Soldering manual Master Unit

Section 1: Power Supply & Protection Stage (First Assembly & QC)

Before populating the sensitive digital logic (ESP32-C6, W5500), you must assemble, solder, and test the power input and regulation stage. This ensures your board handles power correctly and prevents accidental damage to the main chips.

1.1 Components to Solder in this Step

Solder *only* the following components on the PCB:

- **J2:** 12V input connector
- **F2:** Fuse Holder with 3A mini Car fuse
- **R24:** 0.025ohm shunt Resistor
- **D3:** SS34 Schottky Diode
- **D5:** PTVS18V Transient Voltage Suppressor
- **U1:** AP63203WU Buck Regulator.
- **L2:** 3.9µH Inductor (2A)
- **Associated Capacitors:** Input decoupling capacitors (C1_10uf, C3_10uf) and output smoothing capacitors (C19_22uf,C30_22uf), including the Bootstrap capacitor (C4_100nf).

1.2 Quality Control & Bench Testing Procedure

Once soldered, connect the module to a regulated laboratory power supply.

Step 1: Continuity & Short-Circuit Check (Power OFF)

Set your multimeter to Continuity/Diode mode (Beep test).

1. Measure across the 12V Input terminals. The meter must not beep (Open Line / high resistance).
2. Measure across the 3.3V Output capacitor. The meter must not beep.
    - *If your meter beeps at 0 Ohms, stop immediately. Check for solder bridges underneath the AP63203WU or shorted capacitors.*

Step 2: Reverse Polarity Protection Test

1. Set your lab power supply to **12.0V** and strictly limit the current to **100mA**.
2. Connect the power supply **backwards** to the input (Positive to GND, Negative to VIN).
3. **Expected Result:**
   - The lab power supply should read **0.00A** (no current draw).
   - Measure the voltage on the 3.3V rail. It must read **0V**.
   - *(Note: You might measure a tiny ghost voltage behind the SS34 due to its minor Schottky leakage current, but this will drop to 0V once fully clamped by the internal AP63203WU circuitry).*

Step 3: Voltage Regulation Test (Normal Power)

1. Keep the lab power supply at **12.0V** (Current limit at **300mA**).
2. Connect the power supply **correctly** (Positive to +12V, Negative to GND).
3. **Expected Result:**
   - The current draw on the lab supply should be very close to **0.00A** (idle consumption of the regulator).
   - Measure the voltage on the 3.3V rail with your multimeter. It must read a stable +/-**3.3V**.

Only proceed to Section 2 if your 3.3V rail measures perfectly stable and your reverse polarity test draws zero current. If you read a lower voltage (e.g., 2.2V) or 0V, reflow the pins of the AP63203WU and check the bootstrap capacitor before moving forward!

Section 2: The Brain (ESP32-C6 & USB Interface Assembly)

In this stage, you will populate the core intelligence of the Main Module: the ESP32-C6 microcontroller, its essential passive stabilizing components, the physical control tactile switches, and the USB-C interface for initial programming.

2.1 Components to Solder in this Step

Solder the following components onto the PCB:

- **U3:** ESP32-C6 Module.
- **Capacitors:** C6_100nf, C7_100nf, C11_100nf and C8_10uf.
- **Resistors:** R5_10K, R6_10K, R7_5K1, R8_5K1
- **Tactile Switches:** Boot button_SW1 and Reset button_SW2.

- **USB1 Interface:** USB-C Connector.

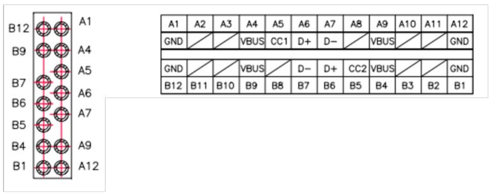
2.2 Quality Control & Pre-Power Checks

Because the USB-C connector pads and the ESP32-C6 pins are extremely fine-pitched, conducting physical checks before applying voltage is **critical** to avoid frying the microcontroller.

Step 1: USB Data Line Check

Set your multimeter to Continuity/Diode mode (Beep test).

1. Refer to the USB-C pinout diagram below to locate the GND, D+, and D- pads.
2. Measure between GND and D+, then between GND and D-.
3. **Expected Result:** The meter must not beep. If there is a short circuit here, it will prevent your PC from recognizing the board or could trip your computer's USB port protection.



Step 2: Main Logic Rail Check (CRITICAL)

Before connecting any power source, measure directly between the 3.3V rail and GND on the ESP32-C6 pads.

