This document explains the function of the Solar Panel, its schematic level design, and its board level design.

Umbilical

Umbilical Design

Revision: 1.0.0

Jayson Lopez

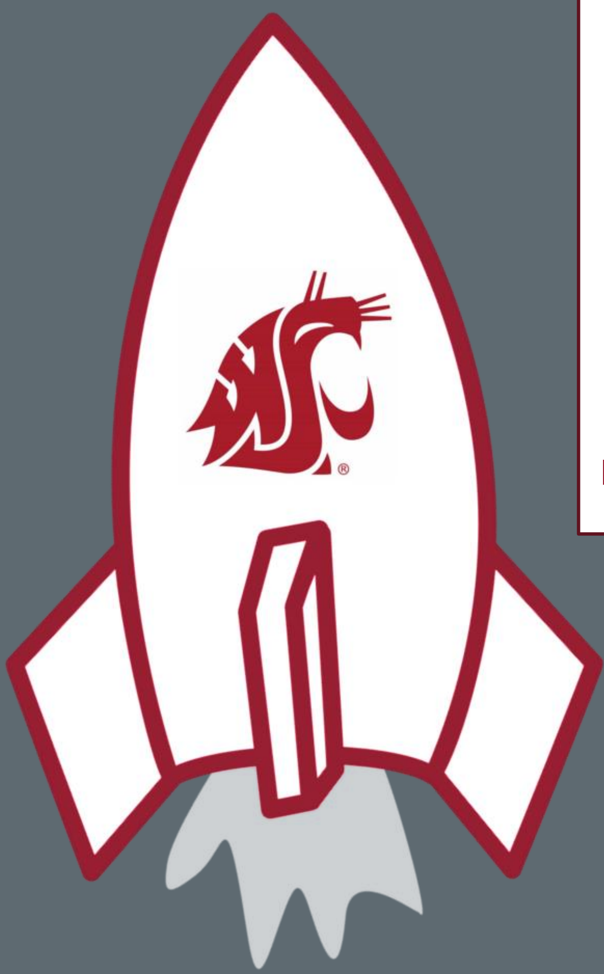

Table of Contents

1 Introduction

This document explains how the Umbilical will fulfill the following Functions and conform to the following requirements. This document refers to the Umbilical version 1.0.0

1.1 Function

The Umbilical is responsible for the following:

- Charging the battery on earth without the sun
- Provide a bi-directional testing communication link to CD&H subsystem

1.2 Requirements

1. Provide 4.2 V 0.8 A battery charging
2. Provide a UART to USB conversion of at least up to 115200 BAUD

2 Detailed Description

This section references the Umbilical schematic. Page numbers will be listed and may have coordinates listed (number and letter combination found around the frame).

2.1 Functional Block Diagram

2.1.1 Battery charger function

Charges the battery at 5V and 2A from the wall adapter. Charges lithium -ion batteries at 4.1V.

2.1.2 UART to USB

Converts USB into a UART signal. USB 2.0 compatible which is what we will be using. Also contains USB Battery Detection[1].

2.2 Schematic

2.2.1 Battery Charger

The battery charger portion of the umbilical uses the BQ24650[2] battery charger IC to charge the lithium-ion batteries found on the EPS board. The purpose of this is to provide a fast, reliable way to charge the batteries without having to rely on the sun during testing.

[1] USB Battery Detection
[2] BQ24650

Power for the battery charger will be supplied using an AC-to-DC wall adapter which will be capable of supplying 5 V_{dc} and 2 A. An ideal diode[3] is used to ensure that current can only flow one way from the wall adapter to the battery, without allowing the battery to discharge back to the supply.

Unless otherwise stated, the typical application layout for the BQ24650 IC was used for this design. The notable changes will be detailed below.

No thermistor was used in this design next to the battery because the batteries that will be charged by the umbilical board are located on a different PCB, making the use of a thermistor for this chip impractical. Instead, the voltage divider between VREF and TS was set such that the temperature registered by the BQ24650 will always be between the low and high acceptable temperatures, meaning that the charger will never turn off due to temperature. The temperature of the batteries on the EPS board are already monitored by the processor on the EPS so battery temperature should not be an issue. It is still recommended to not leave the charger on and unattended for long periods of time as lithium-ion batteries are very sensitive and may be dangerous.

The MPPSET pin was tied to a constant 5 V, meaning that the maximum power point will allows be tracked, as this occurs anytime that this pin is at least 1.2 V.

The resisters that go to STAT1 and STAT2 were changed from 10 kΩ to 1 kΩ to maintain consistency with how LED's have designed for other boards on Cousat1. This will just lead to slightly less bright LED's compared to those from the original datasheet.

