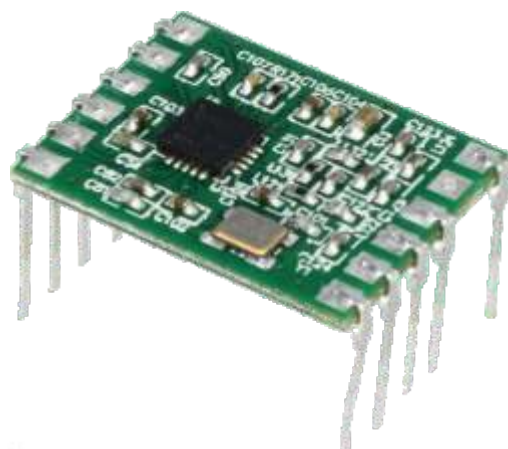


Sub 1GHz Multichannels Radio Transceiver

It is a low cost sub 1GHz multichannels transceiver designed for low-consumption wireless applications. The hardware is based on Texas Instruments CC1101 component. Programmable from external microcontroller via SPI interface.



Technical Characteristics

Characteristics	MIN	TYP	MAX	UNIT
Supply Voltage	1.8	3	3.6	VDC
Supply Current (RX mode / TX mode)		18/30		mA
Supply Corrente Power Down mode		0.2		µA
Operative Frequency		868.30		MHz
RF Power Output 50ohm		+10		dBm
RF Sensitivity (1.2 Kb/sec data rate)		- 112		dBm
Data Rate	1.2		500	Kbit/s
Operating Temperature Range	-20		+70	°C

For more information and details, please refer to the CC1101 Texas Instruments datasheet.

Applications :

- Wireless security systems
- Home and building automation
- Automatic Measure Reading
- Industrial Control and Monitoring
- Wireless Sensor Network

Feature :

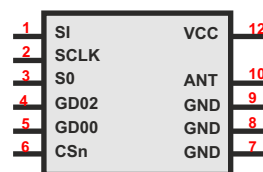
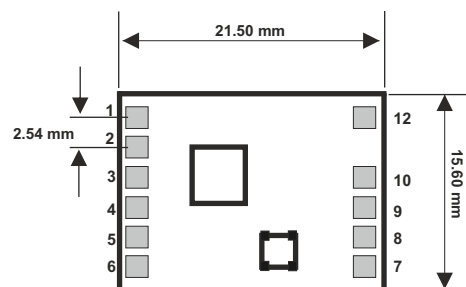
- Bidirectional multichannel Link
- Low consumption technology
- DIL Package

