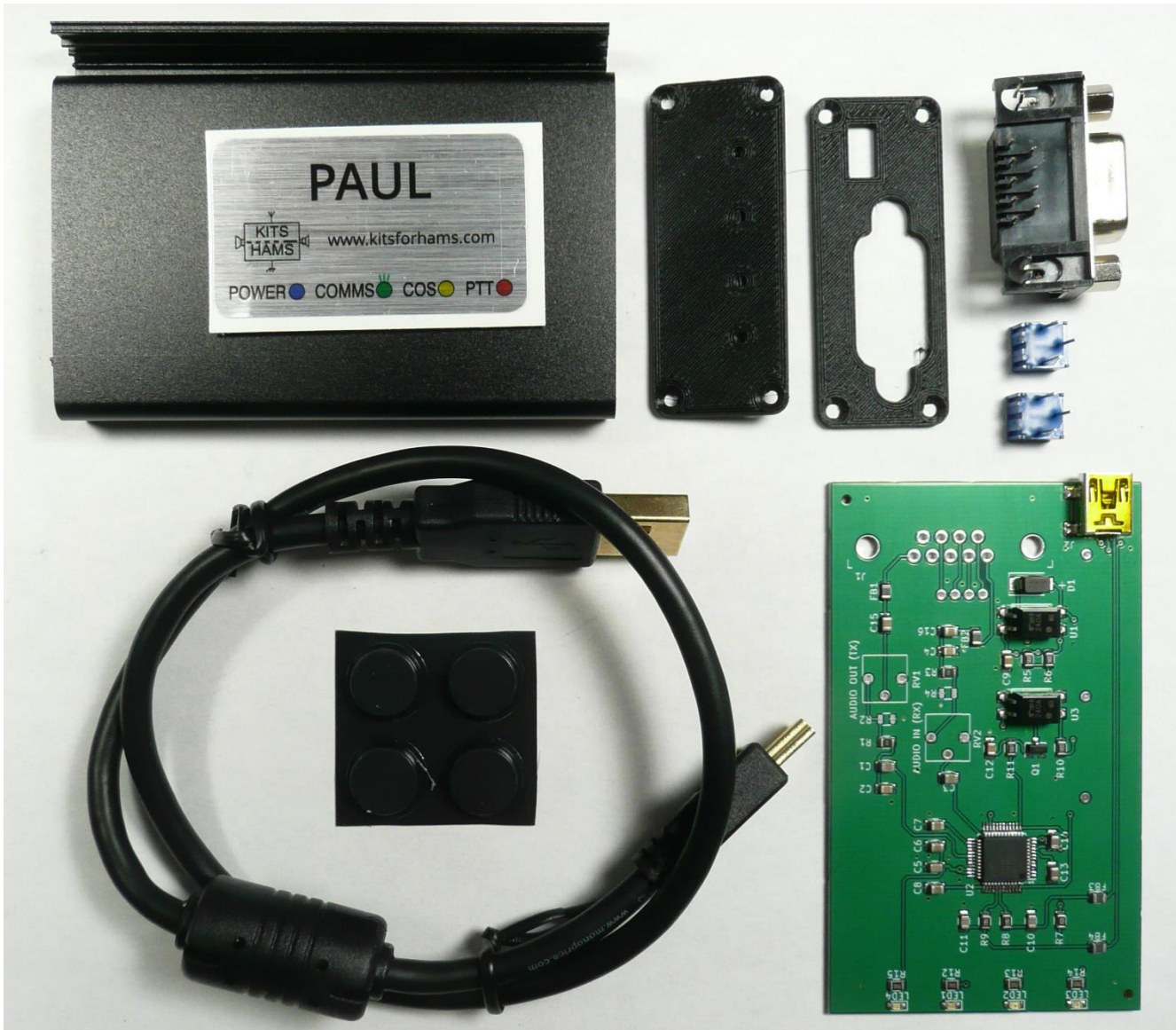


FIGURE 1 - PAUL Kit Parts

Tools

- Low wattage (50 watt) solder pencil with small tip and solder
- #1 Phillips screwdriver
- Small side cutters
- Xacto knife or equivalent if modifying some solder bridge jumpers
- Drill, drill bit set and a set of small files if using metal end caps

