Approval:	**Global Application Engineering Manager** *Massimiliano Lonzar*
Approval:	**Design Services Manager** *Fabio Deperini*
Author:	**Technical Support Engineer** *Marino J. Marson*

DISCLAIMER

The contents are confidential and any disclosure to persons other than the officers, employees, agents or subcontractors of the owner or licensee of this document, without the prior written consent of Telit, is strictly prohibited. Telit makes every effort to ensure the quality of the information it makes available. Notwithstanding the foregoing, Telit does not make any warranty as to the information contained herein, and does not accept any liability for any injury, loss or damage of any kind incurred by use of or reliance upon the information.
Telit disclaims any and all responsibility for the application of the devices characterized in this document, and notes that the application of the device must comply with the safety standards of the applicable country, and where applicable, with the relevant wiring rules.

Contents

1. Scope

Aim of this document is to describe suggestions and corrections that Telit advises to improve **STF Irrigation *"B101AA"* (Weather Station)** application that integrates Telit module **LE910S1-EAG**.

2. Design review

Design review is based on the following received documentation:

- Schematic files: *"EA34174 - 13.09.2022 - R2\Schematic Prints"*
- Gerber files:
- Other: *"Telit Design Review P101DA-R01 (Schematic).pdf*

Summary Tables:

Schematic Review	P	F	I	MI	N/A
Power Supply	√				
SIM Pins	√			√	
Digital Pins		√			
Audio					√
RF		√			

PCB Layout Review	P	F	I	MI	N/A
General Placement					√
Antenna Waveguide					√
RF Aspects					√
Audio Aspects					√

P: Pass; F: Fail; I: Improvements possible; MI: Missing Information; N/A: Not Applicable

The following symbols will be used throughout the Design Review to indicate:

√ OK: No design changes are required.

Tip: information or possible improvement, not mandatory but recommended.

Warning: if you don't follow the recommendation there's a risk of malfunctioning or issues during the homologation phase, strongly recommended.

Error: it's mandatory to follow the recommendation otherwise the module could be damaged or could not work properly or there's high probability of facing issues during the homologation phase.

? Missing Information: some relevant information is missing therefore the DR cannot be accurate on this item.

2.1. Schematic review

2.1.1. Power supply

- OK

2.1.2. SIM pins

- ESD protections on SIM lines are not specified. Those used in previous design were good.

2.1.3. Digital pins

- **GNSS-EN*(G14) is not actually driven.** The line "*GNSS_3V3_EN"* which control the open-collector Q7 is routed to the mPCIe connector J1(36), but mate connector J2 on mother board B905AA has this pin left unconnected. This line is not routed to the application MCU.

- **FORCED_USB_BOOT(F14) should be connected to a test-point** in order to allow recovery boot from USB. In current design FORCED_USB_BOOT is routed to the mPCIe connector J1(40), but mate connector J2 on mother board B905AA has this pin left unconnected.

- **USB is not connected.** Pins USB_D+(B15),USB_D-(C15),USB_VBUS(A13) are routed to mPCIe connector J1(11,13,9-10) but mate connector J2 on mother board has these pins left unconnected. We strongly suggest adding accessible test points or PCB connector pads at USB port: this gives convenient access for network certification testing, firmware upgrade and module debug logs. The USB connector can be "DNP" until needed.

- **GNSS_PPS(P7) must not be tied to GND**. It is an output, internally connected to N9. Leave it floating.

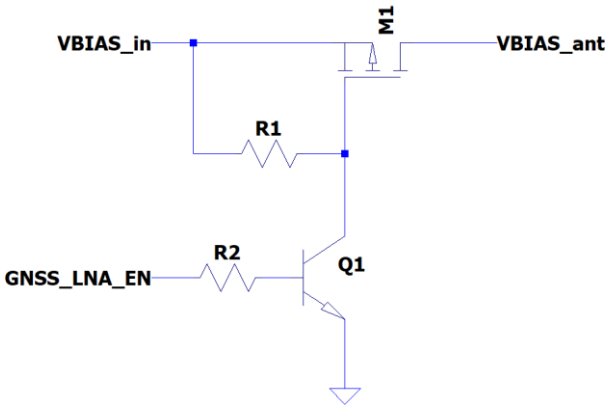
2.1.4. Audio pins

N/A

2.1.5. RF aspects

- The circuit used to feed the GNSS antenna with DC bias is inappropriate: the DC voltage applied to the antenna (Q4 emitter) is linked to the voltage on the base (less than 1.8V)

rather than VCC. Considering the voltage drop VBE the bias will be less than 1V. We suggest a solution like the one sketched in figure below.



⚠ A low-noise design of power supply is a key point for best GNSS receiver performances. For this reason we recommended using a dedicated low-noise source for the DC-bias of the antenna, or at least adding an EMI filter. Using directly VCC is not the best choice.

ℹ To avoid harmonics we suggest making provision for, on modem board and any other board connected or close to it, 33pF capacitor to GND, placed very close to component and on its same path and layer, in the following cases only if those components are not shielded:

- Power Sources and signals on I/O connectors
- On power supply input PADs
- On power supply output PADs
- Diodes in forward conduction, like LEDs, on anode and/or cathodes if not directly tied to a power net
- Transistor bases, phototransistors and opto-isolator

We suggest predisposing 33pF capacitors because in general this is a value where SMD 0603/0402, 25V/50V COG capacitors have their self-resonant frequency at about 1GHz.

2.2. PCB Layout review

2.2.1. General placement

N/A

2.2.2. RF aspects

N/A

2.2.3. Audio aspects

N/A

2.3. General comments

Review is related to received application information and the supposed use of it.

3. Quality record

This design review is registered internally in JIRA with ID#**34174**.
The customer request is registered internally in Salesforce with ID#**00348293**.