**ETSI EN 300 220
COMPLIANCE**

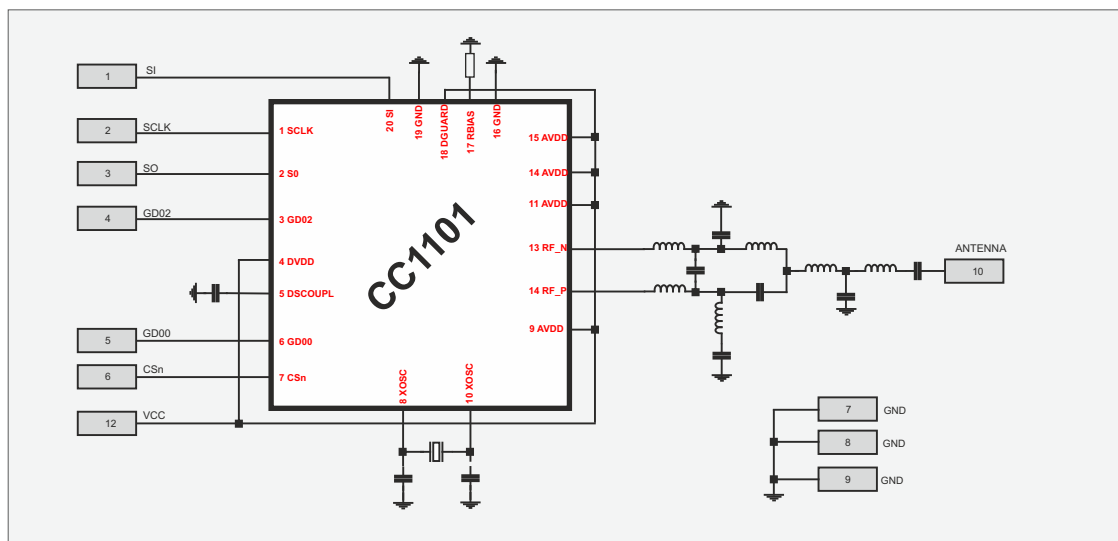
PIN OUT RC-CC1101-SPI-868

Pads	Name	Description
1	SI	Serial configuration interface SPI -----> Data Input
2	SCLK	Serial configuration interface SPI -----> Clock Input
3	S0	Serial configuration interface SPI -----> Data Output
4	GD02	Digital Output configurable
5	GD00	Digital Output configurable
6	CSn	Serial configuration interface SPI -----> Chip Select
7	GND	Ground
8	GND	Ground
9	GND	Ground
10	ANT	Antenna
11		
12	VCC	Supply Voltage

Mechanical dimensions



868MHz SCHEMATIC VERSION



Recommended Hardware design

1) Hardware

All unused pins should be left floating; do not ground.

All GND pins must be well grounded.

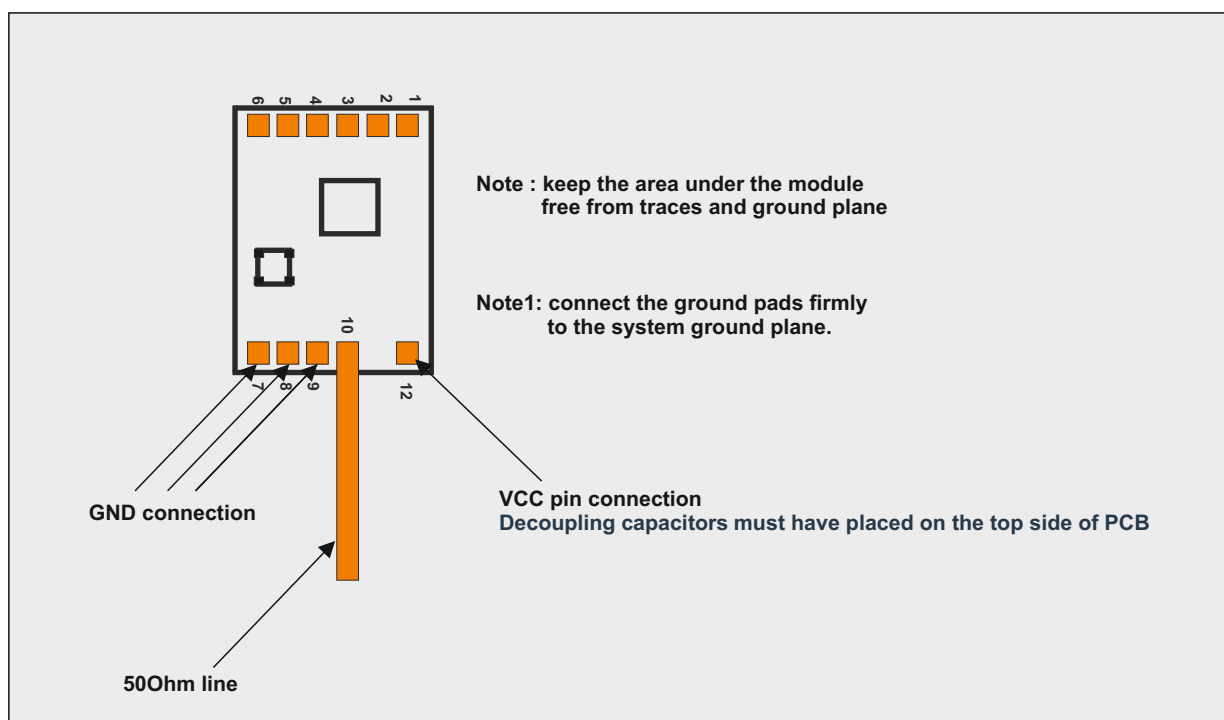
Keep the area under the module free from traces and ground plane.

The «50 Ohm line» trace, must be as short as possible.

2) Power Supply

The transceiver module must be powered from a regulated voltage.

It is recommended to keep the power supply line for VCC as short and low impedance as possible. Near the power pins it is recommended to insert a ceramic the decoupling capacitor (100nF) placed on the top side of PCB.



Ground Layer :

- Must be present around the antenna output area
- Circuit design should be on a 2-side PCB, connecting both ground planes with pass-through vias at least 15 mm each other.