Step 1. Disassembly

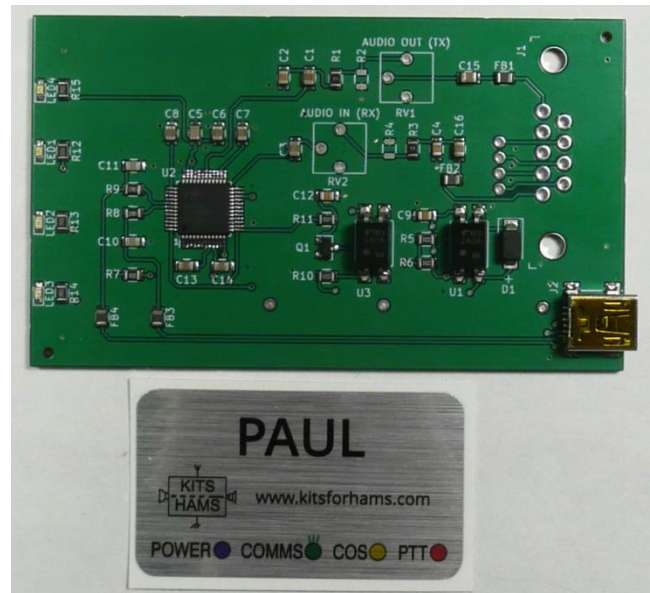
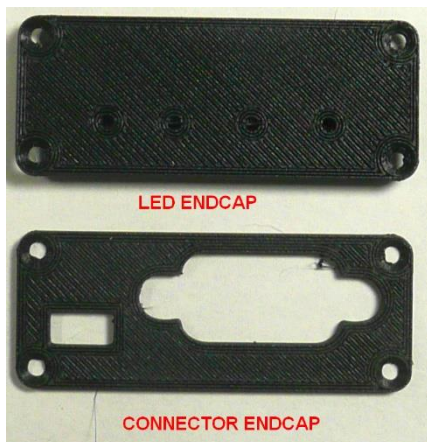
The PAUL kit is shipped with the PC board and the PAUL label located inside the assembled case. In this step you will disassemble the PAUL case to access these parts.

CAUTIONS:

- The PAUL PC board contains static sensitive parts. Use static prevention procedures when working with the PC board.
- The 3D printed LED end cap has cavities which fit over the SMD LED's. Always remove the PC board by removing the four fastening screws from the LED end cap and lifting it up and away from the case.

DO NOT remove the connector end cap first and pull on the PC board.