The output voltage to the battery was determined using Equation 1 below:

$$V_{bat} = 2.1V * \left[1 + \frac{R_2}{R_1}\right] \qquad (1)$$

The desired battery voltage was approximately 4.2 V, this was achieved by using $R_1 = R_2 = 10$ kΩ as seen below:

$$V_{bat} = 2.1V * \left[1 + \frac{10\ k\Omega}{10\ k\Omega}\right] = 4.2v$$

The battery charge current was designed using Equation 2 below:

$$I_{charge} = \frac{40\ mV}{R_{SR}} = \frac{40\ mV}{50\ m\Omega} = 0.8\ A \qquad (2)$$

[3] Ideal Diode

Where R_{SR} is the current-sense resistor chosen to measure the output current. In order to obtain an output charging current of 0.8 A, a value for R_{SR} was chosen to be 50 mΩ:

Two N-channel MOSFETs were used to control the output of battery charger. The MOSFET controlled by HIGHDV switched the connection between the power supply and PH while the MOSFET controlled by LODRV switched between PH and ground. A truth table showing the switching combinations for these two MOSFETs can be seen below in ***Table 1:***

Table 1: Switching Truth Table

High Drive	Low Drive	Output Enable
0	0	0
0	1	0
1	0	1
1	1	0

The only time that the output will be enabled is when HIGHDV supplied a high voltage and LODRV supplies a low voltage to the gates of their respective MOSFETs.

2.2.2 UART to USB Transceiver

This section of the umbilical contains an adapter that takes in a UART signal from another PCB (such as the EPS) and converts it into USB via the FT230XQ[4] IC.

An RC time constant of 1 ms was added on to the RESET pin of the FT230XQ to ensure that the power supply to that pin has enough time to reach its full steady-state value before the chip begins to function. This time constant was achieved using a 10 kΩ resistor and a 100 nF capacitor.

[4] FT230XQ

The ferrite bead shown between V_{CC} of the USB adapter and V_{CC} of the FT230XQ was used to filter out high frequencies. Additional shunt capacitors were also added around the ferrite bead for further filtering purposes. The ferrite bead is using the typical application from its data sheet.[5]

Other than what has been mentioned above, the typical application schematic for the FT230XQ was used for designing the UART-USB portion of the umbilical.

[5] Ferrite Bead

2.3 Board

The board shall be double-layered with 1 oz copper and ENIG finish.

2.3.1 Layout Constraints

Unless specified in the following subsections, all signals shall use the default parameters below. Signals in the following subsections do not include their sense signals unless otherwise specified. Trace width can be broken if a trace needs to bottleneck down to a pin, the bottleneck shall be minimized.

Trace width:	$0.16mm$
Vias:	$\emptyset 0.3mm$, unlimited count
Separation:	$0.16mm$
Length:	unlimited

Devices with specific placement and routing considerations are called out on the schematic, see “CAD Note:”

2.3.1.1 Charger Power Traces- 5.0V, DIODE_OUT, PH, SR_P, UMB_IN, PGND

PGND applies to between the input and output connectors

Trace Width: $0.4mm$

2.3.1.2 USB Differential Pair – USB_CONN_N, USB_CONN_P, USB_N, USB_P

Length:	Length match $\pm 1.0\ mm$
Trace width:	$0.8mm$
Gap width:	$0.16mm$

2.3.1.3 VREF, LODRV

Trace width: $0.32mm$

Trace with is 0.32mm to fix clearance issues.

2.3.1.4 SR_P, CHARGING_STAT_G

Trace width: $0.35mm$

Trace width is 0.35mm to fix clearance issues

3 Testing

All tests shall be performed at room temperature and will not be performed under vacuum since the umbilical PCB will not be included on the actual satellite. If any modifications are performed, take note. Include enough information to understand circuit behavior and for others to replicate the results. Include any software written to execute the test and link it in the test notes section. Save all software, waveforms, etc. in a subfolder of the board's test folder for each test. When testing, keep the following guidelines in mind:

- Waveforms shall be captured whenever appropriate
- Have the event take fill the screen (for fast events, zoom in, for slow events, zoom out)
- Label each channel accurately
- Only have bandwidth limiting if necessary, for the test (this applies to the oscilloscope and probe settings)
- If ringing or overshoot occurs, use a ground spring or differential probe

Test results location:

Common test instructions can be found on the wiki.

3.1 Before First Power-On Check

This test is required to be executed before the umbilical board is connected to any external power source.

3.1.1 Test Instructions

Measure the resistance of various points in reference to PGND located at the output connector.

3.1.2 Test Data

The data for this test will be recorded into the table below: