# SX1262 Telemetry Module — KiCad/Eagle Symbol Pin Mapping & Single-Sheet Schematic

**Target:** EBYTE E22-900M30S (SX1262 SPI module) + STM32F103C8T6 MCU + 3.3V regulator + Pixhawk DF13 6-pin TELEM connector

**Deliverables in this document** - One-page schematic (symbol pin mapping and net names) ready to copy into KiCad/Eagle - Pin-to-pin mapping for the EBYTE module, STM32F103C8T6, and DF13 telem connector - Suggested KiCad net names and footprints - Single-sheet BOM (short) and PCB notes for layout

---

## 1) Global nets (use these KiCad net names)

- **3V3** — Main 3.3 V rail
- **GND** — Ground plane
- **VBAT** — Optional battery input (before regulator)
- **SPI_SCK** — SPI clock (MCU PA5)
- **SPI_MOSI** — SPI MOSI (MCU PA7)
- **SPI_MISO** — SPI MISO (MCU PA6)
- **RADIO_CS** — Radio chip select / NSS (MCU PA4 or any GPIO)
- **RADIO_BUSY** — SX1262 BUSY (input to MCU PB1)
- **RADIO_DIO1** — SX1262 DIO1 (interrupt input to MCU PB0)
- **RADIO_RESET** — SX1262 RESET (MCU PC13)
- **UART_TX** — MCU -> Pixhawk RX (PA9)
- **UART_RX** — MCU <- Pixhawk TX (PA10)
- **ANT** — Antenna RF pad / connector

Use consistent naming in KiCad symbols to make netlist and PCB wiring obvious.

---

## 2) EBYTE E22-900M30S module — Pin mapping (KiCad symbol)

> Use the EBYTE datasheet for exact pad numbering. The following lists logical pins and suggested symbol names.

| Symbol name | KiCad pin name | Type | Notes |
|---|---|---|---|
| VCC | VCC | PWR | Connect to 3V3 |
| GND | GND | PWR | Connect to GND |
| SCK | SPI_SCK | I/O | HW: connect to MCU SPI SCK (PA5) |
| MISO | SPI_MISO | I/O | HW: connect to MCU SPI MISO (PA6) |

| Symbol name | KiCad pin name | Type | Notes |
|---|---|---|---|
| MOSI | SPI_MOSI | I/O | HW: connect to MCU SPI MOSI (PA7) |
| NSS | RADIO_CS | I/O | Active low chip select (MCU GPIO PA4) |
| BUSY | RADIO_BUSY | Input | Connect to MCU input (PB1) — poll or check before SPI |
| DIO1 | RADIO_DIO1 | Input/ IRQ | Connect to MCU external-interrupt pin (PB0) |
| DIO2 | RADIO_DIO2 | Input/ IRQ | Optional |
| RESET | RADIO_RESET | Output | Active-low reset from MCU (PC13) |
| ANT | ANT | RF | Route to U.FL/SMA via 50Ω |

*Footprint:* place as a single module (SMD module footprint 24×38 mm — exact mechanical from EBYTE datasheet).

---

## 3) STM32F103C8T6 suggested pin usage & KiCad symbol mapping

*Use an LQFP48 symbol and map the nets below. Replace pin numbers according to the chosen MCU package footprint.*

| MCU signal | MCU pin (symbol) | Net | Function |
|---|---|---|---|
| PA5 | SPI_SCK | SPI_SCK | SPI1 SCK |
| PA6 | SPI_MISO | SPI_MISO | SPI1 MISO |
| PA7 | SPI_MOSI | SPI_MOSI | SPI1 MOSI |
| PA4 | RADIO_CS | RADIO_CS | SPI chip select (GPIO) |
| PB1 | RADIO_BUSY | RADIO_BUSY | SX1262 BUSY (input) |
| PB0 | RADIO_DIO1 | RADIO_DIO1 | SX1262 DIO1 (EXTI) |
| PC13 | RADIO_RESET | RADIO_RESET | SX1262 RESET (active low) |
| PA9 | UART_TX | UART_TX | USART1 TX -> DF13 Pin3 (Pixhawk RX) |
| PA10 | UART_RX | UART_RX | USART1 RX <- DF13 Pin2 (Pixhawk TX) |
| VDD | VCC | 3V3 | connect to regulator |
| VSS | GND | GND | ground plane |

| MCU signal | MCU pin (symbol) | Net | Function |
|---|---|---|---|
| SWDIO | SWDIO | SWD | programming header |
| SWCLK | SWCLK | SWD | programming header |

*Notes:* - Mark PB0 as an external interrupt capable pin (EXTI0) in the schematic symbol. - Provide decoupling caps near VDD pins: 0.1uF + 10uF recommended.