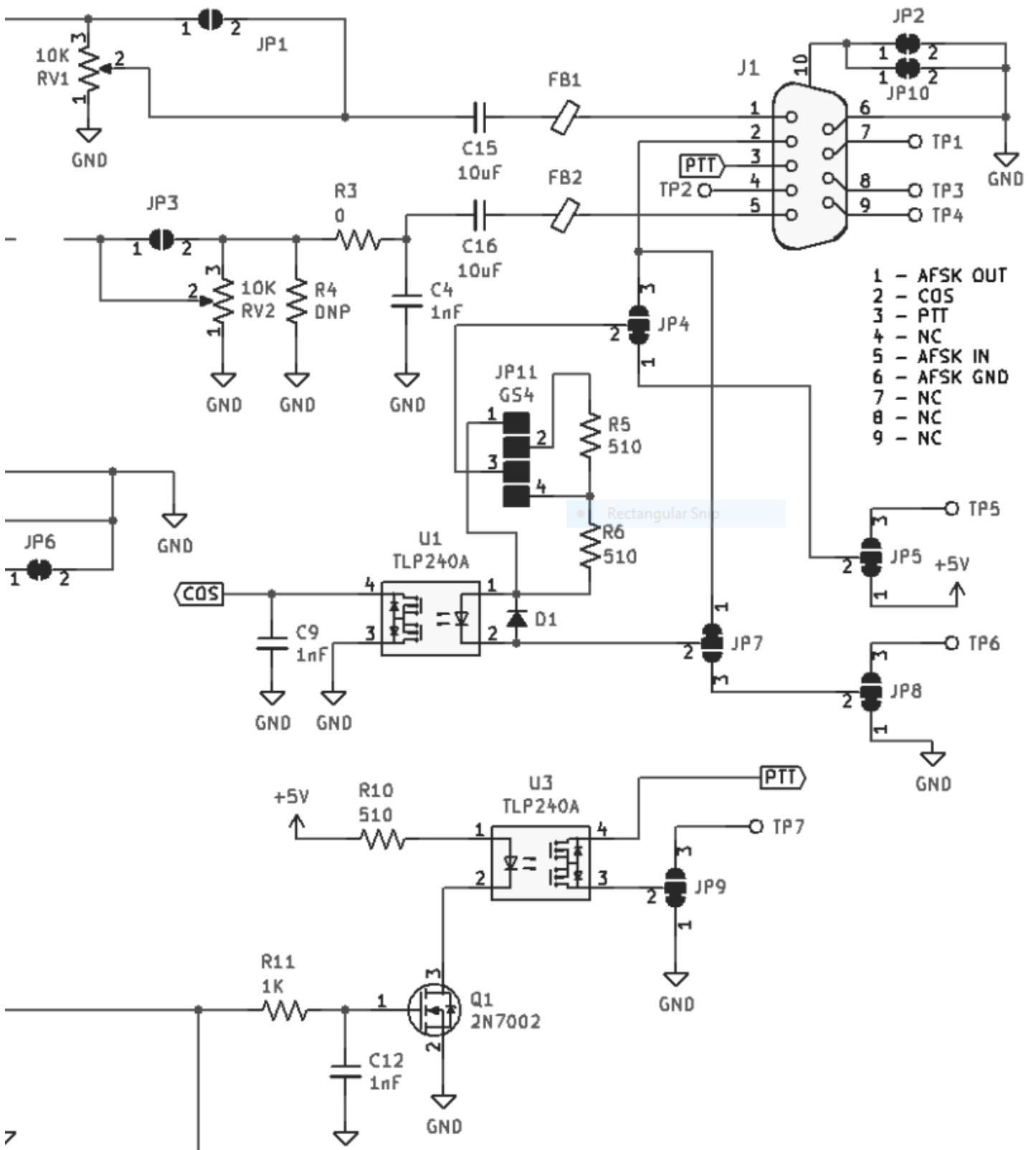
Remove the LED end cap from the case by first removing four fastening screws from the LED end cap and lifting it up and away from the case. Remove the PC board and label located in the case and set them aside for later installation. Remove the two screws holding the top of the case to the connector end cap. Set these parts aside also.

