Soldering Manual: Solar Module

Section 1: Power Supply & Protection Stage

Before populating the sensitive digital measurement circuitry, you must assemble, solder, and verify the local power input and 3.3V voltage regulation stage. This ensures the digital bus transceiver and monitor chip operate under safe, regulated logic levels.

1.1 Components to Solder in this Step

Solder *only* the following low-profile components onto the Main PCB:

- **U1**: L78L33 Linear Voltage Regulator (3.3V Output)
- **C1, C3:** 10µF Capacitors
- **C2**: 1µF Capacitor
- **JST 4P Connectors**: SystemBus input/output headers

1.2 Quality Control & Bench Testing Procedure

Once these initial components are securely soldered, prepare your testing wire harness to safely introduce power to the board and verify the local 3.3V rail.

Step 1: Test Wire & Bench Setup

1. Take a standard 4-pin JST mating cable harness.
2. Connect **Pin 1 (+12V)** of the JST connector directly to the positive terminal of your laboratory power supply.
3. Connect **Pin 2 (GND)** of the JST connector to the negative (GND) terminal of your laboratory power supply.
4. Set your laboratory bench power supply to **12.0V** and limit the current to **100mA** as an essential safety precaution.

Step 2: Voltage Regulation Test

1. Plug the 4-pin JST test wire into one of the soldered JST bus connectors to power up the rail.
2. Take your multimeter and switch it to **DC Voltage mode**.
3. Place the black probe on a known GND point (such as JST Pin 2) and the red probe on the output pin of the **L78L33 (U1)** or across the pads of **C3**.

📋 **Expected Result**

The multimeter must display a stable **3.3V**.

- **If the voltage is correct:** Disconnect the power supply. You are ready to move on to the next assembly stage.
- **If the voltage reads 0V or U1 gets hot:** Immediately shut off power. Inspect your solder joints for shorts between the input, output, and ground pads.

Section 2: High-Precision Telemetry Stage

In this stage, you will populate the core sensor intelligence of the Solar Module: the INA226 hardware monitor and its high-frequency noise decoupling.

2.1 Components to Solder in this Step

Solder the following surface-mount (SMD) components onto the board. *Ensure correct Pin 1 orientation for the IC by aligning the dot/bevel on the chip package with the silkscreen indicator on the PCB:*

- **U3 (INA226)**: High-Precision I2C Current/Power Monitor IC (MSOP-10 or SOIC package).
- **C5**: 100nF Ceramic Decoupling Capacitor.

2.2 Hardware I2C Address Selection

Directly after soldering the INA226, configure its permanent hardware I2C address using the onboard solder jumpers (**A0, A1**).

- Apply solder to bridge the pads according to your system's master address map.

Section 3: High-Current Measurement & SystemBus Integration

To finalize the Main PCB, you will install the heavy-duty current shunt interface and the main power distribution terminals.

3.1 Components to Solder in this Step

Solder the final high-profile and high-power components onto the PCB:

- **Screw Terminals**: Heavy-duty terminal blocks for the solar panel charge controller input/output feed.

3.2 🚫 System Bus Architecture Rule (R3 & R4 Pull-up Exclusion)

- **DO NOT populate R3 and R4.** These pads are designed for optional I2C bus pull-up resistors.
- Because the central **Master Module** already integrates dedicated, hardware-stabilized pull-up resistors on its core logic rails, adding additional pull-ups on the expansion modules will lower the bus resistance excessively, overloading the data lines and causing communication failures across the SystemBus. Leave these spots completely empty.