Bike Safety System — Schematic & KiCad Layout

This document contains a complete, ready-to-follow schematic description, BOM, power-distribution and PCB layout guidance tailored for your bike safety/security system (ESP32 + NEO-M8N + MPU6050 + optional GSM). Use this as the source-of-truth when creating the KiCad schematic and PCB.

---

1) Design goals (short)

- Single 12 V input from bike. Robust input protection against reverse polarity and transients.
- Primary 5 V bus from a buck converter (automotive-capable) sized for bursts.
- Clean 3.3 V rail (LDO) for ESP32 + IMU + GNSS (optionally use module breakout 5→3.3V if it has low-noise regulator).
- Optional dedicated GSM power handling (large reservoir caps and/or dedicated regulator).
- Minimize switching noise coupling into GNSS/antenna and IMU.

---

2) High-level schematic (textual — map this directly into KiCad symbols)

Connectors

- **J1: BAT_IN** — 2-pin screw terminal (VIN+, GND)
- VIN+ → protection (reverse diode / MOSFET) → TVS → bulk cap → BUCK_IN
- GND → system ground
- **J2: ANT_GPS** — u.FL or SMA depending on antenna (connect to NEO-M8N ANT pad)
- **J3: USB_PROG** — USB-C micro B as optional programming / serial power (connect via ideal-diode OR allow USB to power 5 V bus when present)
- **J4: OUTPUTS** — Relay/Buzzer (open-drain via transistor)
- **J5: SIM_PWR** — if using external SIM battery connector

Power block

- **D1** — Reverse-protection (Schottky or ideal diode MOSFET assembly). Example footprint: SOD-128 for Schottky or MOSFET SOT-23.
- **TVS1** — Automotive TVS (e.g., SMAJ58A or equivalent) across VIN after D1.
- **C_IN** — Bulk electrolytic 220 μF — 1000 μF (low ESR) at VIN.
- **LDO1** — OPTIONAL: 5 V input soft-start or inrush limit circuit (NTC or small series resistor + MOSFET) if huge caps present.
- **BUCK1** — 12 V → 5.0 V buck converter. Rating: choose module/IC capable of continuous 3 A (or rated per your GSM/ESP peaks). Add **LC filter** at its output: ferrite bead F1 + L1 (10–47 μH) + C_OUT (100 μF electrolytic + 10 μF tantalum + 0.1 μF MLCC).
- **C_5V_local** — 470 μF low-ESR on 5 V bus near distribution.

3.3 V domain

- **LDO3V3** — Low-noise LDO (e.g., TLV70033, MIC550, or similar). Input: 5 V bus after LC filter. Provide 10 µF + 0.1 µF close to LDO in/out.
- **C_3V3_local** — 100 µF low-ESR + MLCCs near ESP32.
- 3.3 V net connects to: ESP32 VCC, MPU6050 VCC, (optionally) NEO-M8N VCC if using direct 3.3 V rather than onboard regulator.

Highest-noise device (GSM)

- If using **SIM800L / BG95 / etc.**: either create separate regulated 4 V rail or feed from 5 V bus but:
- Add **C_GSM**: 2200 µF–4700 µF low-ESR cap right at module power pins.
- Add a series sense resistor or PFET disconnect and a polyfuse (resettable) so bursts don't pull down system.
- Place GSM power/antenna connectors at the board edge, physically separated from GNSS.

Controller & sensors

- **U1: ESP32-WROOM** (or WROVER if you need PSRAM) — 3.3 V VCC. Expose EN, IO0, TX0/RX0 for programming. Add 10 k pull-ups/pull-downs per reference schematic. Add decoupling: 100 nF + 10 µF.
- **U2: MPU6050** — I2C (SDA, SCL) to ESP32 I2C pins. Add 2.2 k pull-ups to 3.3 V if module lacks them. Decouple with 0.1 µF + 4.7 µF.
- **U3: NEO-M8N (GNSS)** — UART (TX/RX) to ESP32 Serial1 (use hardware serial). Also PPS pin optional to ESP32 GPIO. Power: 3.3 V is preferred. Antenna to J2. Decouple and add LC filter close to module: ferrite bead + 10 µF + 0.1 µF.

Misc

- **Level translators**: not needed if all are 3.3 V. If any module is 5 V tolerant only (LCD), use level shifters.
- **Buzzer / Relay**: NPN transistor (e.g., 2N2222 or SMD equivalent) with base resistor (10 k pull-down or 100 k?), flyback diode for coils, and emitter to GND. Drive from ESP32 GPIO through base resistor (4.7k).
- **LEDs**: status LED with 4.7k series resistor.