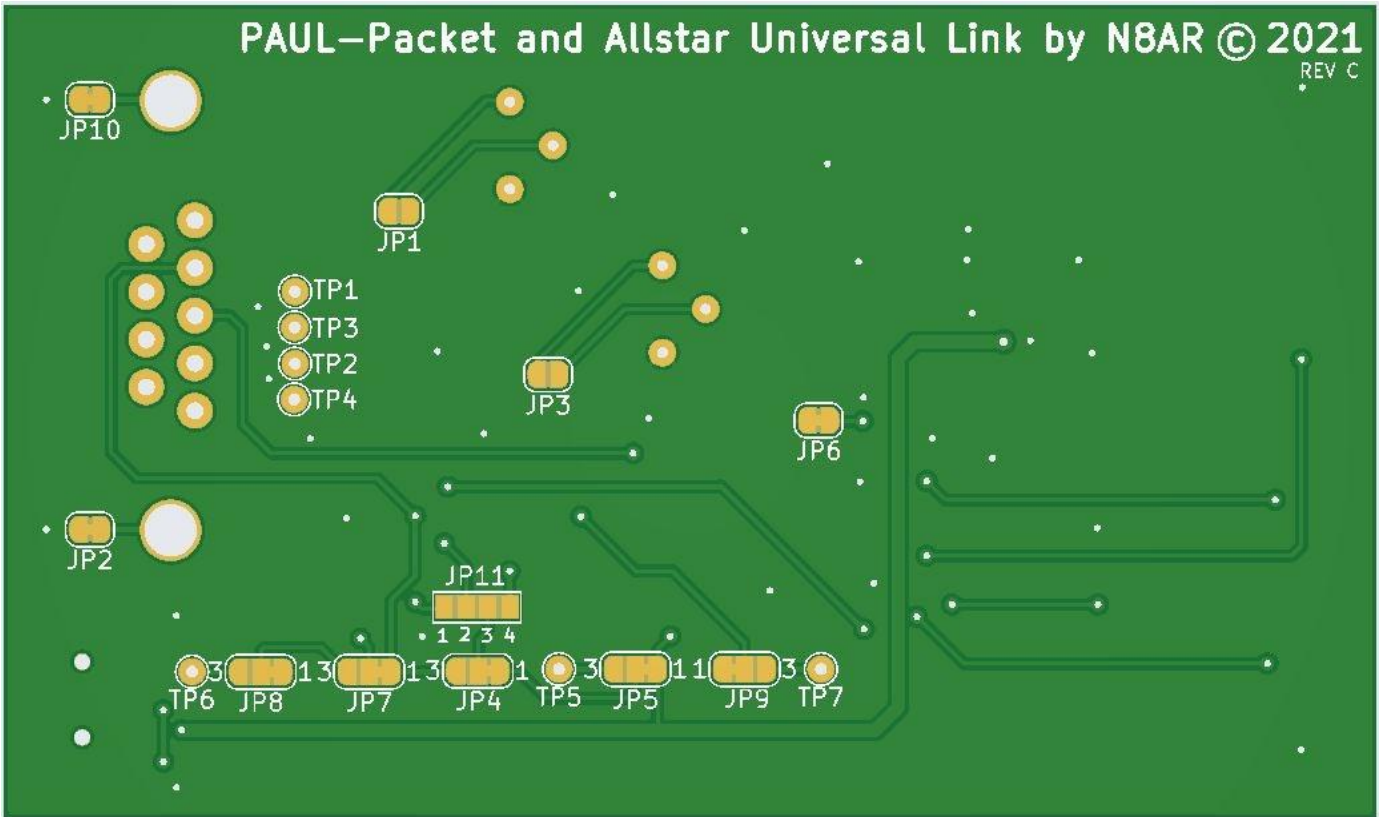


Step 2. Configuring the Solder Bridge Jumpers

There are 11 solder bridge jumpers on the bottom of the PAUL PC board to provide flexibility in it's use. JP1 through JP10 have a default open or short configuration etched into the PC board. **In the default configuration (as shipped) JP11 must be configured by the kit builder with a solder bridge from pads 3 to 4 if the board is being used for Allstar (i.e. using COS).**

A portion of the PAUL schematic showing the electrical connections of the jumpers as well as the pinout of J1, the DE-9 dsub connector is shown in the following figure. A view of the bottom of the board showing the location of the jumpers and a table showing their purpose is shown on the following page.





JUMPER	# OF PINS	DEFAULT CONFIGURATION	PURPOSE
JP1	2	Open	Used to short potentiometer RV1 input to output to eliminate RV1. One use for this would be when connected to a 6 pin miniDIN packet connector.
JP2	2	Shorted	Used along with JP10 to disconnect the cable shield from ground.
JP3	2	Open	Used to short potentiometer RV2 input to output to eliminate RV2. One use for this would be when connected to a 6 pin miniDIN packet connector.
JP4	3	1-2 shorted	This jumper selects what signal or voltage level is routed to the anode of the LED diode of the COS input SSR (U1). The default is 5 VDC via JP5.
JP5	3	1-2 shorted	This jumper selects between 5 VDC or TP5 being routed to the anode of the LED diode of the COS input SSR (U1). The default is 5 VDC via JP5.
JP6	2	Shorted	This jumper is shorted if a CM108B is installed and open if a CM119B is installed. This should already be configured as required by the factory.
JP7	3	1-2 shorted	This jumper selects what signal or voltage level is routed to the cathode of the LED diode of the COS input SSR (U1). The default is COS via JP7.
JP8	2	1-2 shorted	This jumper selects between TP6 and ground being routed to JP7
JP9	3	1-2 shorted	This jumper determines whether PTT is connected to ground or TP7. If using TP7, TP7 would be connected via a jumper wire to TP1,2,3 or 4 on the dsb.
JP10	2	Shorted	Used along with JP2 to disconnect the cable shield from ground.
JP11	4	All open	This jumper is used to select the current determining resistor for the input LED diode of the COS SSR (U1). If all are open COS is disconnected. Use a solder bridge from 3-4 to configure for an open collector/drain COS signal. This will select 510 ohms of resistance for about 7.5 mA thru the diode.

