Changes and corrections for the PacSat Prototype PCB

These parts were flagged by the PCB assembly folks. Most of the issues were

addressed but the list should be reviewed.

1. C96 the part is too big for the board layout.

2. C601 the part is too small for the board layout.

3. C714 the part is too big for the board layout.

4. D26 the part is too small for the board layout.

5. L154, L155 the part is too small for the board layout.

6. L710 the part is too big for the board layout.

Please provide the correct p/n for these positions.

7. Please identify pin one for Q601 and U28, as the board layout is not clear.

Consider removing all the MAX4995ALAUT devices. The one used for 1.2-volt supplies is not capable of controlling the low voltage. The others are only useful if you can power down the circuit being serviced by the device. The Power AMP and AX5043 devices my be powered down but then you will have no RF. So thinking about whether these devices are needed would be good.

If MAX4995ALAUT parts are kept carefully consider the options on these parts and relocate the select driving pins on the TMS570 per the Rev D schematic.

Replace all the voltage regulators with regulators which are more robust. Pay close attention to the max voltage which may be developed on the 3.3V and 1.2V supply as the TMS570 is easily damaged with excess voltage.

Correct the I2C address on one of the temperature sensors, probably U30. The correction is shown in the Rev D schematic. Otherwise, they will be at the same address and not work properly.

Do not connect any 5V monitor signals to the TMS570 as this will be an overvoltage condition and damage the device.

Select the crystal to be used for flight based on crystals that can be purchased. The board layout accommodates two possible crystals and can be left this way.

Consider how many AX5043 receivers are needed for the application. Remove any that are not needed. Simplify the Rx distribution network if only two or one are required.

Correct the footprint on T601. Do not use the default KiCAD footprint.

Determine if differential signals will be used on the Rx inputs. If only single ended is required reconfigure the Rx input section of the AX5043 devices. If only one Rx device is used, consider making all the connections with traces on the PCB. If multiple devices are used or differential inputs, then consider continuing to use the rf jumpers and plan on soldering the connectors in place before flight.

Consider adding more capacitance for the Real Time Clock (RTC). These must be low leakage capacitors and the board must be cleaned well after assembly to minimize the possibility of the capacitors discharging their potential which is needed to preserve the clock. The current needed by the clock is picoamps and very low level.

Remove all LEDs from the board preflight to reduce power needs.

Follow Rev D for proper RF amp biases of the input preamplifier.

The power plane of the PA amp and the input 5V regulator must be connected to work properly in the PCB layout. This was not done in Rev C.

Most corrections are shown in the Rev D schematic, but no changes have been made to the PCB.

KiCAD has advanced to Version 8. If the design is pulled forward, you will need to verify there is no issue with schematic and PCB.

Prototy boards were built with Lead Free solder however most NASA designs call for using solder with lead to reduce the possibility of developing shorts.

The negative supply voltage is provided by an optoisolator photodiode stack. The current for the LED on the isolator is derived by putting the LED in series with the bias for the 1.2-volt regulator. If the new 1.2-volt regulator design will not support this then the current bias for the LED will need to be taken from the 5V supply or the 3.3-volt supply.

N5BRG 4/19/24