---

3) BOM (suggested — verify footprints)

Ref	Part	Value / Notes	Qty
J1	Screw terminal	2-pin, 5.08 mm pitch	1
D1	Schottky	40 V, SOD-128	1
TVS1	TVS	600 W, 58 V standoff or automotive-rated	1
C_IN	Electrolytic	470 µF, 25 V low-ESR	1

Ref	Part	Value / Notes	Qty
BUCK1	Buck module	5 V, 3 A (module or integrated IC + inductor)	1
L1	Inductor	10–47 µH (for LC)	1
F1	Ferrite bead	SMD ferrite (1206 or 0805)	1
C_OUT	Electrolytic	220 µF, 10 V low-ESR	1
LDO3V3	LDO	3.3 V low-noise LDO, SOT-223 or SOT-23	1
C_3V3	MLCC	10 µF + 0.1 µF	2
U1	ESP32 module	WROOM or DEVKIT footprint	1
U2	MPU6050	QFN breakout or module	1
U3	NEO-M8N	GNSS breakout module	1
U4	SIM800L	if used	1
Q1	NPN	SOT-23 transistor for buzzer/relay	1
Rb	Resistors	4.7 k, 10 k etc	several

Note: exact package choices depend on your footprint preferences for soldering (hand-solder vs reflow). I recommend through-hole for connector/screw terminals and SMD for everything else.

---

4) PCB layout plan & rules (KiCad)

Layer stackup

- 2-layer board is fine for this project. Use:
- Top: components, routing
- Bottom: ground plane + routing
- Recommend at least 35 µm copper (1 oz). If you expect >2 A continuous on some traces, consider 2 oz copper or heavier.

Placement priorities

1. **Input protection & buck**: place near J1 (VIN). Keep switching components local to minimize noise loops.
2. **Bulk caps**: place right at buck input & output.
3. **GSM module**: place at board edge, keep away from GNSS and ESP32 antenna path.
4. **GNSS module & antenna**: place on board corner/edge with a keep-out area around antenna. Keep it physically distant (≥30 mm ideally) from buck and GSM.
5. **ESP32**: central, but not next to buck or GSM. Keep its antenna clear (if module has antenna).
6. **IMU**: close to ESP32 to shorten I2C traces; avoid being next to switching loops.
7. **Test pads & headers**: program pins, 5 V, 3.3 V, GND, and serial lines.

Grounding

- Use a continuous ground plane on bottom.
- Route high-current return paths directly under their supply traces when possible so the plane carries return current.- Use stitching vias if you have localized ground islands.
- Avoid splitting the ground plane under the GNSS antenna area.

Trace widths (conservative recommendations)

- Use these conservative widths on 1 oz copper:
- Signals (I2C, UART, GPIO): 0.25 mm (10 mil)
- 0.5 A traces: 0.8 mm
- 1 A traces: 1.5 mm
- 2 A traces: 2.5 mm
- 3 A traces: 3.0–3.5 mm
- For the 5 V bus feeding multiple modules, use a wide plane or multiple 3.0 mm traces in parallel.

Decoupling & filtering layout

- Put MLCC 0.1 μF as close as possible to each IC Vcc pin.
- Place larger electrolytic/tantalum caps near the module entry points.
- Place ferrite bead + LC filter components in series with the 5 V feed to GNSS and ESP32 LDO input.

Thermal & mechanical

- If buck IC is in SMD, provide thermal vias under its pad connected to ground plane per datasheet.
- Keep mounting holes and mechanical pads clear of the antenna area.

---

5) KiCad project checklist (step-by-step)

1. Create new project in KiCad and set board units to mm.
2. Create schematic sheets: PWR, CONTROLLER, SENSORS, COMM, MECH.
3. Add symbols and annotate. Assign footprints (SOT-23, SOT-223, 0805, 1206, etc.).
4. Add net labels: VIN, GND, 5V_BUS, 3V3, UART1_TX, UART1_RX, I2C_SDA, I2C_SCL, PPS.
5. Run ERC and resolve warnings.
6. Generate netlist and import to PCBNew.
7. Place components following placement priorities above.
8. Create ground zone filled on bottom. Set thermal relief on through-hole pads.
9. Route high-current tracks first (5 V, VIN, GND returns). Keep them short and wide.
10. Add decoupling and testpoints. SILK notes for antenna keep-out and orientation.
11. DRC and Gerber generation.

---

6) Testing & debug suggestions

- Bring up the board with a current-limited bench supply and scope on the 5 V & 3.3 V rails. Watch for droops during ESP32 Wi-Fi TX and (if present) GSM bursts.
- Verify GNSS fix stability with the antenna placed in intended mounting location before sealing the enclosure.
- Check temperature of buck and LDO on continuous operation.
- If you see jitter in GNSS, add more LC filtering or move buck further away.

---

7) Files & footprints I recommend creating in KiCad

- Footprints: 2-pin screw terminal, u.FL, SOT-23 (D1), SOT-223 (LDO), 1206/0805 (passives), 3.3 V regulator SOT-23-5 for small LDOs.
- Symbols: ESP32 module symbol with nets exposed (EN, IO0, IO2, GND, 3V3, TX0, RX0).

If you want, I can now: - produce a **schematic PDF / SVG** layout text that you can import or copy into KiCad, or - produce a **detailed KiCad component placement and routing plan** (with exact footprints and grid coordinates) for an assumed board size (e.g., 60×40 mm).

Tell me which of those two you want next and I will put it into the same canvas as a follow-up file.