The diode input of the COS SSR (solid state relay, U1) requires a minimum of 5 mA to activate the SSR. The nominal recommended current is 7.5 mA. **Do not exceed 28 mA through the diode.** JP11 is used to adjust this current depending on the voltage being impressed on the diode/resistor combination.

If the voltage is 5 to 12 volts, use a solder bridge between J11, pads 3 and 4. This results in 510 ohms of resistance. The diode voltage drop is about 1.2 VDC. For the defaults used on the other jumpers with 5 VDC, we have 3.8/510 mA or about 7.5 mA.

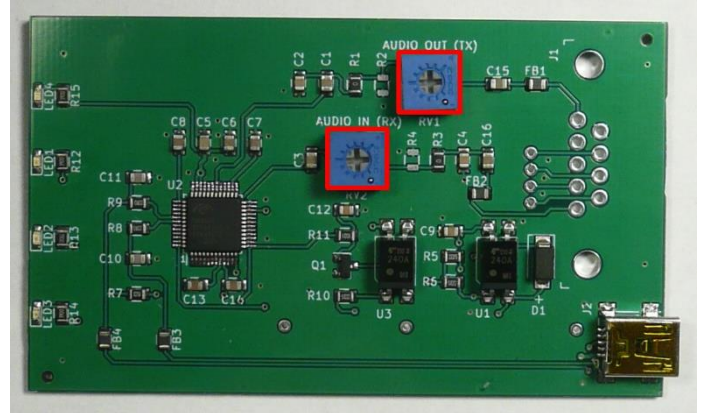
For 12 to 28 VDC, use a solder bridge between J11, pads 2 and 3. This results in 1020 ohms of resistance. The diode voltage drop is about 1.2 VDC. With 12 VDC, we have 10.8/1020 mA or about 10.5 mA.

For 3.3 VDC up to 5 VDC, use a solder bridge between J11, pads 1 and 2 and a second bridge between pads 3 and 4. This results in 255 ohms of resistance. At 3.3 VDC we have 2.1/255 mA or about 8 mA.

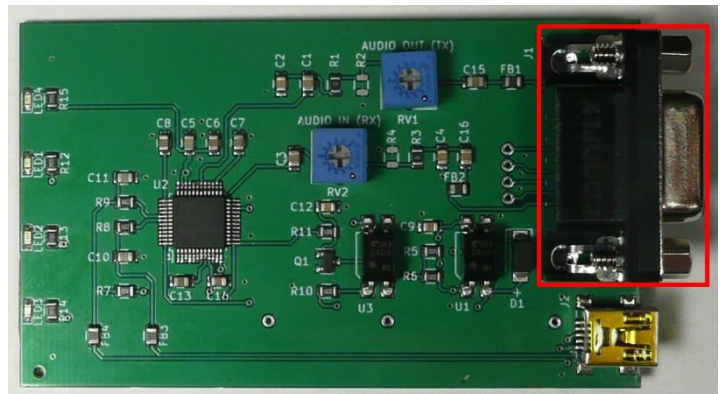
Step 3. Constructing PAUL

In this section you will assemble the PC board by installing three through hole parts and configuring the 11 jumpers. You will then perform final mechanical assembly.

Install the two potentiometers labeled RV1 and RV2 into their locations on the PC board. Ensure they are flush with the PC board and solder in place. Trim any excessive lead length.



