Soldering Manual Relay Module.

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

Before populating any sensitive downstream components, you must assemble, solder, and test the power input and local 3.3V regulation stage. This ensures your Relay Module handles power correctly and prevents accidental damage to future integrated components.

1.1 Components to Solder in this Step

Solder only the following components onto the PCB:

- **U1**: L78L33 Linear Voltage Regulator
- **C2, C3**: 22µF Ceramic Input Filter Capacitors
- **C9**: 1µF Input Capacitor
- **C10**: 10µF Output Smoothing Capacitor
- **C1**: 10µF Decoupling Capacitor
- **J1 & J6**: JST Input Connectors

1.2 Quality Control & Bench Testing Procedure

Once these initial components are securely soldered, prepare a custom test wire to safely introduce power to the board.

Step 1: Test Wire Preparation

1. Take a standard JST mating cable.
2. Connect **Pin 1** of the JST connector to the **+12V** line of your laboratory power supply.
3. Connect **Pin 2** of the JST connector to the **GND** (Negative) line of your power supply.

Step 2: Voltage Regulation Test (Normal Power)

1. Set your bench power supply to **12.0V** and limit the current to **100mA** as a safety precaution.
2. Plug the test wire into the soldered JST connector (**J1** or **J6**) to power up the rail.
3. Take your multimeter and set it to **DC Voltage mode**.
4. Place the black probe on a known **GND** point and the red probe on the output pin of the **L78L33** (or across the pads of **C10**).

📋 **Expected Result**

The multimeter must display a stable **3.3V**. If you read significantly lower voltage or 0V, turn off the power immediately and check for solder bridges or a reversed U1 orientation.

Section 2: Logic Control, Relay Driver & Address Selection (SMD & IC Assembly)

In this stage, you will assemble the core digital control section of the Relay Module. This includes the I2C I/O expander, the inverting buffer, the Darlington transistor array, and the hardwired I2C address configuration.

2.1 Components to Solder in this Step

Carefully solder the following surface-mount (SMD) and integrated circuit (IC) components onto the board. Ensure correct pin 1 orientation for all ICs (aligning the dot or notch with the silkscreen indicator):

- **U3 (PCF8574)**: Remote 8-Bit I/O Expander for I2C-bus.
- **U5 (74HC540D)**: 8-Bit Inverting Buffer/Line Driver.
- **U6 (ULN2803A)**: 8-Channel Darlington Transistor Array (Relay Driver).
- **C5, C6, C7**: 100nF Ceramic Decoupling Capacitors.

2.2 Hardware I2C Address Selection

Before proceeding to any testing, you must physically set the I2C address of the module using the onboard solder jumpers (**A0, A1, A2**).

- Bridge the solder pads according to your desired address mapped out in the general address section of this manual.
- Ensure the solder joint is clean, fully bridges the intended pads, and does not accidentally short to neighboring traces.

2.3 Quality Control & Logic Rail Check (Power OFF & ON)

Before installing the physical mechanical relays, perform a quick check to verify the logic stage is safely integrated.

Step 1: Visual Inspection (Power OFF)

1. Use a magnifying glass or microscope to inspect the pins of **U3, U5, and U6**.
2. Verify there are no solder bridges between the fine-pitch pins.
3. Confirm that the orientation of all three ICs matches the PCB silkscreen.

Section 3: Safe Communication & Bus Verification (NodeMCU Bench Test)

3.1 Temporary Testing Interconnect & Wire Harness

To safely test the module without risking your Master Unit, construct a temporary test wire using a 4-pin JST mating cable connected to an external 12V laboratory power supply and a temporary ESP32 NodeMCU (powered via USB).

⚠ CRITICAL POWER RULES:

- DO NOT connect the NodeMCU's 3V3 pin to the Relay Module. The Relay Module generates its own 3.3V locally via U1. Connecting two active 3.3V regulators in parallel can destroy both devices.
- The NodeMCU must be powered via its own USB cable, but it MUST share a common ground (GND) with the laboratory power supply to allow I2C data signals to pass cleanly.

Wire Harness Pinout Configuration:

The 4-pin JST connector plugs directly into the Relay Module system bus (J1 or J6). The open ends of the wire harness must be interconnected as follows:

- JST Pin 1 (+12V): Connect directly to the +12V DC positive terminal of your laboratory power supply.
- JST Pin 2 (GND): Connect to the GND negative terminal of the laboratory power supply AND bridge it directly to a GND pin on the NodeMCU.
- JST Pin 3 (SDA): Connect directly to GPIO 4 on the NodeMCU.
- JST Pin 4 (SCL): Connect directly to GPIO 5 on the NodeMCU.

💡 *Required I2C Bus Pull-Up Resistors:*

Because this test is performed on an isolated bench setup without the Master Unit's built-in hardware bus pull-ups, you must manually introduce pull-up resistors to keep the open-drain I2C bus operational:

1. Connect a 4.7kΩ resistor between the SDA line (GPIO 4) and the 3V3 output pin of the NodeMCU.
2. Connect a second 4.7kΩ resistor between the SCL line (GPIO 5) and the 3V3 output pin of the NodeMCU.

3.2 Verification Procedure

Step 1: Power Sequencing

1. Power up the Relay Module by connecting **12V** and **GND** to the JST input (as done in Section 1).
2. Connect the ESP32 NodeMCU to a power supply USB cable.
3. Ensure both systems share the common GND wire from the step above.

Step 2: Running the I2C Scanner

1. Flash a standard I2C Scanner sketch (via the Arduino IDE or ESPHome) onto the NodeMCU.
2. Open the Serial Monitor at the designated baud rate.

📋 **Expected Result**

The Serial Monitor should successfully discover an I2C device at the specific hex address you configured with the solder jumpers in Step 2 (e.g., 0x20).