- **Expected Result:** The meter must not beep (Open Line or high resistance).
- ⚠ If your multimeter registers a short circuit (0 Ohms), **do not power up the board.** Inspect the fine legs of the ESP32-C6 and the USB-C pads under a microscope for microscopic solder bridges or tin whiskers. Resolve the short circuit before proceeding.

2.3 Initial Flash & Validation Test

Once all short-circuit tests pass successfully, you can proceed to boot and program the microcontroller for the first time.

1. Set your bench power supply to **12.0V** (Current limit at **300mA**).
2. Connect the main power. The current draw should read **0.00A** to **0.02A** (minimal standby current).
3. Connect a USB-C cable from your PC to the Main Module.
4. Put the chip into flashing mode: Hold down the **BOOT** button, press and release the **RESET** button, then release the **BOOT** button.
5. Deploy a stripped-down validation firmware via ESPHome to verify the stability of the core system.

```yaml
esphome:
  name: master_unit
  friendly_name: master_unit

esp32:
  board: esp32-c6-devkitc-1
  framework:
    type: esp-idf

# Enable logging via USB
logger:

# Enable Home Assistant Native API
api:
  encryption:
    key: ""

ota:
  - platform: esphome
    password: ""

wifi:
  ssid: !secret wifi_ssid
  password: !secret wifi_password
  id: wifi_module

  # Enable fallback hotspot (captive portal) in case wifi connection fails
  ap:
    ssid: "master_unit Fallback Hotspot"
    password: ""

font:
  - file: "gfonts://Roboto"
    id: font_main
    size: 14
  - file: "gfonts://Roboto"
    id: font_small
```

```
    size: 10

i2c:
  - id: bus_a
    sda: GPIO6
    scl: GPIO7
    scan: true
    frequency: 100kHz  # Standard I2C operational frequency

# Note: Hardware packages are commented out during initial MCU validation
# packages:
#   - !include c6_module.yaml
```

Successful Outcome: The log window should show a compiled firmware upload success, and the I2C bus will initiate its automatic scan. You now have a fully functional "brain" ready for peripheral integration!

Section 3: Local Power Monitoring (INA226 & I2C Bus Assembly)

With the core microcontroller validated, you will now assemble the local diagnostic stage. By populating the INA226 hardware monitor, the module gains the ability to measure its own input voltage, shunt voltage, and real-time current consumption.

3.1 Components to Solder in this Step

Solder the following components onto the PCB:

- **IC:** U4 - INA226 Current/Power Monitor (Pay attention to the Pin 1 orientation dot).
- **Resistors:** R3, R4 (4.7kΩ I2C pull-up resistors for the SDA and SCL lines).
- **Capacitors:** C20 (100nF decoupling capacitor for the INA226 power pin).

3.2 I2C Bus Validation & Address Scanning

Before configuring specific ESPHome sensor components, we utilize ESPHome's built-in I2C scanner to verify that the bus is electrically healthy and that the INA226 is responding.

1. Set your bench power supply to **12.0V** (Current limit at **300mA**).
2. Connect the Main Module to your PC via USB or power it up via the 12V inputs if wireless logging is established.
3. Open the ESPHome clean logs console.
4. Look closely at the boot sequence. Because scan: true is enabled under the i2c component in your YAML config, the ESP32-C6 will query every possible address on the bus.

Expected Log Output:

You should see a compilation log line indicating that a device has been successfully discovered:

Plaintext

```
[I][i2c.arduino:143]: Found I2C device at address 0x44
```

ℹ **Note on I2C Addressing:** The INA226 hardware address on this PCB is hardwired to **0x44** based on its physical address pin configurations. If the log reads 0x44, your I2C hardware bus lines (SDA on GPIO6 and SCL on GPIO7) along with their pull-up resistors are 100% correct.

Section 4: Wired Networking Interface (W5500 Ethernet Assembly)

In this stage, you will populate the high-speed wired networking interface. By adding the W5500 Ethernet controller and its isolated magnetic RJ45 interface, the Main Module gains a highly reliable, low-power network connection optimized for hardwired vehicle bus communication.

