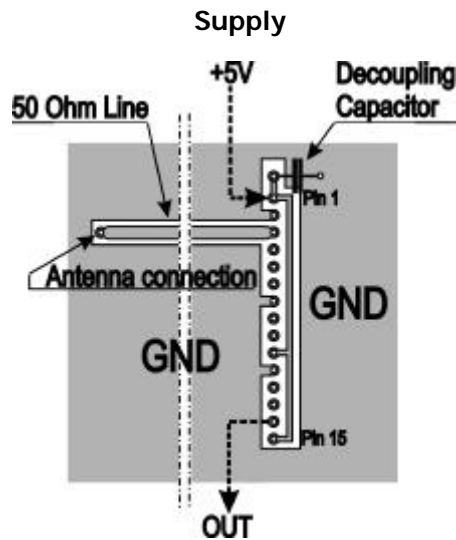


Correct use of RF receivers

Following are some design rules to be applied to PCB design to obtain the best performances and to keep Certification limits out of AUREL RF Modules.

This is the typical lay out of the solder side of a PCB suited for a receiver; same rules apply for a transmitter:



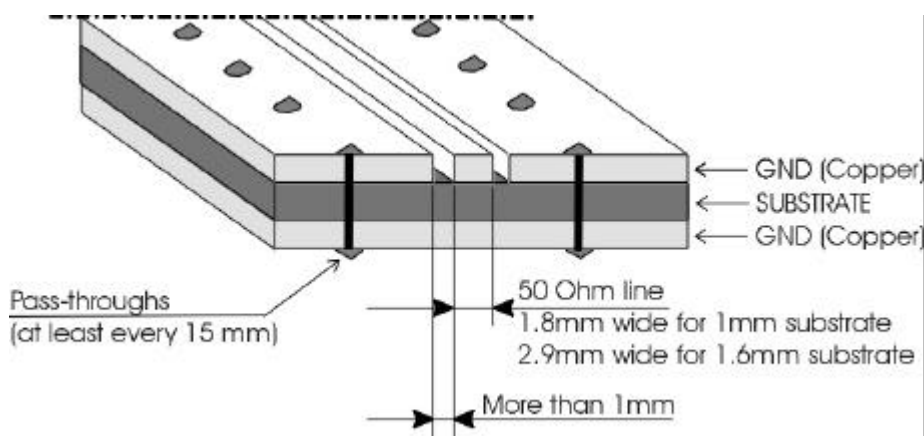
Note: example pin-out.

Supply:

1. Good filtered Dc, max. tolerance $\pm 10\%$
2. Use a decoupling ceramic capacitor (10.000 pF) directly on supply pin(s).

Ground:

1. Must cover all area around the module. Circuit should be etched on two side PCB, with sides connected every 15 mm (at least)
2. Ground must be present around the antenna output area



Le caratteristiche tecniche possono subire variazioni senza preavviso. La AUREL S.p.A non si assume la responsabilità di danni causati dall'uso improprio del dispositivo.

50 Ohm lines:

1. Should be as short as possible
2. Wide 1.8 mm for FR4 PCBs (1 mm thick) and 2.9 mm for FR6 (1.6 mm thick). Distance from surrounding GND is more than 1 mm (2 mm is better).
3. On reverse side of PCB, should have a rather large GND area

Antenna connection:

1. Can be used to directly connect a radiating stylus (165 mm for 433.92 MHz, 85 mm for 868.3 MHz, straight wire)
2. Can be used to connect the central conductor of a coaxial cable to remote antenna. The cable out braid must be connected to GND near the antenna connection.

Other components:

1. Keep Aurel module as far as possible from other circuit components (min. 5 mm)
2. Keep as far as possible microprocessor and clock circuitry. Apply GND shields
3. Do not install components around the 50 Ohm line(s). Keep at least 10 mm clearance
4. If antenna is directly connected to PCB antenna connection point (see above) keep a 50 mm radius area with no components, but adequate GND. If antenna connection point is used for coax cable connection, it is possible to move components up to 5 mm.