---

## 4) DF13 / Pixhawk 6-pin TELEM connector mapping

Pixhawk TELEM common wiring (confirm for your board variant). Use either DF13 or JST-GH depending on frame.

| DF13 pin | Signal | KiCad net | Notes |
|---|---|---|---|
| Pin1 | VCC (+5V) | (optional) VIN | Do not power radio directly from Pixhawk unless designed for it. |
| Pin2 | TX (Pixhawk->FC TX) | UART_RX | Connect to MCU UART_RX (PA10) |
| Pin3 | RX (Pixhawk<-FC) | UART_TX | Connect to MCU UART_TX (PA9) |
| Pin4 | CTS | (optional) | Not used for basic setup |
| Pin5 | RTS | (optional) | Not used for basic setup |
| Pin6 | GND | GND | Common ground |

*Label connector footprint DF13-6P or JST-GH 6-pin depending on your planned cable.*

---

## 5) Power/regulator & decoupling

- Use a switching regulator if powering from battery (e.g., step-down to 3.3V, rated ≥ 800mA to support high TX bursts if module requests it). KiCad net: VBAT -> VIN of regulator -> 3V3 net.
- Add bypass capacitors near regulator output: 10uF (X7R) + 0.1uF close to module and MCU.
- Reset pull-ups: add 10k pull-up on MCU NRST if required by your microcontroller symbol.

Suggested regulator footprints: SOT-23-5 (for LDO) or LGA/SOT-23-6 for switching regulators. Include thermal vias under the regulator.

---

## 6) Single-sheet schematic layout (textual order for drawing in schematic tool)

1. **Power block**: VBAT -> VIN connector -> regulator -> 3V3 -> decoupling caps -> nets to MCU & E22 VCC
2. **MCU block**: LQFP48 symbol with labeled nets (VDD, VSS, PA5, PA6, PA7, PA4, PB1, PB0, PA9, PA10, SWD pins)
3. **Radio module block**: EBYTE module symbol with pins named (VCC, GND, SCK, MISO, MOSI, NSS, BUSY, DIO1, RESET, ANT)
4. **Connector block**: DF13 6-pin with pin labels and nets
5. **Misc**: LED indicators (PWR, TX, RX) with resistors, reset button, test points, SWD header, U.FL/SMA

Draw wires according to the net names in section 1. Attach text labels for nets rather than long wires where convenient.

---

## 7) Suggested footprints & BOM (compact)

- **Module**: EBYTE E22-900M30S — pad module footprint 24×38.5 mm (vendor mechanical drawing). Use module connector pads per datasheet.
- **MCU**: STM32F103C8T6 — LQFP48 footprint
- **Regulator**: MP1584/RT8059 or equivalent (SOT-23-5 / SOT-23-6) — choose based on power.
- **Connector**: DF13 6-pin (or JST-GH 1.25mm 6-pin) — PCB vertical right-angle as needed
- **RF connector**: U.FL PCB JACK (surface mount) or SMA bulkhead (edge mount) — use 50Ω transmission routing
- **LEDs**: 0603
- **Capacitors**: 10uF X7R (1206) ×2, 0.1uF (0603) near MCU & module
- **Resistors**: 10k pull-ups ×2, LED series resistors ×3 (2.2k or 4.7k for 3.3V)

**Short BOM table** | Ref | Qty | Part | Footprint | |---|---:|---|---| | U1 | 1 | EBYTE E22-900M30S | Module 24×38 mm | U2 | 1 | STM32F103C8T6 | LQFP48 | U3 | 1 | 3.3V regulator (switching recommended) | SOT-23-5 | J1 | 1 | DF13/JST-GH 6-pin | Connector 6-pin | J2 | 1 | U.FL (or SMA) | RF connector | C1 | 2 | 10uF X7R | 1206 | C2 | 2 | 0.1uF | 0603 | R1 | 2 | 10kΩ | 0603 | LED1..3 | 3 | 0603 LED | 0603 | SWD | 1 | 2x5 SWD header (optional) | 2x5 2.54mm

---

## 8) PCB & RF layout reminders

- Keep a 50Ω microstrip between module RF pad and U.FL/SMA. If the EBYTE module has its own antenna, follow vendor recommendations and keep the module antenna area clear.
- Place decoupling directly at MCU and module VCC pins. Use ground vias under radio module area.
- Provide test pads for SPI lines, DIO1, BUSY, and UART for debugging.

---

## 9) Follow-up options

If you'd like, I can now: - Produce a **KiCad Schematic .sch text mapping** (component symbols & net names) that you can paste into KiCad, or - Produce a **one-page PDF** schematic visual (single-sheet) rendered from this mapping.

Tell me which you want and I will generate it next.

---

*End of document.*