4.1 Components to Solder in this Step

Solder the following components onto the PCB:

Component Type	Designators	Value / Specification
Integrated Circuit	U5	W5500 Ethernet Controller
Connectors & Filters	J9	RJ45 MagJack Connector (with internal magnetics)
	FB1	Ferrite Bead (600 Ω @ 100MHz)
Capacitors	C10, C12, C22, C23, C24, C25, C26	10uF
	C15	4.7uF
	C13, C27	1uF
	C14	10nF
	C9, C28, C29	100nF
	C17, C18	10pF *(Load capacitors for the crystal)*
	C21	10nF / 2kV (Size: 1206)

Component Type	Designators	Value / Specification
	C5	10uf
Resistors	R22	1M
	R17	12K4 (1%) *(Critical EXRES bias resistor)*
	R9, R10, R23, R28	10K
	R13, R14, R15, R16	49.9 ohm(1%)
	R11, R12	330 ohm *(Optional: MagJack status LEDs)*
	R18, R29, R30, R31, R32	0 ohm (Jumpers)

💡 Camper Integration Tip (Stealth Mode):

To prevent annoying light pollution inside the camper cabin at night, you can choose not to populate R11 and R12. Leaving these two resistor pads empty completely disables the physical RJ45 MagJack status LEDs without affecting network data functionality.

4.2 Quality Control & Pre-Power Checklist

Because the W5500 layout contains fine-pitched pins and sensitive clock circuitry, perform these verification steps with your multimeter before applying power.

Step 1: Crystal Pinout Orientation Check

Most standard 3225-sized 25MHz passive crystals feature a diagonal pin configuration (Pins 1 and 3 are connected to the crystal, while Pins 2 and 4 are hardwired to the internal metal casing for grounding).

Step 2: XO Rail Short-Circuit Check

- Set your multimeter to Continuity/Diode mode (Beep test).
- Measure between GND and the XO trace (Pin 31 of the W5500).
- Expected Result: The meter must not beep. If a short circuit to GND is detected here, the crystal cannot oscillate, and the W5500 chip will fail to boot.

4.3 ESPHome Software Configuration

To activate the Ethernet interface and transition away from wireless operation, update your ESPHome YAML configuration file with the following block.

```
ethernet:
  type: W5500
  clk_pin: 18
  mosi_pin: 19
  miso_pin: 20
  cs_pin: 1
  interrupt_pin: 2
  reset_pin: 0
  clock_speed: 26MHz # Can be lowered to 8MHz during initial hardware validation
```

⚠ **Critical Step: Disable Wi-Fi and Fallback AP**

When shifting your node over to a dedicated wired Ethernet connection, you must completely comment out or remove the wifi: component and any ap: (Fallback Access Point) blocks from your active ESPHome configuration.

```
# Remove or comment out these wireless blocks completely:
# wifi:
#   networks:
#     - ssid: "YourWiFi"
#       password: "YourPassword"
# captive_portal:
```

4.4 Functional Validation & Expected Logs

- Set your bench power supply to 12.0V (Current limit at 300mA).
- Plug an active network cable from your router/switch into the RJ45 MagJack.
- Boot up the module and observe the ESPHome clean logs console.

Successful Outcome:

The log output should display a successful hardware handshake, followed by a rapid transition into full link speed and a valid DHCP IP assignment:

If you see Link Speed: 100 and Full Duplex: YES, your W5500 controller is perfectly soldered, clocked, and ready for off-grid operations!

Section 5: Fieldbus Interface (MAX13487 RS485 Assembly)

In this stage, you will populate the RS485 communication backbone. Utilizing the MAX13487 transceiver with automatic hardware direction control eliminates the need for a dedicated TX/RX toggle pin in the software application layer.

5.1 Components to Solder in this Step

Solder the following components onto the PCB:

- **IC:** U2 - MAX13487 RS485 Transceiver
- **Resistors:** R19 (10K, R20 (4.7K, R21 (4.7K)
- **Capacitor:** C16 (100nF decoupling)
- **Protection Diodes:** D1, D4 (SMBJ6.8 Transient Voltage Suppressor)
- **Overcurrent Protection:** F4, F5 (0.1A Resettable Fuses / PTCs)

5.2 RS485 Bus Termination

- **R27 120ohm:** This termination resistor must **only** be populated or jumpered if this module is configured as the absolute first or absolute last node on the linear physical network chain. For all intermediate nodes passing through, leave this resistor unpopulated to prevent signal loading. In this case the Master Unit is your first module, so must be populated.

Section 6: Local Audio Indicator (Buzzer) & Topboard Interface

In this final SMD assembly stage, you will populate the acoustic alert circuit and the inter-board connector that links the Main Module to the Topboard UI.