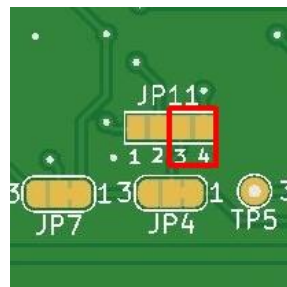
Install the 9 pin dsub connector labeled J1 into its location on the PC board. Ensure it snaps into place and is flush with the board. Lightly solder the two locating tangs and then solder the 9 pins on the connector. Trim back the excessive lead length on the two locating tangs to eliminate interference when the board is inserted into the lower part of the case as described in the next step.



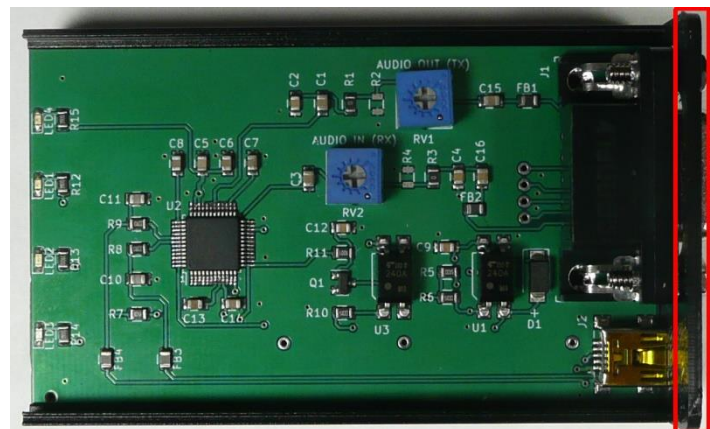
This step assumes you are using the default jumper configuration for JP1 thru JP10 and you want to use COS activated by an open collector/drain output from your radio.

Use your soldering iron and solder to create a solder bridge between pads 3 and 4 of JP11.

If you are using a different jumper scheme, cut traces and create solder bridges as required for your scheme



Slide the finished PC board into the lower half of the case which has the attached connector end cap. Place the top cover in place and install two more screws in the connector end plate. Slide the LED end cap in place over the LED's and fasten in place using four screws.



Remove the PAUL label from its backing and install it on the top of the case in the position shown in the photo.

Install the four polyurethane feet in place on the bottom of the case at the four corners.



Step 4. Custom Interface Cable

If you are using PAUL to interface to a standard 6 pin miniDIN packet connector, we suggest that you consider acquiring a DINAH instead because the required cable is provided with DINAH. One exception to this that we are aware of is interfacing to an Alinco packet connector which is a DE-9 rather than a 6 pin miniDIN. The following table shows the design of a cable for an Alinco DR-435.

PAUL DE-9 CONNECTOR PIN NUMBER	PAUL SIGNAL NAME	DR-435 DE-9 CONNECTOR PIN NUMBER	ALINCO DR-435 SIGNAL FUNCTION
1	DATA IN	9	Packet transmission DATA input (1200 bps). Input level 100 mV/600 Ohm
6	GND	5	Ground
3	PTT	7	PTT signal input: Low (GND)=TX, Open=RX
4	TP2		
5	1200	4	Packet reception DATA output (1200 bps). Output level 100mv/600 ohm
2	COS	1	SQC squelch signal output. Carrier in: closed. Open collector output
7	TP1		
8	TP3		
9	TP4		

For other cables, you may find what you need at one of the following vendors:

<https://hammadeparts.com>

<http://uricable.com/>

Step 5. OPTIONAL - Cutting Holes in the Aluminum End Caps

The PAUL kit ships with 3D printed plastic end caps that align with the connectors and LEDs on the printed circuit board. The enclosure's original metal end caps are also included. Drilling and cutting the aluminum end caps is required only if you decide not to use the plastic end caps because they are not as pretty, you need additional EMI shielding or you just like the challenge.

You can use the printed end caps as a template to mark the metal end caps for drilling, cutting and filing.

After creating the required holes, we have found that a black permanent marker works nicely to "touch-up" the exposed aluminum.