- **If no device is found:** Check your solder joints on the SDA/SCL lines, check U3 (PCF8574) pins for bridges, and verify the solder jumpers are fully closed.
- **If the NodeMCU drops offline:** Disconnect USB immediately. This indicates a short circuit between the data lines or GND.

Section 5: Power Actuators & High-Current Terminals (Relay Assembly & Mechanical QC)

In this stage, you will mount and solder the heavy physical switching components. Because these components handle high-current DC loads for camper appliances (up to 10A/16A depending on your relay rating), proper solder thermal mass and structural integrity are critical.

5.1 Assembly Prerequisite Check

⚠ **CRITICAL:** Before soldering the relays, install the topconnector X1 first, as it is positioned very tight in between the relays.

5.2 Components to Solder in this Step

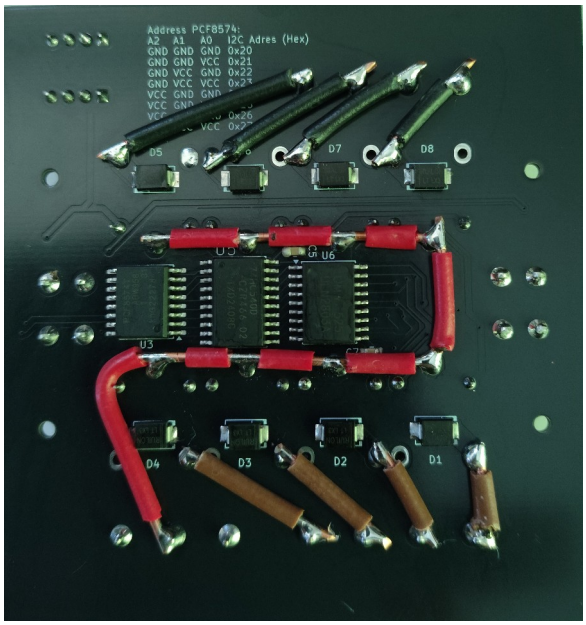
Solder the following through-hole technology (THT) components onto the mainboard:

- **K1 to K8**: 12V Mechanical Relays
- **Screw Terminals**: Heavy-duty terminal blocks for the input distribution and the individual relay output channels (Normally Open / Common paths).

5.3 Copper Wire Trace Reinforcement Procedure (Critical for 20A Capacity)

Although the PCB layout includes dedicated 12V copper planes, continuous high-current loads over time can cause thermal stress and overheat the traces. To ensure maximum safety and eliminate trace resistance, you must reinforce the high-power paths using solid copper wire:

1. Strip a piece of standard 1.5mm² solid copper wire (household installation wire).
2. Bend and pre-form the bare copper wire so it lays perfectly flat along the exposed high-current trace tracks on the bottom/top of the PCB.
3. Solder the 1.5mm² copper wire directly between the main 12V external power supply input terminal and the relay contact points.
4. Solder a second reinforced copper wire path between the output contact points of the relays and their corresponding screw terminal blocks.



5.4 Optional Transient Voltage Protection (TVS Diode Assembly)

The PCB layout includes dedicated solder pads for an optional, but highly recommended, **SMBJ18A TVS (Transient Voltage Suppressor) diode** for each relay channel.

Why this is recommended:

When switching **inductive loads** inside your camper (such as 12V water pumps, compression refrigerators, or heavy ventilation fans), a massive high-voltage spike (*flyback voltage*) is generated every time a relay opens. Without protection, this voltage spikes across the circuit, causing micro-arcing that prematurely burns out the physical relay contacts and introduces electrical noise to the logic bus. The SMBJ18A clamps these dangerous spikes instantly, safely dissipating the energy.

Section 7: Topboard User Interface Assembly (SMD & THT)

The Topboard contains the physical user interface components, including the manual override slide switches, the status LEDs, and their corresponding current-limiting resistors. The final component on this board is the Flexible Printed Circuit (FPC) connector.

7.1 Components to Solder in this Step

Solder the components onto the Topboard PCB in the following order.

1. **Resistors**: Small SMD current-limiting resistors for the LEDs.
2. **Status LEDs**: Through-hole LEDs (Ensure the correct polarity! The longer leg is the positive Anode, matching the '+' or longer pad on the silkscreen).
3. **Slide Switches**: The physical manual override switches. Ensure they sit completely flush and square against the PCB surface before soldering the structural anchoring pins.
4. **FPC Connector**: Solder the Flexible Printed Circuit connector with care. Ensure there are no micro-solder bridges between the fine-pitch pins.

Section 8: Final Mechanical Assembly & Enclosure Integration

In this final stage, you will mechanically combine the completed 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.
- **Housing**: Aluminum/Plastic extrusion PCB holder (profile shell).
- **End Caps / Side Pieces**: Your chosen mounting variant (DIN-rail mountable or standard screw holes).

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 corner mounting holes from the top side.
2. Connect the **FPC cable** into the FPC connectors to bridge the data/logic lines between both boards (if applicable), and carefully align the female header pin interface of the **Topboard** with the male **X1 interconnect header** on the Main PCB.
3. Press the Topboard straight down with even pressure until it sits completely flush on top of the 20mm standoffs and the electrical connection is fully seated.

Step 2: Cover Installation

1. Place the acrylic or 3D-printed **protective cover** over the Topboard, aligning its holes with the exposed threads of the standoffs.
2. Secure the cover and Topboard stack down using M2 screws into the standoffs. Do not over-tighten to prevent cracking the cover.

Step 3: Enclosure Rail Integration

1. Slide the combined double-decker PCB assembly smoothly into the designated guide slots of the **PCB holder (extrusion profile)**.
2. Ensure the board slides all the way back until the ports and connectors align perfectly with the front/rear orientation.

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.