6.1 Components to Solder in this Step

Solder the following components onto the PCB:

Component Type	Designators	Value / Specification
Acoustic Indicator	LS1	Magnetic / Piezo Buzzer (5V active)
Transistor	Q2	N-Channel MOSFET (or NPN Transistor, e.g., BSS138 / BC817)
Resistor	R26	$1\,k\,\Omega$ *(Base/Gate current limiting resistor)*
Connector	X1	Topboard Interconnect Header (SMD Male/Female)

6.2 ESPHome Buzzer Validation

- To test if the transistor circuit opens cleanly and drives the buzzer, you can implement a simple RTTTL or automated switch component linked to the GPIO pin driving R26.

YAML

```
output:
  - platform: ledc
    pin: GPIO21
    id: buzzer_output

rtttl:
  output: buzzer_output
  id: buzzer_audio

# Test trigger via a simple template button in Home Assistant
button:
  - platform: template
    name: "Beep Test"
    on_press:
      - rtttl.play: "beep:d=4,o=5,b=120:c"
```

🎉 SMD Assembly Complete!

With X1 and the buzzer circuit in place, all surface-mount devices (SMD) are officially completed. Before moving on to the bulkier through-hole components (THT) like terminal blocks, fuse holders, and RJ45 jacks, it is highly recommended to perform one final, thorough inspection of the board using a magnifying glass or microscope to catch any loose solder balls or flux residue.

Section 7: Topboard UI & Antenna Assembly

The Topboard acts as the user interface and wireless transceiver extension for the Main Module. It breaks out the local display, user input, visual indicators, and the external antenna routing via an IPEX-to-SMA link.

7.1 Components to Solder in this Step

Component Type	Designators	Value / Specification
Tactile Switch	S1	Push button *(User Input)*
Resistors	R1, R5	$10\,\mathrm{k}\,\Omega$ *(Pull-up/down)*
RF Connector	U2	IPEX MHF1 Connector *(For internal/external antenna link)*
FPC Interface	FPC1	Flexible Printed Circuit Connector *(Main interconnect)*

Component Type	Designators	Value / Specification
LED	D1	$3\,\mathrm{mm}$ Status LED

Component Type	Designators	Value / Specification
Antenna Port	J2	SMA/RP-SMA Edge-Launch Connector
Display Module	U1	OLED Display (I2C interface, e.g., SSD1306 / SH1106)

Dit is de perfecte afsluiter voor de complete handleiding. Hiermee transformeer je de twee losse, gesoldeerde printplaten tot één robuuste, trillingsbestendige module die klaar is voor de camper.

Omdat dit om de mechanische passing en behuizing gaat, schrijven we dit strak, no-nonsense op in dezelfde professionele stijl. We verwerken ook direct de twee opties voor de montage-beugels (DIN-rail of schroefgaten).

Section 8: Final Mechanical Assembly & Enclosure Integration

In this final stage, you will mechanically combine the Main PCB and the Topboard, mount the protective cover, and slide the complete sub-assembly into its modular extrusion profile housing.

8.1 Hardware Requirements

Ensure you have the following mechanical fasteners ready:

- **Standoffs**: 4x M2 x 20mm Male-Female (M-F) brass or nylon threaded studs / standoffs.
- **Screws**: Corresponding M2 screws for securing the top layers.
- **End Caps / Side Pieces**: Your chosen mounting variant:
  - *Option A*: DIN-rail mountable side pieces (for central electrical cabinet installation).
  - *Option B*: Standard side pieces with screw flanges (for direct wall/surface mounting).

8.2 Assembly Procedure

Step 1: Board Interconnect & Standoff Installation

1. Take the completed **Main PCB** and screw the four **M2 x 20mm Male-Female standoffs** into the designated mounting holes from the top side. Tighten the nut firmly by hand or with small pliers.
2. Carefully insert the FPC strip into the connector of the Topboard and into the connector on the Main PCB.

Step 2: Cover Installation

1. Place the 3D-printed **protective cover** over the Topboard.
2. Secure the cover and Topboard stack down using M3 screws into the sides.

Step 3: Enclosure Rail Integration

1. Slide the combined double-decker PCB assembly smoothly into the designated guide slots of the PCB holder.

Step 4: Side Piece Mounting

1. Position your chosen side pieces (either the DIN-rail variant or the surface-screw variant) onto the open ends of the PCB holder.
2. Secure the side pieces using the housing assembly screws to lock the PCB securely inside the enclosure, protecting it against vehicle vibrations.